

Keysight Logic and Protocol Analyzer

User Guide

Notices

© Keysight Technologies 2008-2022

No part of this manual may be reproduced in any form or by any means (including electronic storage and retrieval or translation into a foreign language) without prior agreement and written consent from Keysight Technologies as governed by United States and international copyright laws.

Trademarks

ARM® and Thumb® are registered trademarks and ARM7TDMI(TM) is a trademark of ARM Limited.

Intel® and Pentium® are U.S. registered trademarks of Intel Corporation.

Java(TM) is a U.S. trademark of Sun Microsystems, Inc.

Software Release version

Version 07.00.0000

Edition

April 2022

Available in electronic format only

Keysight Technologies
1900 Garden of the Gods Road
Colorado Springs, CO 80907 USA

Warranty

THE MATERIAL CONTAINED IN THIS DOCUMENT IS PROVIDED "AS IS," AND IS SUBJECT TO BEING CHANGED, WITHOUT NOTICE, IN FUTURE EDITIONS. FURTHER, TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, KEYSIGHT DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED WITH REGARD TO THIS MANUAL AND ANY INFORMATION CONTAINED HEREIN, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. KEYSIGHT SHALL NOT BE LIABLE FOR ERRORS OR FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES IN CONNECTION WITH THE FURNISHING, USE, OR PERFORMANCE OF THIS DOCUMENT OR ANY INFORMATION CONTAINED HEREIN. SHOULD KEYSIGHT AND THE USER HAVE A SEPARATE WRITTEN AGREEMENT WITH WARRANTY TERMS COVERING THE MATERIAL IN THIS DOCUMENT THAT CONFLICT

WITH THESE TERMS, THE WARRANTY TERMS IN THE SEPARATE AGREEMENT WILL CONTROL

Technology Licenses

The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license.

U.S. Government Rights

The Software is "commercial computer software," as defined by Federal Acquisition Regulation ("FAR") 2.101. Pursuant to FAR 12.212 and 27.405-3 and Department of Defense FAR Supplement ("DFARS") 227.7202, the U.S. government acquires commercial computer software under the same terms by which the software is customarily provided to the public. Accordingly, Keysight provides the Software to U.S. government customers under its standard commercial license, which is embodied in its End User License Agreement (EULA), a copy of which can be found at <http://www.keysight.com/find/sweula>. The license set forth in the EULA represents the exclusive authority by which the U.S. government may use, modify, distribute, or disclose the Software. The EULA and the license set forth therein, does not require or permit, among other things, that Keysight: (1) Furnish technical information related to commercial computer software or commercial computer software documentation that is not customarily provided to the public; or (2) Relinquish to, or otherwise provide, the government rights in excess of these rights customarily provided to the public to use, modify, reproduce, release, perform, display, or disclose commercial computer software or commercial computer software documentation. No additional government requirements beyond those set forth in the EULA shall apply, except to the extent that those terms, rights, or licenses are explicitly required from all providers of commercial computer software pursuant to the FAR and the DFARS and are set forth specifically in writing elsewhere in the EULA. Keysight shall be under no obligation to update, revise or otherwise modify the Software. With respect to any technical data as defined by FAR 2.101, pursuant to FAR 12.211 and 27.404.2 and DFARS 227.7102, the U.S. government acquires no

greater than Limited Rights as defined in FAR 27.401 or DFAR 227.7103-5 (c), as applicable in any technical data.

Safety Notices

CAUTION

A CAUTION notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in damage to the product or loss of important data. Do not proceed beyond a CAUTION notice until the indicated conditions are fully understood and met.

WARNING

A WARNING notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in personal injury or death. Do not proceed beyond a WARNING notice until the indicated conditions are fully understood and met.

Safety Summary

If the instrument is used in a manner not specified by the manufacturer, the protection provided by the instrument may be impaired. Use the power cord included with the instrument. See *For Your Safety booklet* publication number 9320-6797.

The following general safety precautions must be observed during all phases of operation of this instrument. Failure to comply with these precautions or with specific warnings or operating instructions in the product manuals violates safety standards of design, manufacture, and intended use of the instrument. Keysight Technologies assumes no liability for the customer's failure to comply with these requirements. Some product manuals may be provided on your instrument at **C:\Program Files\Keysight Technologies\Logic Analyzer\help\pdfs**. Printed manuals are an option for many products. Manuals may also be available on the Web. Go to www.keysight.com and type in your product number in the Search field at the top of the page.

WARNING

General

Do not use this product in any manner not specified by the manufacturer. The protective features of this product may be impaired if it is used in a manner not specified in the operation instructions.

WARNING

Before Applying Power

Verify that all safety precautions are taken. Make all connections to the unit before applying power. Note the instrument's external markings described in "Safety Symbols".

WARNING

Ground the Instrument

If your product is provided with a grounding type power plug, the instrument chassis and cover must be connected to an electrical ground to minimize shock hazard. The ground pin must be firmly connected to an electrical ground (safety ground) terminal at the power outlet. Any interruption of the protective (grounding) conductor or disconnection of the protective earth terminal will cause a potential shock hazard that could result in personal injury.

Install the instrument so that the detachable power cord is readily identifiable and is easily reached by the operator. The detachable power cord is the instrument disconnecting device. It disconnects the mains circuits from the mains supply before other parts of the instrument. The front panel switch is only a standby switch and is not a LINE switch.

WARNING

The instrument is for use only on circuits that are NOT directly connected to mains. It is NOT intended for measurements on CAT II, CAT III, or CAT IV circuits.

WARNING**Fuses**

See the service manual for information about line-fuse replacement. Some instruments contain an internal fuse, which is not user accessible.

WARNING

Do not operate the instrument in an explosive environment or in the presence of flammable gases or fumes.

WARNING

Only qualified, service-trained personnel who are aware of the hazards involved should remove instrument covers. Always disconnect the power cable and any external circuits before removing the instrument cover.

CAUTION

Clean the outside of the instrument with a soft, lint-free cloth dampened with water. Do not use too much liquid or any chemicals or detergent.

WARNING




Do not install substitute parts or perform any unauthorized modification to the product. Return the product to an Keysight Sales and Service Office for service and repair to ensure that safety features are maintained.















WARNING






Instruments that appear damaged or defective should be made inoperative and secured against unintended operation until they can be repaired by qualified service personnel.

Safety Symbols

Table 1 Safety Symbol

Symbol	Description
	Direct current
	Alternating current
	Both direct and alternating current

Symbol	Description
	Three phase alternating current
	Earth ground terminal
	Protective earth ground terminal
	Frame or chassis ground terminal
	Terminal is at earth potential
	Equipotentiality
N	Neutral conductor on permanently installed equipment
L	Line conductor on permanently installed equipment
	On (mains supply)
	Off (mains supply)
	Standby (mains supply). The instrument is not completely disconnected from the mains supply when the power switch is in the standby position
	In position of a bi-stable push switch
	Out position of a bi-stable push switch
	Equipment protected throughout by DOUBLE INSULATION or REINFORCED INSULATION
	Caution, refer to accompanying documentation
	Caution, risk of electric shock



Symbol	Description
	Do not apply around or remove from HAZARDOUS LIVE conductors
	Application around and removal from HAZARDOUS LIVE conductors is permitted
	Caution, hot surface
	Ionizing radiation
	Indicates that antistatic precautions should be taken
CAT II	Measurement Category II
CAT III	Measurement Category III
CAT IV	Measurement Category IV

Compliance and Environmental Information

Table 2 Compliance Information

Safety Symbol	Description
	CSA is the Canadian certification mark to demonstrate compliance with the Safety requirements.
	The C-tick mark is a registered trademark of the Spectrum Management Agency of Australia. This signifies compliance with the Australia EMC Framework regulations under the terms of the Radio Communication Act of 1992.
	CE compliance marking to the EU Safety and EMC Directives. ISM GRP-1A classification according to the international EMC standard. ICES/NMB-001 compliance marking to the Canadian EMC standard.
	KC certification mark to demonstrate compliance with the South Korean EMC requirements. South Korean Class A EMC declaration This equipment is Class A suitable for professional use and is for use in electromagnetic environments outside of the home.
ISM	This is the symbol for an industrial, Scientific, and Medical Group 1 Class A product
ICES/NMB-001	ICES/NMB-001 indicates that this ISM device complies with the Canadian ICES-001 Cet appareil ISM est conforme a la norme NMB du Canada.
	Product With Toxic Substance 40 yr EPUP
	This mark denotes compliance with the essential requirements of the following applicable UK regulations: <ul style="list-style-type: none"> • Electromagnetic Compatibility Regulations 2016 No. 1091 (as amended) • Electrical Equipment (Safety) Regulations 2016 No. 1101 (as amended) • The Restriction of the Use of Certain Hazardous Substances in Electrical & Electronic Equipment Regulations 2012 No. 3032 (as amended)

Table 3 Environmental Information

Safety Symbol	Description
	The crossed out wheeled bin symbol indicates that separate collection for waste electric and electronic equipment (WEEE) is required, as obligated by DIRECTIVE 2012/19/EU and other National legislation. Please refer to about.keysight.com/en/companyinfo/environment/takeback.shtml to understand your Trade in options with Keysight in addition to product takeback instructions.
	Packaging is recyclable . Please recycle in a responsible way.

Using the Keysight Logic and Protocol Analyzer

Instructions for Use:

“This product has been designed and tested in accordance with accepted industry standards, and has been supplied in a safe condition. The documentation contains information and warnings that must be followed by the user to ensure safe operation and to maintain the product in a safe condition.”

The *Keysight Logic and Protocol Analyzer* application is used with:





- modules in Keysight AXIe chassis (for example the U4154A/B Logic Analyzer module in M9502A portable 2-slot chassis)
- modules in Keysight Digital Test Console chassis (for example the U4002A portable 2-slot chassis)
- 16850-series logic analyzers.
- It can also be used by itself on a Windows Vista/7/8/8.1 or Windows Server 2008/2012 computer for *remote access* (see [page 59](#)) of logic analysis systems on the network, or for *offline analysis* (see [page 217](#)) of captured data.

This user guide provides the following information.

- **What's New** (see [page 25](#))
- **Getting Started** (see [page 39](#))
- **Probing the Device Under Test** (see [page 57](#))
- **Connecting to a Logic Analysis System** (see [page 59](#))
- **Setting Up the Logic Analyzer** (see [page 71](#))
 - Configuring Logic Analyzer Modules (see [page 72](#))
 - "Setting Up Probes" (in the online help)
 - Setting the Logic Analyzer Threshold Voltage (see [page 77](#))
 - Defining Buses and Signals (see [page 83](#))
 - Choosing the Sampling Mode (see [page 96](#))
 - Setting Up Symbols (see [page 132](#))
 - Installing Licensed Hardware Upgrades (see [page 139](#))
- **Capturing Data from the Device Under Test** (see [page 141](#))
 - Setting Up Quick (Draw Box) Triggers (see [page 143](#))
 - Specifying Simple Triggers (see [page 146](#))
 - Specifying Advanced Triggers (see [page 152](#))
 - Triggering From, and Sending Triggers To, Other Modules/Instruments (see [page 186](#))
 - Storing and Recalling Triggers (see [page 194](#))
 - Running/Stopping Measurements (see [page 196](#))
 - Saving Captured Data (and Logic Analyzer Setups) (see [page 198](#))
 - Extending Capture Capability with COM/DCOM (see [page 206](#))
- **Analyzing the Captured Data** (see [page 207](#))
 - Offline Analysis (see [page 217](#)) (after Loading Saved Data and Setups (see [page 208](#)))
 - Analyzing Waveform Data (see [page 220](#))
 - Analyzing Listing Data (see [page 235](#))
 - Displaying Names (Symbols) for Bus/Signal Values (see [page 243](#))
 - Marking, and Measuring Between, Data Points (see [page 245](#))
 - Searching the Captured Data (see [page 264](#))
 - Comparing Captured Data to Reference Data (see [page 274](#))

- Viewing Source Code Associated with Captured Data (see [page 277](#))
- Analyzing Packet Data (see [page 283](#))
- Analyzing the Same Data in Different Ways (Using the Window) (see [page 307](#))
- Setting the System Trigger and Skew Between Modules (see [page 312](#))
- Using Display Windows (see [page 313](#))
- Printing Captured Data (see [page 314](#))
- "Using Tools" (in the online help)
- "External Oscilloscope Time Correlation and Data Display" (in the online help)
- "Using the PCIe Gen3 Analyzer"
- Managing Software Licenses (see [page 321](#))
- Updating Software (see [page 327](#))
- Solving Problems (see [page 331](#))
- Concepts (see [page 353](#))
- Reference (see [page 385](#))
- Glossary (see [page 613](#))
- "COM Automation" (in the online help)
- "XML Format" (in the online help)

See Also

-  *"AXIe based Logic and Protocol Analysis – Installation Guide"*
-  *"AXIe based Logic and Protocol Analysis – Quick Start Guide"*
-  *"16850 Series Portable Logic Analyzers Installation/Quick Start Guide"*
-  *"16860 Series Portable Logic Analyzers Installation/Quick Start Guide"*

Contents

Safety Summary	3
Compliance and Environmental Information	7
Using the Keysight Logic and Protocol Analyzer	8

1 What's New

Version 06.71 What's New	27
Version 06.70 What's New	28
Version 06.60 What's New	29
Version 06.50 What's New	30
Version 06.40 What's New	31
Version 06.30 What's New	32
Version 06.20 What's New	33
Version 06.10 What's New	35
Version 06.00 What's New	38

2 Getting Started

Tutorial - Getting to know your logic analyzer	40
Turning on the logic analyzer	40
Connecting to the device under test	41
Setting up bus/signal names	42
Setting the acquisition mode	43
Setting up a simple trigger	43
Open the tutorial configuration file	44
Using markers	44
Zooming in on the data	45
Loading and saving configuration files	46
Saving and recalling trigger setups	46
Quick marker measurements	47
Searching data	48
Toolbars, tool tips, and mouse shortcuts	49

Measurement Examples	50
Making a timing analyzer measurement	50
Making a state analyzer measurement	51
To trigger on one of multiple edges	52
To trigger on ranges	53
Demo Center	55
3 Probing the Device Under Test	
To control signals in the device under test	58
4 Connecting to a Logic Analysis System	
Connecting to the U4154A/B Logic Analyzer Module	59
To add a logic analysis system to the list	61
To delete a logic analysis system from the list	62
To refresh the logic analysis system list	63
To view logic analysis system details	64
To enter your "System In Use" comments	65
To select a logic analysis system for auto-connect	66
To chat with another logic analysis system user	67
Setting Up Passwords for Remote Access	68
5 Setting Up the Logic Analyzer	
Configuring Logic Analyzer Modules	72
To disable and enable modules	72
Creating a Logic Analyzer Multi-card Set	74
Repartitioning a U4164A Multi-card Set	75
Setting the Logic Analyzer Threshold Voltage	77

Defining Buses and Signals	83
To add a new bus or signal	84
To delete a bus or signal	84
To rename a bus or signal	85
To rename the bits of a bus	86
To assign channels in the default bit order	88
To assign channels, selecting the bit order	89
To use clock channels as extra data channels	89
To define buses and signals by importing netlist files	90
To reorder bits by editing the Channels Assigned string	91
To set the default number base	93
To set polarity	94
To add user comments	94
To add a folder	95
To alias a bus/signal name	95
To sort bus/signal names	95
Choosing the Sampling Mode	96
Selecting the Timing Mode (Asynchronous Sampling)	96
Selecting the State Mode (Synchronous Sampling)	98
In Either Timing Mode or State Mode	128
Using Timing Zoom	129
Setting Up Symbols	132
To create and edit user-defined symbols	132
To load symbols from a file	134
To run the symbol reader outside the application	135
To create an ASCII symbol file	135
To change symbol reader options	136
To enter symbolic bus/signal values	138
Installing Licensed Hardware Upgrades	139

6 Capturing Data from the Device Under Test

Setting Up Quick (Draw Box) Triggers	143
To set a Quick Trigger in the Waveform window	143
To set a Quick Trigger in the Listing window	144
To set a Quick Trigger in the Source window	145
Specifying Simple Triggers	146
To specify bus patterns in a simple trigger	147
To specify signal levels in a simple trigger	148
To set a bus/signal edge in a simple trigger	148

Specifying Advanced Triggers	152
Reading Event and Action Statements	153
Using Timers	153
Using Counters	154
Using Flags	157
To replace or insert trigger functions into trigger sequence steps	158
To specify bus/signal patterns	159
To set a bus/signal edge in an advanced trigger	160
To specify packet events (in "Find a packet" trigger function)	164
To specify a trigger sequence step's goto or trigger action	166
To specify default storage	167
To insert or delete events	168
Using Burst Patterns	170
To negate events	177
To change the evaluation order of AND/OR'ed events	177
To choose between a duration or occurrence count for events (timing mode)	178
To insert or delete actions (in a trigger sequence step)	179
To cut, copy, and paste trigger sequence steps	182
To delete trigger sequence steps	182
To show a trigger sequence step as Advanced If/Then trigger functions	182
To convert a trigger sequence step to Advanced If/Then trigger functions	183
To display or hide "If" clause comments	183
To clear the trigger sequence	185
Triggering From, and Sending Triggers To, Other Modules/Instruments	186
To arm one module with another module's trigger	186
To trigger other instruments - Trigger Out	191
To trigger analyzer from another instrument - Trigger In	192
Storing and Recalling Triggers	194
To store a trigger	194
To recall a trigger	194
To set the trigger history depth	195
Running/Stopping Measurements	196
To change the "Go to Trigger on Run" option	196
Saving Captured Data (and Logic Analyzer Setups)	198
To save a configuration file	198
To export data to standard CSV format files	199
To export data to module CSV format files	201
To export data to module binary (ALB) format files	203
Extending Capture Capability with COM/DCOM	206

7 Analyzing the Captured Data

Loading Saved Data and Setups	208
To open a configuration file	208
To recall a recently used configuration file	210
To transfer module setups to/from multi-module systems	210
Using Data Import Modules	211
Offline Analysis	217
Offline Analysis on Logic Analyzers	217
Offline Analysis on Personal Computers	218
Offline File Formats	218
Analyzing Waveform Data	220
To change the display scale (time/division)	221
To go to different locations in the captured data	222
To re-arrange waveforms	223
To overlay waveforms	223
To find a bus/signal row	224
To view bus data as a chart	224
To show/hide parts of the waveform display	226
To insert or delete buses/signals	227
To group signals into a bus	227
To expand/collapse buses	227
To insert separator rows	227
Changing Waveform Window Properties	227
Changing Bus/Signal Row Properties	230
Changing Analog Signal Row Properties	233
Analyzing Listing Data	235
To go to different locations in the captured data	236
To re-arrange bus/signal columns	237
To find a bus/signal column	237
To show/hide parts of the listing display	237
To insert or delete buses/signals	238
Changing Listing Window Properties	238
Changing Bus/Signal Column Properties	240
Displaying Names (Symbols) for Bus/Signal Values	243

Marking, and Measuring Between, Data Points	245
To read the markers display and overview bars	245
To create new markers	246
To place markers in data	247
To go to a marker	249
To center the display about a marker pair	249
To change a marker's snap to edge setting	250
To delete a marker	250
To create a new time interval measurement	251
To create a new sample interval measurement	252
To create a new value at measurement	253
To rename a marker	254
To send a marker to the back	254
Changing Marker Properties	255
Using Y-Axis Markers for Analog Signals (in the Waveform Display)	257
Searching the Captured Data	264
To quickly find bus/signal patterns	264
To find bus/signal patterns in the captured data	265
To find packet patterns in the captured data	267
To find complex patterns in the captured data	270
To store, recall, or delete favorite find patterns	271
To specify "found" marker placement	272
Comparing Captured Data to Reference Data	274
To copy data to the reference buffer	274
To find differences in the compared data	274
To compare only a range of samples	275
To offset the reference data	275
To run until a number of compare differences	275
To set Compare window properties	276
Viewing Source Code Associated with Captured Data	277
To step through captured data by source lines	277
To go to captured data associated with a source line	278
To browse source files	278
To search for text in source files	279
To go to the source line associated with the listing center rectangle	279
To edit the source code directory list	279
To select the correlation bus	280
Changing Source Window Properties	281

Analyzing Packet Data	283
Viewing the Packet Summary	285
Viewing a Selected Packet	293
Changing Packet Summary Event Colors	301
Changing Protocol Viewer Window Properties	302
Changing Packet Summary Column Properties	304
Analyzing the Same Data in Different Ways (Using the Overview Window)	307
To open or display the Overview window	307
To add, duplicate, or delete windows and tools	307
To edit window or tool properties	310
To rename windows, tools, and modules	311
To redraw the Overview window	311
To delete the Overview window	311
Setting the System Trigger and Skew Between Modules	312
Using Display Windows	313
To add or delete display windows	313
To turn window tabs on/off	313
Printing Captured Data	314
To print captured data	314
To copy text to the clip board	314
To copy a screen to the clip board	315
To install a printer	315
To connect a LAN	315
8 B4661A Memory Analysis Software Tools	
Tools installed with the B4661A Memory Analysis Software	318
9 Managing Software Licenses	
To view active software license information	322
To activate software licenses	323
To access floating license servers	324
To borrow floating licenses and return them early	326
10 Updating Software	
11 Solving Problems	
If starting in offline mode is unexpected	332
If an ALA format configuration file won't open	333

Interpreting Error Messages	334
Error Messages	334
Warning Messages	339
Informational Messages	340
License Problems	342
License Not Available	342
Floating License Server Communication Timeout	342
Running Self Tests	343
Accessing Japanese Online Help	345
Network Troubleshooting Guide	348
Remote Desktop Set Up	350
For More Information	351

12 Concepts

When should you use an oscilloscope?	354
When should you use a logic analyzer?	355
What is a logic analyzer?	356
Sampling clock in the timing analyzer	356
Sampling in the timing acquisition mode	356
Triggering the timing analyzer	357
Sampling clock in the state analyzer	358
Sampling in the state acquisition mode	359
Triggering the state analyzer	359
Pod and Channel Naming Conventions	361
Why Are Pods Missing?	362
Memory Depth and Channel Count Trade-offs	363
Transitional Timing	365
Understanding State Mode Sampling Positions	367

Understanding Logic Analyzer Triggering	369
The Conveyor Belt Analogy	369
Summary of Triggering Capabilities	370
Sequence Steps	371
Boolean Expressions	372
Branches	373
Edges	373
Ranges	373
Flags	373
Counters	373
Timers	374
Storage Qualification	375
Strategies for Setting Up Triggers	376
Conclusions	379
ALA vs. XML, When to Use Each Format	380
Multiframe Logic Analysis Systems	381

13 Reference

Menus	386
File Menu	386
Edit Menu	387
View Menu	388
Setup Menu	389
Tools Menu	390
Markers Menu	391
Run/Stop Menu	392
Overview Menu	393
Listing Menu	393
Waveform Menu	394
Compare Menu	394
Source Menu	395
Protocol Viewer Menu	395
Window Menu	396
Help Menu	397

Toolbars	398
Standard Toolbar	398
Analyzer Setup Toolbar	399
External Oscilloscope Setup Toolbar	399
Data Import Toolbar	400
Probes Toolbar	400
Markers Toolbar	400
Run/Stop Toolbar	401
Visual Basic Toolbar	401
To customize toolbars	402
Marker Measurement Display Bar	403
Windows	404
Waveform Display Window	404
Listing Display Window	409
Compare Display Window	411
Source Display Window	412
Protocol Viewer Display Window	413
ONFi Analysis Window	416
Overview Window	418

Dialogs	421
Advanced Clocking Setup Dialog	422
Advanced Trigger Dialog	424
Keysight Logic Analyzer Upgrade Dialog	425
Analyzer Setup Dialog	426
Chat Dialog	429
Chat Select Destination Dialog	430
Choose a Protocol Family and Bus Dialog	430
Create a New Configuration Dialog	431
DDR Accumulate Dialog	431
E-mail Dialog	432
Event Editor Dialog	433
Export Dialog	434
Export File Selection Dialog	435
External Application Setup Dialog	436
External Trigger Dialog	437
Find Dialog	438
Frame/Module Information Dialog	439
Instrument Details Dialog	441
Import Dialog	442
Import Setup Dialog	442
Module Mapping Dialog	444
Module Skew and System Trigger Dialog	446
Netlist Import Dialog	446
Offline Startup Options Dialog	447
Options Dialog	447
Pod Assignment Dialog	450
Printing Data Dialog	451
Properties Dialog	452
Range Properties Dialog	452
Recall Trigger Dialog	453
Run Properties Dialog	454
Search Dialog	455
Select Symbol Dialog	458
Select System to Use Dialog	459
Software Licensing Dialog	461
Source Viewer Properties Dialog	466
Specify Mapping Dialog	468
Status Dialog	469
Symbols Dialog	471
System Summary Dialog	472
Target Control Port Dialog	475
TimingZoom Setup Dialog	476

Trigger Functions	478
Timing Mode Trigger Functions	478
State Mode Trigger Functions	490
Data Formats	504
ALA Format	504
Standard CSV Format	504
Module CSV Format	504
Module Binary (ALB) Format	511
Object File Formats Supported by the Symbol Reader	513
General-Purpose ASCII (GPA) Symbol File Format	514
General-Purpose ASCII (GPA) Record Format Summary	514
SECTIONS	515
FUNCTIONS	516
USER	516
VARIABLES	518
SOURCE LINES	518
START ADDRESS	519
Comments	519
Product Overviews	520
U4164A Logic Analyzer Product Overview	520
U4154A Logic Analyzer Product Overview	522
U4154B Logic Analyzer Product Overview	523
16850-Series Logic Analyzer Product Overview	524
16860-Series Logic Analyzer Product Overview	528
Keysight Logic and Protocol Analyzer Application Product Overview	532
Logic Analyzer Notes	536
U4164A Logic Analyzer Notes	536
U4154A Logic Analyzer Notes	544
U4154B Logic Analyzer Notes	549
16850-Series Logic Analyzer Notes	553
16860-Series Logic Analyzer Notes	555
Pod and Channel Naming Conventions in Logic Analyzer	561
Changing Advanced Probe Settings for Logic Analyzers	563
Setting Clock Hysteresis for the U4164A State Sampling Clock	566
Setting up and Running Eyescans in Logic Analyzers	567

Specifications and Characteristics	587
U4164A Logic Analyzer Specifications and Characteristics	587
U4154A Logic Analyzer Specifications and Characteristics	592
U4154B Logic Analyzer Specifications and Characteristics	596
16850-Series Logic Analyzer Specifications and Characteristics	600
16860-series Logic Analyzer Specifications and Characteristics	604
What is a Specification?	611
What is a Characteristic?	611

14 Glossary

Index

1 What's New

In this release, version 07.00, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **New DDR5 and LPDDR5 decoders** - Two new tools, DDR5 Decoder and LPDDR5 Decoder, have been introduced to allow you to decode and view transactions and commands from a DDR5 and LPDDR5 SDRAM memory bus respectively. These decoders work with the Keysight Logic and Protocol Analyzer software application (in online / offline mode) to decode the acquired DDR5/LPDDR5 traces. The decoded data is displayed in a Listing window.
To know more, refer to the online help installed with these decoders.
- **Refresh Rate Overview tab in the Memory Analysis window** - This tab has been redesigned to show graphs for:
 - the number of Refresh commands found in each sampled Refresh Window.
 - the exact locations of Refresh commands and Self Refresh entry/exit periods in each sampled Refresh Window.To know more, refer to the help book Analyzing Refresh Performance of an SDRAM.
- **DDR/LPDDR Post Process Compliance Tool** - You can now use this tool to configure one or more triggers so that Logic Analyzer can acquire relevant traces with event(s) of interest for running compliance tests.
To know more, refer to the topic Configuring Trigger(s) to Acquire Data Relevant for Compliance Tests in the tool's online help.
- **Markers with transaction names and color-coding in the Memory Analysis and ONFi Analysis viewers** - You can now place a marker with the name and color-coding of the decoded memory transaction on which it is placed.
To know more, refer to the topic Placing Markers on Decoded Transactions.
- **ONFi Decode Customization XML file** - The following changes/additions have been made to this XML file.
 - Ability to customize the address length for the following commands
 - Change Read Column Enhanced
 - Read
 - Page Program
 - Block Erase
 - 6-byte addresses allowed for address lengthTo know more, refer to the topic ONFi Decode Customization XML File Structure and Elements.
- **Count of pass events in the Real Time Compliance Tool** - The tool now displays the following additional information in test results for DDR4, DDR5, and LPDDR5 tests.
 - With each violation, the tool lists the count of compliance limit pass events found before encountering the violation.
 - In case the tool does not encounter any compliance limit violation for a test, it reports the total number of pass events found for the test.To know more, refer to the topic Viewing Results in the online help installed with the tool.

See Also • ["Version 06.71 What's New"](#) on page 27

- ["Version 06.70 What's New"](#) on page 28
- ["Version 06.60 What's New"](#) on page 29
- ["Version 06.50 What's New"](#) on page 30
- ["Version 06.40 What's New"](#) on page 31
- ["Version 06.30 What's New"](#) on page 32
- ["Version 06.20 What's New"](#) on page 33
- ["Version 06.10 What's New"](#) on page 35
- ["Version 06.00 What's New"](#) on page 38

Version 06.71 What's New

In this release, version 06.71, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **New Filter feature in Memory Analysis window and ONFi Analysis window** - You can filter the display of transactions in the Memory Analysis and ONFi Analysis windows as per the filter criteria defined by you.
To know more, refer to the topics **Filtering Decoded Memory Transactions and Configure Transaction Filter Dialog** in the Memory Analysis Window User Guide and ONFi Analysis Window User Guide).
- Support for Infiniium MXR-series and EXR-series oscilloscopes has been added in the ViewScope feature.
To know more, refer to the *External Oscilloscope Time Correlation and Data Display Online Help*.

- See Also
- ["Version 06.70 What's New"](#) on page 28
 - ["Version 06.60 What's New"](#) on page 29
 - ["Version 06.50 What's New"](#) on page 30
 - ["Version 06.40 What's New"](#) on page 31
 - ["Version 06.30 What's New"](#) on page 32
 - ["Version 06.20 What's New"](#) on page 33
 - ["Version 06.10 What's New"](#) on page 35
 - ["Version 06.00 What's New"](#) on page 38

Version 06.70 What's New

In this release, version 06.70, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **Support for GDDR6 decode in Memory Analysis window** - This support is available only for GDDR6 command/address information capture and analysis. The GDDR6 data (DQ) capture and analysis is not supported.

- See Also
- ["Version 06.60 What's New"](#) on page 29
 - ["Version 06.50 What's New"](#) on page 30
 - ["Version 06.40 What's New"](#) on page 31
 - ["Version 06.30 What's New"](#) on page 32
 - ["Version 06.20 What's New"](#) on page 33
 - ["Version 06.10 What's New"](#) on page 35
 - ["Version 06.00 What's New"](#) on page 38

Version 06.60 What's New

In this release, version 06.60, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **Support for Multiple System Clock Speeds in compliance and decode** - This new feature is available only for LPDDR5 tests.

The DDR/LPDDR Post Process Compliance Tool and the Memory Analysis window can now handle the system clock speed changes of an LPDDR5 bus running at multiple clock speeds. For an LPDDR5 bus running at multiple clock speeds, you can specify a set of bus settings for each clock speed to allow these tools to perform correct decode and compliance for changing clock speeds.

Refer to the topic **Configuring the Setup to Handle Multiple System Clock Speeds** in the Memory Analysis window online help to know more about this feature.

Refer to the topic **Customizing the Compliance Setup to Handle Multiple System Clock Speeds** in the DDR/LPDDR Post Process Compliance Tool Online Help to know more about this feature.

- **New features in the ONFi Analysis window** - The ONFi Analysis window has been enhanced to:
 - support the indexing of the commands in the traffic overview by logical units (LUNs).
 - support the decode of new training related commands in ONFi 4.1.

Refer to the topic **Customizing ONFi Decode** in the ONFi Analysis Window User Guide to know more.

- See Also
- ["Version 06.50 What's New"](#) on page 30
 - ["Version 06.40 What's New"](#) on page 31
 - ["Version 06.30 What's New"](#) on page 32
 - ["Version 06.20 What's New"](#) on page 33
 - ["Version 06.10 What's New"](#) on page 35
 - ["Version 06.00 What's New"](#) on page 38

Version 06.50 What's New

In this release, version 06.50, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **Setting compliance tests limits from system parameters** - You can now set compliance tests limits automatically based on the specific characteristics that you specified for your system in the Real Time Compliance Tool and the DDR/LPDDR Post Process Compliance Tool. This new feature is available only for DDR4, DDR5, and LPDDR5 tests.
Refer to the topic **Configuring Tests Limits** in the DDR/LPDDR Post Process Compliance Tool Online Help or the topic **To set or edit specification parameters** in the Real Time Compliance Tool Online Help to know more about this feature.
- **Decoded memory transactions export to a .csv file** - You can now export the decoded memory transactions data displayed in the upper pane of the Memory Analysis window to a specified .csv file. Refer to the topic **Exporting Decoded Memory Transactions** to know more.
- **DDR5, LPDDR5, and LPDDR4x support** - The DDR5, LPDDR5, and LPDDR4x support has been added to the following memory analysis and compliance tools of the Logic and Protocol Analyzer.
 - Memory Analysis Viewer
 - DDR/LPDDR Post-Process Compliance Tool
 - Real Time Compliance Tool
 - DDR Setup Assistant
 - DDR Configuration Creator
 - DDR Eyescan
- **Improvements to the Mode Registers Overview tab in DDR/LPDDR Memory Analysis window** - The features available in the Mode Registers Overview tab have been improved to make this tab more user-friendly and easy to use. Refer to the topic **Analyzing Mode Registers Values** to know more.
- **Updates to the "send e-mail" feature to use SMTP mail server** - The "send e-mail" feature available in multiple dialogs of the Logic and Protocol Analyzer application has been updated to allow the application to communicate directly with the specified SMTP mail server to send the configured email message to the specified recipient(s). Refer to the topic **Options Dialog > Email Settings options** to know more.
- **Improvements to the ONFi Analysis window** - The features available in the ONFi Analysis window have been enhanced to:
 - include additional ONFi decode customization choices in the XML customization file.
 - support multiple ONFi viewers on a single Logic Analyzer module.
 - support the SDR decode mode.
 - allow the export of the payload data from an ONFi trace to a separate binary file.

Refer to the topic **Analyzing ONFi Data using the ONFi Analysis Window** in the ONFi Analysis Window User Guide to know more.

- See Also
- ["Version 06.40 What's New"](#) on page 31
 - ["Version 06.30 What's New"](#) on page 32
 - ["Version 06.20 What's New"](#) on page 33
 - ["Version 06.10 What's New"](#) on page 35
 - ["Version 06.00 What's New"](#) on page 38

Version 06.40 What's New

In this release, version 06.40, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **ONFi decode customization XML file** - You can now customize how the ONFi data is decoded for display in the ONFi Analysis window by specifying the available customization options in an xml data file.
Refer to the topic Customizing ONFi Decode in the ONFi Analysis Window User Guide to know more.
- **ONFi data export to a .csv file** - You can now export either the raw ONFi data or the decoded ONFi transactions data displayed in the ONFi Analysis window to a specified .csv file.
Refer to the topic Exporting ONFi Data to a csv File in the ONFi Analysis Window User Guide to know more.
- **New Timeline view added to the ONFi Analysis window** - You can now get a visual understanding of the timeline for the ONFi transactions for a NAND target. The Timeline view displays the ONFi transactions as linear bars on a timeline.
Refer to the topic Viewing ONFi Data in a Timeline View in the ONFi Analysis Window User Guide to know more.

- See Also
- ["Version 06.30 What's New"](#) on page 32
 - ["Version 06.20 What's New"](#) on page 33
 - ["Version 06.10 What's New"](#) on page 35
 - ["Version 06.00 What's New"](#) on page 38

Version 06.30 What's New

In this release, version 06.30, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **16860-series Logic Analyzers** - The Keysight Logic and Protocol Analyzer application now adds support for the new 16860-series standalone benchtop state and timing logic analyzers. This series of logic analyzers provide 34 to 136 logic acquisition channels, depending on the model. This series can be used as a medium to high performance general purpose logic analyzer as well as for debug, validation and analysis of DDR and LPDDR memory systems.
Refer to the topic "[16860-Series Logic Analyzer Notes](#)" on page 555 to know more about this series.
- **New Multiple clocks state sampling option** - You can now run your logic analyzer in the Single clock or Multiple Clocks state sampling option. The Multiple Clocks sampling option is new and is a low speed option more suitable for general-purpose logic analysis. This option is provided only with the U4164A and 16860-series logic analyzers.
Refer to the topic "[To select the state sampling option](#)" on page 98 to know more about these two sampling options.
- **Multiple clocks support** - You can now set up to four multiple clocks in an Ored combination for your logic analyzer. The multiple clocks feature is provided only with the U4164A and 16860-series logic analyzers.
Refer to the topic "[To set up the state sampling clock](#)" on page 124 to know more about setting up multiple clocks.
- **Advanced clocking feature** - When setting up multiple clocks, you can use the new Advanced Clocking feature to define complex clock descriptions. This feature is provided only with the U4164A and 16860-series logic analyzers.
- **Master/Slave and Demultiplex clock modes** - When setting up multiple clocks, you can now use the Master/Slave or Demultiplex clock modes for the U4164A and 16860-series logic analyzers.
Refer to the topics "[Master/Slave Sampling Clock Mode](#)" on page 101 and "[Demultiplex Sampling Clock Mode](#)" on page 103 to know more about these clock modes.
- **New ONFi Analysis window** - You can now capture and analyze ONFi data from NAND flash memory devices using the new ONFi Analysis window available in the Logic and Protocol Analyzer GUI. You can add an instance of this window to your logic analyzer module in the GUI and then compute decoded ONFi transactions and ONFi traffic statistics from the captured data.
Refer to the ONFi Analysis Window User Guide to know more.
- **New Mode Registers Overview tab** in the DDR/LPDDR Memory Analysis window - You can now use this new tab to obtain an overview of your SDRAM's mode registers values and compare these values in various ways (across ranks or with immediate previous settings or with values at another time instance).
Refer to the chapter **Analyzing Mode Registers Values** in the **DDR/LPDDR Memory Analysis User Guide** to know more.
- **3DS DDR4 rank addressing modes** in the B4661A Memory Analysis tools- You can now select standard, Quad CS, or 3DS rank addressing mode for your DDR4 RDIMM device in the DDR Bus Decoder, Memory Analysis window, Real Time Compliance tool, and DDR/LPDDR Post Process Compliance tool.

- See Also
- "[Version 06.20 What's New](#)" on page 33
 - "[Version 06.10 What's New](#)" on page 35
 - "[Version 06.00 What's New](#)" on page 38

Version 06.20 What's New

In this release, version 06.20, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **U4164A Logic Analyzer module** -The Keysight Logic and Protocol Analyzer application now adds support for a new AXIe-based logic analyzer module U4164A. You can use this module to capture and analyze DDR 2/3/4 and LPDDR 2/3/4 data. U4164A is installed in a Keysight AXIe chassis and configured and controlled using the Keysight Logic and Protocol Analyzer application. This module supports maximum of three cards in a multi-card set and supports quad samples and quarter channel timing mode. This module allows you to perform full DDR4 data capture at speeds > 2.5 Gb/s from a single probe point.
Refer to the U4164A Logic Analyzer Notes to know more about this module.
- **Quad Sample clock mode** - A new State sampling clock mode - Quad Sample is provided for the U4164A module that allows four samples per clock edge to support high speed data capture. It allows you to capture separate rising and falling edge samples of both read and write DDR/LPDDR DQ signals for data rates up to 4 Gb/s from a single probe point. 02G license of U4164A is required for this clock mode. Refer to the topic "[To set up the quad sample sampling clock mode](#)" on page 119 to know more.
- **Dual sample clock mode**, when used with the U4164A module now supports two thresholds that you can assign as separate offsets to capture two samples per clock edge with different thresholds. Refer to the topic "[To set up the dual sample sampling clock mode](#)" on page 116 to know more about how dual sampling functions in the U4164A module.
- **Quarter Channel Timing Mode** - A new Timing (Asynchronous) sampling option has been provided for the U4164A module- Quarter Channel Timing Mode. 01G or 02G license of U4164A is required for this timing mode.
- **Clock Hysteresis** - U4164A provides a new setting called Clock hysteresis for the state sampling clock on Pod1 of the U4164A module. Refer to the topic Setting Clock Hysteresis for the U4164A State Sampling Clock to know more.
- **Signal Deskew Tool** - This new tool allows you to set sampling positions for the Timing (Asynchronous) sampling modes of a U4164A logic analyzer. Refer to the online help available in the tool's GUI to know more.
- **New B4661A Memory Analysis Software Tools** - Following standard and licensed tools are available in B4661A.
 - Standard features
 - Default configurations
 - Setup Assistant
 - Configuration Creator tool
 - Eye Finder and Eye Scan
 - Licensed features
 - DDR decoder with physical address trigger tool
 - LPDDR decoder
 - DDR and LPDDR compliance violation analysis tool
 - DDR3/4 and LPDDR2/3/4 performance analysis tool (Memory Analysis window)
- **New W4640-series DDR/LPDDR BGA Interposers** - Two new interposers W4641A and W4643A have been introduced for use with the U4164A module to achieve higher data rates with smaller KOV. These interposers effectively utilizes the single touch probing and quad sampling features of the U4164A logic analyzer module thereby allowing you to probe DDR4 DQ signals above 2.5Gb/s without double probe load.

See Also • "[Version 06.10 What's New](#)" on page 35

- ["Version 06.00 What's New"](#) on page 38

Version 06.10 What's New

In this release, version 06.10, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **U4431A MIPI M-PHY Analyzer module** - The following new features have been added.
 - **NVMe, AHCI, and SSIC transactions decoding** - The Transaction Decode tab in the Protocol Viewer now supports computing and viewing NVMe, AHCI, and SSIC transactions from the captured data.

Refer to the help book *Viewing Decoded Transactions* in the *U4431A MIPI M-PHY Analyzer module online help* to know more.
 - **SSIC test assertion** - A new tab, Test assertion has been added to the Protocol Viewer window to allow you to run predefined SSIC test assertions on the captured trace.

Refer to the help book *Computing and Viewing the Test Assertions* in the *U4431A MIPI M-PHY Analyzer module online help* to know more.
 - **LTSSM states and state transitions** - You can now view LTSSM states and state transitions as detected in the MPCIe or SSIC data captured in a trace using the newly added LTSSM Overview tab in the Protocol Viewer.

Refer to the help book *Viewing LTSSM States and State Transitions* in the *U4431A MIPI M-PHY Analyzer module online help* to know more.
 - **Command Line Packet Generator (CLPG)** - You can now use the U4431A module for transmitting stimulus on an M-PHY link and capturing bidirectional M-PHY data, that is, capturing the data that it transmits as stimulus as well as the data that it receives from a DUT. For this usage scenario, you need the Keysight Command Line Packet Generator (CLPG), which is available as a licensed software option (U4431A-613) of the U4431A module,

For detailed information on CLPG and how to use it for stimulus transmission and capture, refer to the *Keysight MIPI M-PHY Command Line Packet Generator User Guide* installed with the CLPG software at *C:\Program Files\Keysight Technologies\Technologies\Logic Analyzer\Help\pdfs*.
- **U4301 PCIe Gen3 Analyzer module** - The following new features have been added.
 - **Compact tool** - You can now compact the display of training sequence packets into sets in Protocol Viewer using the new Compact tool available in the Protocol Viewer toolbar.

Refer to the topic *Compacting the Display of Training Sequence Packets* in the *U4301 PCIe Gen3 Analyzer Online Help* to know more.
 - **Auto link width detection** - You can now use the new auto link width detection feature to configure the U4301 module to automatically detect link width during initial link up as well as link width changes during up/down configurations and accordingly sets its link width based on these changes as seen in the captured data. This allows data capture as per the changing link width.

Refer to the topic *Specifying the Connection Setup* in the *U4301 PCIe Gen3 Analyzer Online Help* to know more.
 - **Auto lane reversal** - You can now use the new auto lane reversal feature to configure the U4301 module to automatically detect lane reversal based on the TS ordered sets that it sees during the LTSSM training of the monitored link. Using this feature, the module automatically switches its lane reversal to On or Off to match the lane reversal On/Off that it detects on the monitored link.

Refer to the topic *Specifying the Connection Setup* in the *U4301 PCIe Gen3 Analyzer Online Help* to know more.
 - **L1 substates support** - The U4301 Analyzer module now supports L1 substates, L1.1 and L1.2. You can now:

- capture entry and exit from ASPM and PCI-PM L1 sunstates.
- view L1 substates and their transitions in the LTSSM Overview tab.
- set the trigger on 'CLKREQ# Asserted' and 'CLKREQ# Deasserted' events.
- search and set a filter based on CLKREQ# for the packets displayed in the Protocol Viewer.
- qualify whether or not the CLKREQ# transitions should be stored in the Analyzer module's memory while capturing data.

- **Trigger on physical layer errors** - The U4301 Analyzer module now allows you to set up a trigger on physical layer errors such as symbol errors, disparity errors, block header errors.

Refer to the topic *Setting up a Trigger on Physical Layer Errors* in the U4301 PCIe Gen3 Analyzer Online Help to know more.

- **Trigger on link width change and trigger on link speed change** - The U4301 Analyzer module now allows you to set up a trigger on link width and link speed changes.
- **Lane alignment** - For data capture, two new options have been introduced for lane alignment - Strict and Relaxed lane alignment.

Refer to the topic *Setting the Capture Options* in the U4301 PCIe Gen3 Analyzer Online Help to know more.

- **Trimming non-overlapped data** - You can now configure the U4301 module to enable or disable the trimming of non-overlapped data while capturing data. Refer to the topic *Setting the Capture Options* in the U4301 PCIe Gen3 Analyzer Online Help to know more.
- **Tuning enhancements** - Two new tuning methods, Long Tune and Long Fine Tune have been added to tune the U4301 Analyzer module for a specific DUT. These methods take longer time to complete tuning as compared to other tuning methods but can reduce the trace errors resulting from inadequate tuning (after you have completed standard tune and fine tune).

Refer to the topic *Creating a Physical Layer Tuning File* in the U4301 PCIe Gen3 Analyzer Online Help to know more.

- **Filtering enhancements** - The Filtering feature available in the Protocol Viewer has now been enhanced to support filtering of packets based on the error types.

Refer to the topic *Filtering Packets Displayed in Protocol Viewer* in the U4301 PCIe Gen3 Analyzer Online Help to know more.

- **Packet search enhancements** - The Quick Search feature available in the Protocol Viewer has now been enhanced to support searching the packets based on the error types.

Refer to the topic *Searching for Specific Packets in Protocol Viewer* in the U4301 PCIe Gen3 Analyzer Online Help to know more.

- **Exporting data capabilities added to Traffic Overview and Performance Overview tabs of the Protocol Viewer** - You can now export packet times data and charts data from the Traffic Overview and Performance Overview tabs respectively to a specified .csv or an Excel file.

Refer to the topics *Exporting Performance Summary Charts Data to a .csv or an Excel File* and *Exporting Packet Times from a Traffic Overview Chart to a .csv or an Excel File* in the U4301 PCIe Gen3 Analyzer Online Help to know more.

- **NVMe performance charts** - You can now generate performance charts for the NVMe transactions computed in the Transaction Decode tab.

Refer to the topic *Viewing a Transaction Performance Chart* in the U4301 PCIe Gen3 Analyzer Online Help to know more.

- **Scope Power Analysis** - While using the external oscilloscope time correlation and data display feature, you can now specify the resistance used at the current probe point to allow scope power analysis from the scope voltage and supplied resistance. If the current probe is enabled for an analog signal, Amperage is now used as the Base unit for the signal's data in the Waveform Viewer and Listing Viewer. Refer to the topic *Channels tab* in the *External Oscilloscope Time Correlation and Data Display* Online Help to know more.
- Two new probes, **U4328A M.2 (M-key) Interposer** and **U4330A SFF-8639 Interposer** have been added to the list of currently available probing options for use with the U4301 Analyzer module.

Refer to the *PCI Express Gen3 Hardware and Probing Guide* to get detailed information on each of these new probes.
- **U4421 D-PHY Analyzer and Exerciser module** - The following new features have been added.
 - You can now extract a compressed image from the captured DSI packet formats as well as insert a compressed image while generating stimulus from bitmap files. You can also customize the PPS values for a compressed image using the new DSI Image Compression Settings tool. This tool is integrated with the Image Inserter as well as the Image View tab. Refer to the topics *Extracting Images from the Packet Data* and *Using the Image Inserter Application* in the *U4421 D-PHY Analyzer and Exerciser* Online Help.
 - You can now add background and foreground colors to user-defined symbols while creating or editing these symbols for bus/signal values. These colors are then used for symbols to present data in the Waveform and Listing viewers.

See Also • ["Version 06.00 What's New"](#) on page 38

Version 06.00 What's New

In this release, version 06.00, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- The Keysight Logic and Protocol Analyzer application now adds support for a new AXIe-based module U4154B to capture and analyze DDR 2/3/4 and LPDDR 2/3/4 data. U4154B is installed in a Keysight AXIe chassis and configured and controlled using the Keysight Logic and Protocol Analyzer application. This module supports three cards in a multi-card set and has improved clock qualifiers to capture traces out of reset. This module enables DDR4 interposer to perform full data capture at speeds > 2.5 GB/s. Refer to the U4154B Logic Analyzer Notes to know more.
- The support for legacy logic analysis systems such as 16760/1680XA/1682XA/16962A/168x/9x has been removed from the Logic and Protocol Analyzer software. These systems are supported only on the software version 5.9 or earlier.
- **U4431A MIPI M-PHY Analyzer module -**
 - Transaction decoding - You can now compute and view decoded transactions from the captured MPCle data using the newly added Transaction Decode tab in the Protocol Viewer. Refer to the help book Viewing Decoded Transactions in the U4431A MIPI M-PHY Analyzer module online help to know more.
 - Offline performance summary computation - You can now compute and view offline performance summary from the captured MPCle data using the newly added Performance Overview tab in the Protocol Viewer. This tab presents statistics for various performance parameters in tabular as well as charts form. Refer to the help book Viewing Offline Performance Summary in the U4431A MIPI M-PHY Analyzer module online help to know more.
 - A new feature, Flow Control is now available in the Performance Overview tab of the Protocol Viewer. This feature allows you to compute and track the available flow control credits for the data trace that has bidirectional MPCle traffic. Refer to the topic Viewing Offline Performance Summary in the U4431A MIPI M-PHY Analyzer module online help to know more.
- **U4301B PCIe Gen3 Analyzer module -** The following enhancements have been made.
 - Traffic overview chart - The Traffic Overview tab in the Protocol Viewer now provides a traffic overview band chart based on the packet types.
 - Filtering improvements - The procedure for filtering packets in the Protocol Viewer has been simplified. A new button has been added to the Protocol Viewer to set up the filtering options.
 - Simpler packet search - The procedure for searching packets in the Protocol Viewer has been simplified. A new text box has been added to the Protocol Viewer's upper pane to allow you to search on the basis of packet names. This text box has the AutoComplete feature.
 - Eliminate separate view
 - Capture InitFC - The Performance Overview tab in the Protocol Viewer now has the feature to set the InitFC discovered in the trace as the default InitFC.
 - Flow control for header packets - The Performance Overview tab in the Protocol Viewer now provides options for setting Header Flow Control Init values.
 - Equalization summary for Gen3 training sequences - The LTSSM Overview tab in the Protocol Viewer now provides Equalization Summary section for Gen 3 training sequences exchanged between the link partners.
 - 1 button computation - A new Compute button has been added to the Protocol Viewer that allows you to compute traffic overview statistics, decoded transactions, and offline performance summary for the captured packets by a single click of this button.

2 Getting Started

Tutorial - Getting to know your logic analyzer / 40
Measurement Examples / 50
Demo Center / 55

Tutorial - Getting to know your logic analyzer

The following tutorial is intended to give you a quick of logic analyzer basics. In addition to learning the concepts of logic analysis, you will see some of the logic analyzer's more common features by going through a measurement. Finally, you are shown some easy time saving tasks that can quickly make you as productive as a more experienced user.

Logic analysis basics

- When should you use an oscilloscope? (see [page 354](#))
- When should you use a logic analyzer? (see [page 355](#))
- What is a logic analyzer? (see [page 356](#))
- Timing analyzer:
 - Sampling clock (see [page 356](#))
 - Sampling (see [page 356](#))
 - Triggering (see [page 357](#))
- State analyzer:
 - Sampling clock (see [page 358](#))
 - Sampling (see [page 359](#))
 - Triggering (see [page 359](#))

Measurement

The following does not require an active device under test. However, in order to show features that work on data, you are asked to load a configuration file between steps 5 and 6 that contains data to finish the exercise.

- Turning on the logic analyzer (see [page 40](#))
- Connecting to the device under test (see [page 41](#))
- Setting up bus/signal names (see [page 42](#))
- Setting the acquisition mode (see [page 43](#))
- Setting up a simple trigger (see [page 43](#))
- Open the tutorial configuration file (see [page 44](#))
- Using markers (see [page 44](#))
- Zooming in on the data (see [page 45](#))

Time saving tasks

- Loading and saving configuration files (see [page 46](#))
- Saving and recalling trigger setups (see [page 46](#))
- Quick marker measurements (see [page 47](#))
- Searching data (see [page 48](#))
- Toolbars, tool tips, and mouse shortcuts (see [page 49](#))

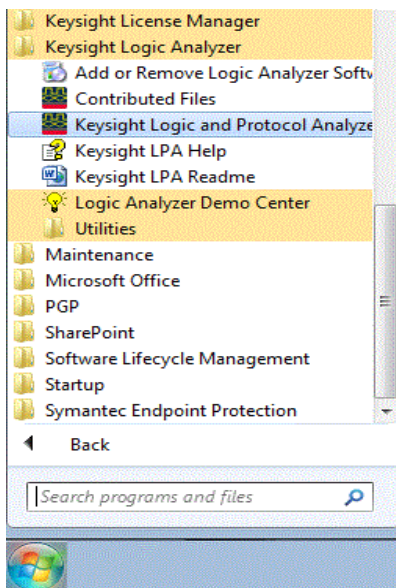
See Also

- Product Overviews (see [page 520](#))

Turning on the logic analyzer

[[Tutorial Home](#) (see [page 40](#))] [[Next Topic](#) (see [page 41](#))] [[Previous Topic](#) (see [page 359](#))]

- 1 Consider ergonomics when positioning the keyboard and mouse.
- 2 Plug in the power cable and press the front-panel Standby button.
- 3 From the Windows Start bar, click **Start>Programs>Keysight Logic Analyzer>Keysight Logic Analyzer**.

**NOTE**

Optional: If you have a logic analyzer shortcut icon on screen, double-click the icon.

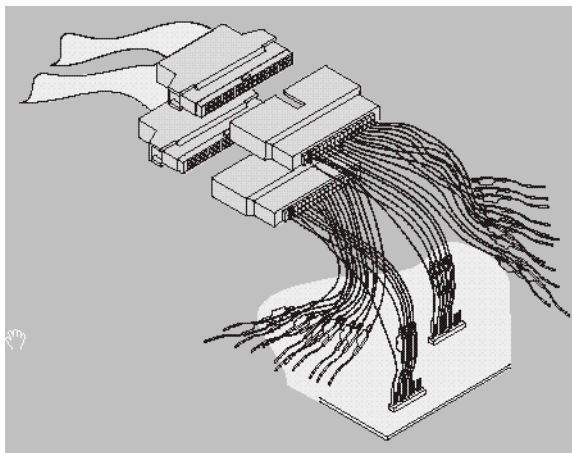
Connecting to the device under test

[[Tutorial Home](#) (see [page 40](#))] [[Next Topic](#) (see [page 42](#))] [[Previous Topic](#) (see [page 40](#))]

The first step in using a logic analyzer is to probe signals in the device under test.

NOTE

In this tutorial, no probe connections are required. Later on in this tutorial, you are asked to load a configuration file containing data to simulate the results of a probed device under test.



For more information about probing options, see [Probing the Device Under Test](#) (see [page 57](#)).

Setting up bus/signal names

[Tutorial Home (see page 40)] [Next Topic (see page 43)] [Previous Topic (see page 41)]

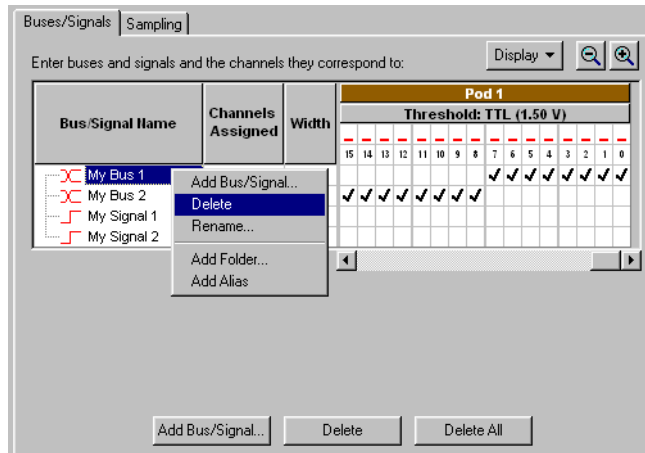
By default, the analyzer has one bus (My Bus 1) set up in the interface. The following exercise cleans up the display defaults and re-configures the bus/signal setup for a new measurement.

Delete bus/signal names

- 1 In the menu bar click **Setup>(Logic Analyzer Module)>Bus/Signal...**
- 2 In the Analyzer Setup dialog that appears, right-click on **My Bus 1**, then select **Delete**. Repeat until all bus signal names are deleted. After the last bus/signal is deleted, "My Bus 1" appears again as a default name.

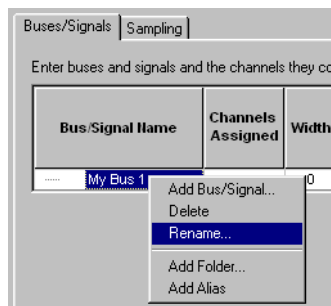
TIP

You can delete all bus/signal configurations at once with the Delete All button.



Add new bus/signal name

- 1 In the Analyzer Setup dialog, right-click on **My Bus 1**, then select **Rename**.
- 2 From the popup keypad that appears, type in the new name "counter".
- 3 Select **OK**.



Map signals into the analyzer

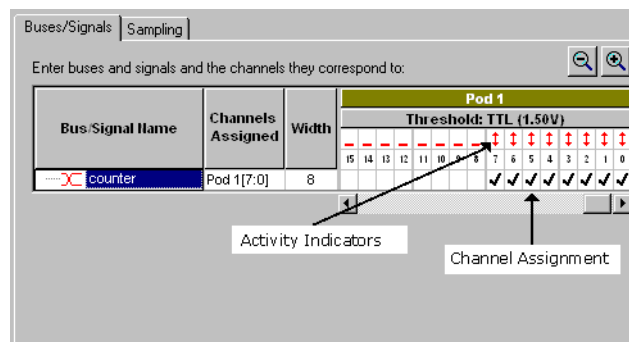
The logic analyzer must be told which probed signals from the device under test are to be included in the measurement, and how you want them grouped. In this exercise, you assign channels 0 - 7 on Pod 1 under the name "counter". Notice that when more than one channel is assigned to "counter" it becomes a bus.

- 1 Check the activity indicators for verification of proper connection to the device under test. You should see a transition arrow on all 8 channels.

NOTE

If you have real device under test hardware, you will see activity indicators as shown below. If you are loading the demo configuration file (later in this tutorial) you will not see activity.


- 2 Click each **channel assignment box** under channels 0 - 7 on Pod 1. Notice that as you assign channels, the configuration information is updated for the bus/signal.
- 3 Click **OK**.

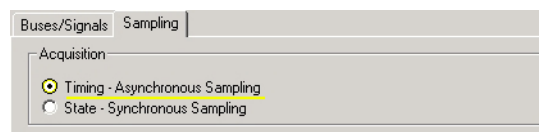


Setting the acquisition mode

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 43](#))] [Previous Topic (see [page 42](#))]

Under the Sampling tab of the Analyzer Setup dialog is where you set the analyzer to be either a timing or state analyzer. You also set either the timing options, such as memory depth or sampling period, or the state clocking options.

- 1 From the menu bar, click **Setup>(Logic Analyzer Module)>Timing/State (Sampling)...**, or click the  icon in the toolbar.
- 2 Select **Timing - Asynchronous Sampling**.





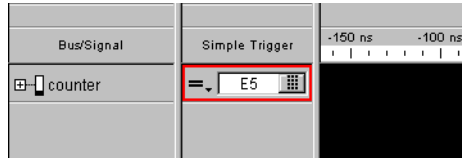
- 3 Click **OK**.

Setting up a simple trigger

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 44](#))] [Previous Topic (see [page 43](#))]

The Simple Trigger is a quick way to configure the analyzer to trigger on either a data pattern on a bus, or an attribute of a single signal such as a rising edge or a low logic level.

- 1 In the Simple Trigger column, click on the **pattern qualifier**  and set it to **Equal**.
- 2 Click in the **text entry field**  and enter the data pattern "E5".



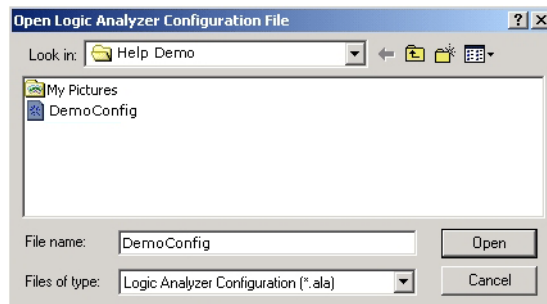
Open the tutorial configuration file

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 44](#))] [Previous Topic (see [page 43](#))]

At this point in a measurement, you would normally run the logic analyzer. However, because you are not connected to a device under test, you cannot capture real data. You will have to load a configuration file that contains this data.

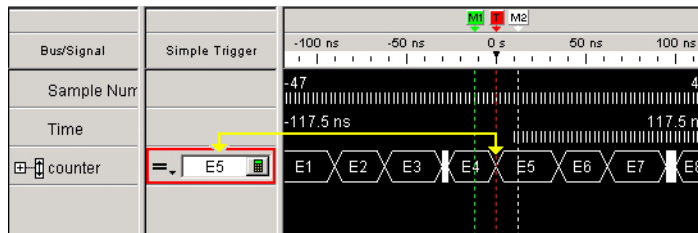
Load the configuration file

- 1 Select **File>Open**.
- 2 From the file manager dialog, select the file named **DemoConfig.ala** from the following directory:
C:\Documents and Settings\All Users\Documents\Keysight Technologies\Logic Analyzer\Default Configs\Keysight\Help Demo
- 3 Select **Open**.



View the data

Notice how the logic analyzer triggered on data pattern E5 and placed it in the center of the display. The red line shows that the trigger point is at the start of the data pattern E5.



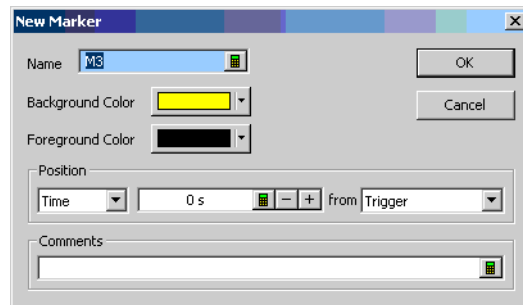
Using markers

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 45](#))] [Previous Topic (see [page 44](#))]

Markers are used for creating reference points in data. Once markers are placed in data, you can use them to quickly see what time, sample, or data value the marker is set on.

To create a marker

- 1 From the menu bar, click **Markers>New**.
- 2 From the New Marker dialog that appears, configure the new marker and if desired, specifically a position it in data. When you do not position the marker, by default it is placed at the trigger point.
- 3 Select **OK**.



To place a marker in data

When you first create a new marker, you have the option to place it in data at a specific point in time or a specific sample number. The following exercise shows you other ways to position markers in data.

- 1 In the display, click on marker M3 (your new marker) and while holding the mouse button down, drag maker M3 to -100ns before trigger, then release. Notice that the marker position value changes as you move it.
- 2 From the menu bar, click **Markers>Place On Screen**, then select M1 and click **OK**. Notice how M1 is placed at center screen at the red trigger line.
- 3 Point the mouse cursor at any desired point in data, then right-click and select **Place Marker**. From the Place Marker dialog that appears, choose the M2 marker. Notice that the marker is placed where the mouse was pointing.

Go To a marker in data

Once you have markers set in data, you can quickly find any of them as follows.

- 1 From the menu bar, click **Markers>GoTo**.
- 2 Select the marker you want to find, and click **OK**.

Zooming in on the data

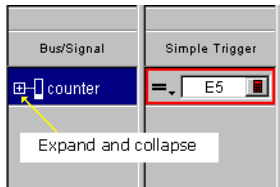
[Tutorial Home (see [page 40](#))] [Next Topic (see [page 46](#))] [Previous Topic (see [page 44](#))]

Data from a timing analyzer is displayed (as on an oscilloscope) as waveforms on a horizontal time axis. To zoom in or out on a waveform, change the Scale (time/division) of the time axis of the waveform.

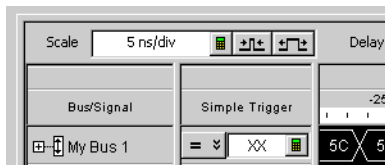
Both state and timing analyzers can have multiple signals grouped together in a bus. To get a view of all signals, you can expand a bus into individual signals.

Expand a bus

Click the "+" symbol just to the left of the bus named "counter". The collection of signals under "counter" breaks out into individual signals named counter[0] - counter[7].



Change the scale Click the zoom out icon to expand the signals to where you want them.



Loading and saving configuration files

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 46](#))] [Previous Topic (see [page 45](#))]

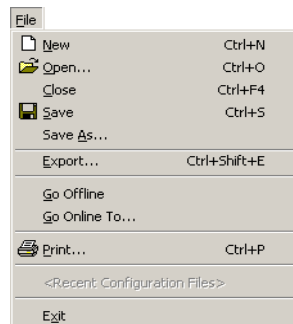
Many times it is quicker to open an existing configuration file with a similar setup than to create a new configuration from scratch. You simply open a similar file, make the appropriate changes to the setup, then save the file as a new filename.

NOTE

When you rename an existing configuration file, you retain the saved trigger setups and "Find" search favorites from the first configuration file.

You already have learned how to open a configuration file. In the following exercise, you will save the "democonfig" file to a new name.

- 1 From the menu bar, click **File>Save As...**
- 2 From the file manager dialog that appears, type in the new name "myconfig", then click **Save**.



Saving and recalling trigger setups

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 47](#))] [Previous Topic (see [page 46](#))]

Each time you set up a new trigger and run the measurement, the trigger setup is saved in the logic analyzer. It is quicker to recall a trigger setup rather than re-configure the trigger setup each time.

NOTE

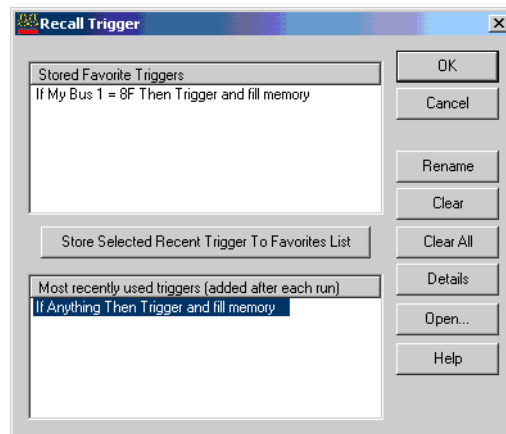
The logic analyzer must be run before the trigger setup is stored. Also, trigger setups are stored as part of the configuration file. If you load a new configuration file, the trigger setups will be overwritten by trigger setups stored with the new file.

To recall a trigger setup

- 1 From the menu bar, click **Setup>(Logic Analyzer Module)>Recall Trigger...**
- 2 From the lower list, select the desired trigger setup, then click **OK**.

TIP

When the list of most recently used triggers get long, you can store the most often used triggers in the upper favorites list.

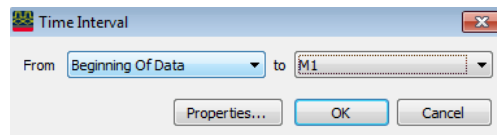


Quick marker measurements

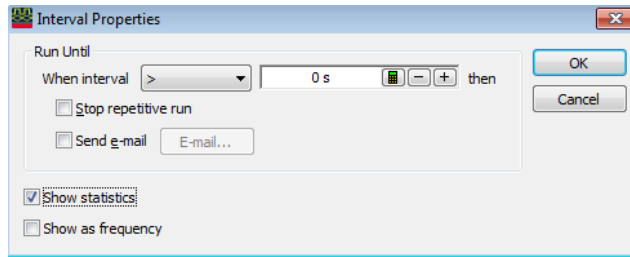
[[Tutorial Home](#) (see [page 40](#))] [[Next Topic](#) (see [page 48](#))] [[Previous Topic](#) (see [page 46](#))]

You can quickly read the time or number of samples between markers.

- 1 Click **Markers>New Time Interval Measurement**.
- 2 Configure the Interval dialog to display the time from **Beginning of Data** to **Trigger** as shown below.



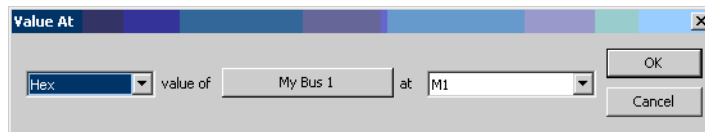
To show statistics with the time interval measurement (after repetitive runs, click **Properties**; then, in the Interval Properties dialog, check **Show statistics**).



Click **OK** to close the Interval Properties dialog, and click **OK** to close the Time Interval dialog. After a repetitive run, the result of the time interval measurement is displayed in the marker measurements display bar.

Beginning Of Data to Trigger = 3.860812 ms (511.164 us / 3.860812 ms / 2.279915 ms)

- 3 Click **Markers>New Value At Measurement**.
- 4 Configure the Value At dialog to display the **Hex** value of **My Bus 1** at **M1** as shown below; then, click **OK**.



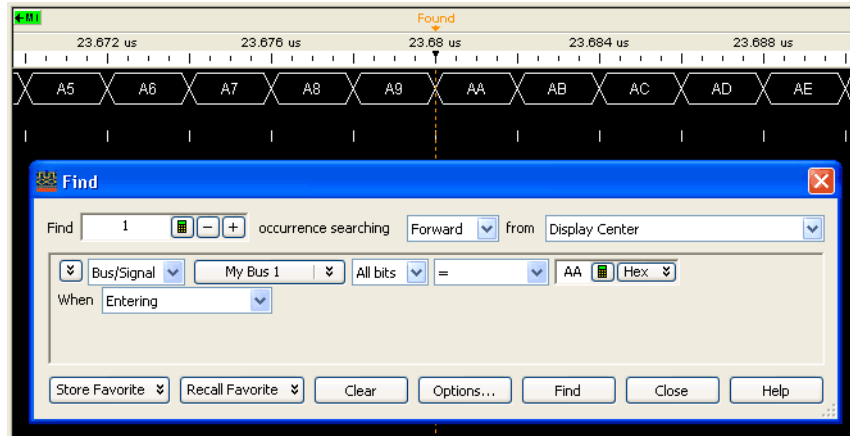
The result of the value at measurement **My Bus 1@M1 = 4B** is displayed in the marker measurement display bar.

Searching data

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 49](#))] [Previous Topic (see [page 47](#))]

You can search for a data pattern on a bus or a signal. You can also choose when the search begins and ends. Finally, you can save the search criterion in a favorites list.

- 1 From the menu bar, click **Edit>Find...**
- 2 In the Find dialog, configure the search criterion as shown below to find "AA".
- 3 Select **Find**.



As you configure the Find dialog, try to think of it as constructing a sentence that reads left-to-right.

"Find the 1st occurrence searching Forward from Display Center, on a bus named My Bus 1, including All bits, a pattern that Equals AA".

Toolbars, tool tips, and mouse shortcuts

[Tutorial Home (see [page 40](#))] [Previous Topic (see [page 48](#))]

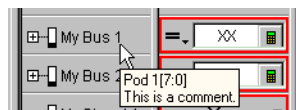
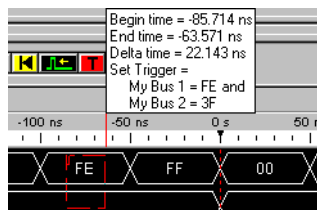
Throughout this tutorial, the menu bar has been used to access features. There are two other ways to access features as well as other useful tips that can save you time.

Toolbars Below the menu bar are groups of icons that represent shortcuts to many dialogs and features. For more information refer to Toolbars (see [page 398](#)) in the main help.



Mouse Shortcuts There are many mouse shortcuts available. To access them simply point the mouse over a screen element such as a marker, or screen area, then right-click the mouse. Mouse shortcuts are especially useful within the waveform and listing data display areas.

Tool Tips Tool tips are small information displays that appear during operations such as moving markers, setting a trigger with the mouse, or hovering the mouse over a bus/signal name. Use them as comments (see [page 94](#)), or to monitor your progress or current positions.



Measurement Examples

The following measurement examples show you the typical order of steps to set up and run a measurement. As you go through the examples, you will encounter steps such as probing or triggering where alternative choices are available. In these steps, select the probing or trigger example that best fits your measurement.

- Making a timing analyzer measurement (see [page 50](#))
- Making a state analyzer measurement (see [page 51](#))
- To trigger on one of multiple edges (see [page 52](#))
- To trigger on ranges (see [page 53](#))

- See Also
- Tutorial - Getting to know your logic analyzer (see [page 40](#))
 - Timing Mode Trigger Functions (see [page 478](#))
 - State Mode Trigger Functions (see [page 490](#))

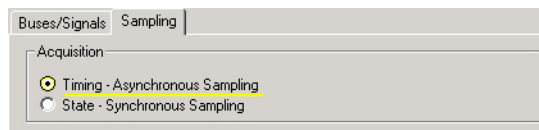
Making a timing analyzer measurement

The following measurement example shows you the steps necessary to configure and run the logic analyzer for a typical timing analyzer measurement. As you go through the example, make the appropriate choices from the selection lists that best match the kind of configuration you need.

TIP

If you are new to logic analysis, refer to "Tutorial - Getting to know your logic analyzer (see [page 40](#))" for a quick tutorial on logic analysis concepts and measurements.

- 1 Connect the probing to the device under test (see [Probing the Device Under Test \(see page 57\)](#) for more information).
 - 2 Turn on the logic analyzer.
- Bus and signal setup
- 1 In the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**
 - 2 From the Buses/Signals tab, assign bus/signal names to the device under test signals probed. You do this by either renaming (see [page 85](#)) existing names, or deleting (see [page 84](#)) and creating (see [page 84](#)) new names.
 - 3 From the Buses/Signals tab, define buses and signals (see [page 83](#)) under the appropriate pods for all probed buses/signals on the device under test.
- Acquisition mode setup
- 1 In the Analyzer Setup dialog, select the **Sampling** tab.
 - 2 From the Sampling tab, set the acquisition mode to Timing - Asynchronous Sampling.



- 3 Set the Sampling Options (see [page 97](#)).
- 4 Set the timing mode Sampling Period (see [page 98](#)).

- Trigger setup
- 1 The trigger required to capture specific data depends on the measurement. However, the trigger is generally set in two ways.
 - From within the data display, set up a simple trigger (see [page 146](#)).
 - From the Advanced Trigger dialog (see [page 424](#)), set up a timing mode advanced trigger (see [page 478](#)) function.
- Run the measurement
- 1 Run (see [page 196](#)) the measurement.
- See Also
- To specify the trigger position (see [page 128](#))
 - To set acquisition memory depth (see [page 129](#))

Making a state analyzer measurement

The following measurement example shows you the steps necessary to configure and run the logic analyzer for a typical state analyzer measurement. As you go through the example, make the appropriate choices from the selection lists that best match the kind of configuration you need.

TIP

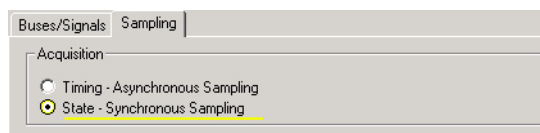
If you are new to logic analysis, refer to "Tutorial - Getting to know your logic analyzer (see [page 40](#))" for a quick tutorial on logic analysis concepts and measurements.

- 1 Connect the probing to the device under test (see Probing the Device Under Test (see [page 57](#)) for more information).

NOTE

Be sure that the clock signals of your device under test are connected to clock channels on the pods. Any unused clock channels can be used for additional data channels and will not feed into the state clock setup.

- 2 Turn on the logic analyzer.
- Bus and signal setup
- 1 In the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**
 - 2 From the Buses/Signals tab, assign bus/signal names to the device under test signals probed. You do this by either renaming (see [page 85](#)) existing names, or deleting (see [page 84](#)) and creating (see [page 84](#)) new names.
 - 3 From the Buses/Signals tab, define buses and signals (see [page 83](#)) under the appropriate pods for all probed buses/signals on the device under test.
- Acquisition mode setup
- 1 In the Analyzer Setup dialog, click the Sampling tab.
 - 2 From the Sampling tab, set the acquisition mode to State - Synchronous Sampling.



- 3 Set the state clock mode (see [page 99](#)).
- 4 Set the state sampling clock (see [page 124](#)).

- 5 If necessary, set the advanced state clocking (see [page 127](#)).

Trigger setup

- 1 The trigger required to capture specific data depends on the measurement. However, the trigger is generally set in two ways.
 - From within the data display, set up a simple trigger (see [page 146](#)).
 - From the Advanced Trigger dialog (see [page 424](#)), set up an advanced trigger (see [page 490](#)) function.

Run the measurement

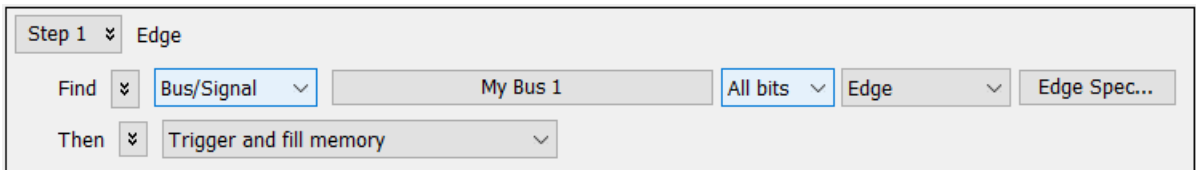
- 1 Run (see [page 196](#)) the measurement.

See Also

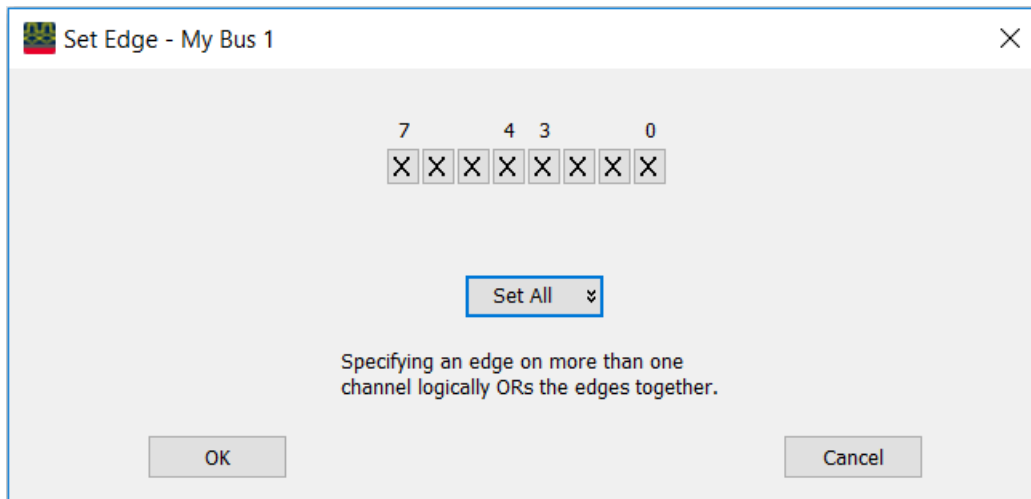
- To specify the trigger position (see [page 128](#))
- To set acquisition memory depth (see [page 129](#))

To trigger on one of multiple edges

- 1 In the timing sampling mode, set up an Advanced Trigger.
- 2 Select the bus on which you're looking for one of multiple edges.
- 3 Select **All bits** in the bus.
- 4 Select **Edge**.



- 5 Click **Edge Spec...**
- 6 In the Set Edge dialog, specify edges you are looking for; use the **Set All** button to make a selection for all signals in the bus.



- 7 Click **OK** to close the Set Edge dialog.
- 8 Click **OK** to close the Advanced Trigger dialog.


To trigger on ranges

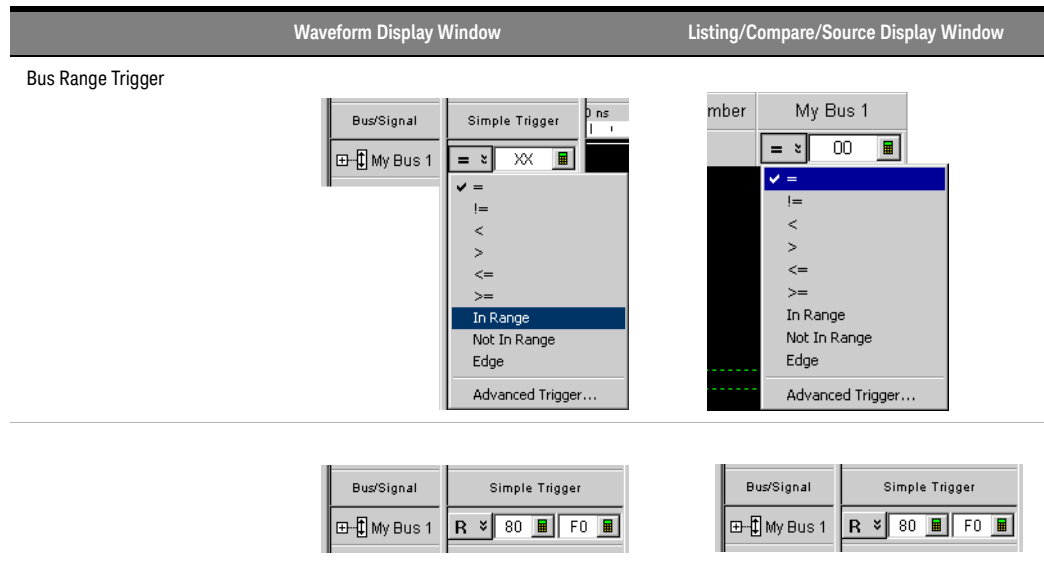
NOTE


In order to trigger on ranges of bus values, the bus:

- Must not have reordered bits.
- Must not contain clock bits that span pod pairs.
- Must span 2 or fewer pod pairs (up to 64 bits wide).


When setting up simple triggers (see [page 146](#))

- 1 In the Simple Trigger field for a bus, click the  operator button; then, choose either **In Range** or **Not In Range**.




- 2 Specify the range values, either by entering values in the low range and high range text entry fields or, when the **Symbol** number base is selected, by using the Select Symbol dialog (see [page 458](#)).
- 3 From the menu bar, choose **Run/Stop>Run**, or click the  icon from the run/stop toolbar (see [page 401](#)).

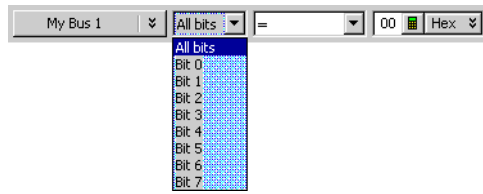
When setting up advanced triggers (see [page 152](#))

- 1 Click  in the analyzer setup toolbar, or choose **Setup>(Logic Analyzer Module)>Advanced Trigger...** from the menu bar.
- 2 In the Advanced Trigger dialog, select the bus.

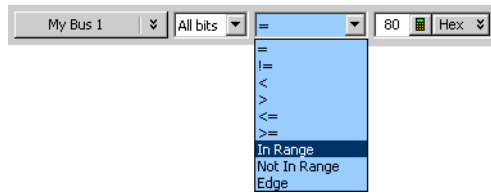


Clicking  lets you select from recently used bus/signal names. Clicking elsewhere on a bus/signal name button opens a Select dialog for selecting a different name.

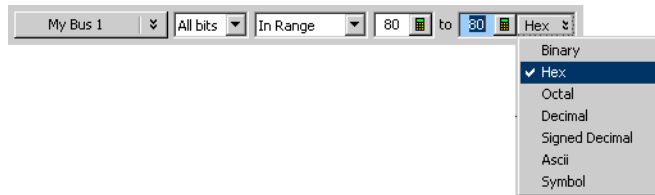
- 3 Select **All bits** on the bus.




- 4 Select either the **In Range** or **Not In Range** operator.



- 5 Select the number base (**Binary**, **Hex**, **Octal**, **Decimal**, **Signed Decimal**, also known as two's complement, **Ascii**, or **Symbol**).




- 6 Specify the range values, either by entering values in the low range and high range text entry fields or, when the **Symbol** number base is selected, by using the Select Symbol dialog (see [page 458](#)).
- 7 From the menu bar, choose **Run/Stop>Run**, or click the  icon from the run/stop toolbar (see [page 401](#)).

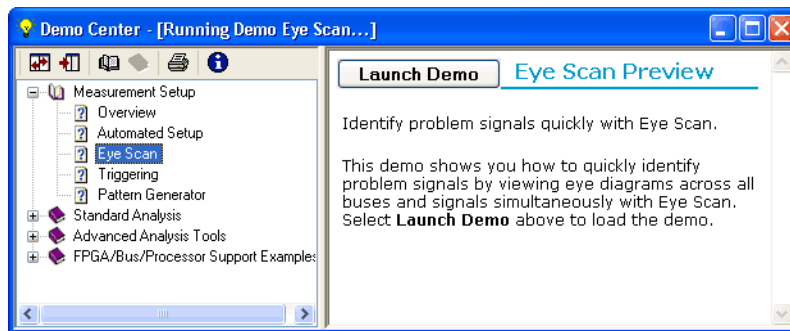
Demo Center




Demo Center is an application that demonstrates logic analysis system features. It loads illustrative configurations into the *Keysight Logic Analyzer* application's *offline-demo* mode and highlights feature capabilities.

To launch Demo Center




- From the *Keysight Logic Analyzer* application's main menu, choose **Help>Show Demo...**
- From the *Keysight Logic Analyzer* application's Demo Center toolbar, click the  Show Demo icon.
- From the Windows Start bar, click **Start>All Programs>Keysight Logic Analyzer>Run Logic Analyzer Demo Center**.

The Demo Center application and the *Keysight Logic Analyzer* application are tiled horizontally on the desktop.



The  Float button maximizes the *Keysight Logic Analyzer* application's window lets the Demo Center application window float over it. The  Locate button returns to horizontally tiled windows. The  What is Demo Center button displays more information.

To use Demo Center

- 1 Use the left pane to navigate to the feature demonstration you want to view. The  Expand and  Collapse buttons affect the feature hierarchy tree.
- 2 Select the feature you want to learn about by clicking it. Information about the feature appears in the right pane.
- 3 Click **Launch Demo**.
A configuration file that illustrates the feature is loaded into the *Keysight Logic Analyzer* application, and more information about the feature appears in Demo Center's right pane. The  Print button lets you print the information.
- 4 When you are done exploring the feature, click **Press to Select Another Demo**.

3 Probing the Device Under Test

Before you can make logic analysis measurements on a device under test, you must connect the logic analyzer channels to (in other words, probe) the device under test.

There are several options for probing a device under test:

- Connecting to individual IC pins or test points. – This is called *general purpose probing* and is accomplished with flying-lead probe sets. Accessories that help with general purpose probing are also available.
- Connecting to all the pins of a specific QFP package. – This is called *QFP package probing* and is accomplished with optional elastomeric probe adapter and 1/4 flex adapter products available for several types of QFP packages.
- Designing connectors (or pads and mounting holes for soft-touch connector less probes and retention modules) into the device under test. – This is called *target connector probing or soft-touch connector less probing* and is accomplished with optional probes available for various signal and connector types.
- Using processor- or bus-specific probes. – These are called *analysis probes* (formerly called *preprocessors*) and are available for many processors and buses. Analysis probes include configuration files for setting up the logic analyzer, and they may include inverse assemblers or other post-processing tools for decoding captured data.

See Also For more information on general-purpose probing, QFP package probing, target connector and connector less probing, and other probing options, see:

- ["Probing Selection Quick Reference Card"](#)
- ["Probing Solutions for Logic Analyzers"](#) (🌐 **"latest version on web"**)
- [📄 "Logic Analyzer Probing Solutions"](#)

For more information on analysis probes and other processor and bus solutions, see:

- ["Processor and Bus Support for Logic Analyzers"](#) (🌐 **"latest version on web"**)
- [📄 "Processor, Bus, and FPGA Support for Logic Analyzers"](#)

For more information on probing signals internal to an FPGA or setting up definitions for the probes used, see:

- "Setting Up Probes" (in the online help)

For more information on controlling signals in the device under test from a logic analysis system, see:

- To control signals in the device under test (see [page 58](#))

To control signals in the device under test

The 16850 series logic analysis system *frames* (see [page 615](#)) have a *target control port*, an 8-bit, 3.3V port that can be used to send signals to a device under test. The target control port does not function like a pattern generator, but more like a remote control for switches in the device under test.

To use the target control port outputs:

- 1 Connect the target control port cable to the logic analysis system frame.
The target control cable is keyed, so it can be inserted only one way. Plug it into the target control port with the key up and the cable hanging down.
The wires on the target control port cable are color-coded. Bit 0 is brown, bit 1 is red, bit 2 is orange, and so on up to bit 7 (gray). The black and white wires are both ground. Pins 0, 2, 4, and 6 are on the top of the connector and arranged in the same order as the wires.
- 2 If you plan on using open collector, remember to install pull-up resistors to a maximum 4V. The maximum sink current into the Target Control port is 12 mA and includes a 51 ohm series resistor.

Example: Resetting Your Device Under Test

This example also applies to other types of signals you may want to send to your device under test. The reset line in this case is active low.

- 1 Attach one of the wires from the target control port cable to the reset line, using proper termination.
- 2 In the *Keysight Logic Analyzer* application, choose **Setup>Target Control Port...**
- 3 In the Target Control Port dialog (see [page 475](#)), check **Enabled** for the target control port signal you are using.
- 4 Click **1** to output an active high.
- 5 When the device under test needs to be reset, click **Pulse**.

4 Connecting to a Logic Analysis System

If you opened the *Keysight Logic Analyzer* application on a logic analyzer or logic analysis system frame, you are most likely already connected to the local frame. (The right-most part of the status bar shows "Local".)

However, if you are offline (the right-most part of the status bar shows "Offline") and you want to connect to the local frame, or if you want to connect to a different, remote logic analysis system frame, you can do so from the **File** menu.

To connect to the local frame

- Choose the **File>Go Online To Local Frame** menu command.

To connect to a remote frame

- 1 Choose the **File>Go Online To...** menu command.
- 2 In the Select System to Use dialog (see [page 459](#)), select the system to access; then, click **Connect**.
The logic analysis system can be:
 - *Local* – connected to the same computer or logic analyzer running the *Keysight Logic Analyzer* application.
 - *Remote* – somewhere else on the local area network.
- 3 If you are connecting to a remote logic analysis system that requires a password (see Setting Up Passwords for Remote Access (see [page 68](#))), enter the password in the "Please enter connection password:" dialog, and click **OK**.

When you are connected to a remote logic analysis system frame, the right-most part of the status bar shows "Remote".

Connecting to the U4154A/B Logic Analyzer Module

Connection to the U4154A/B Logic Analyzer module is different from connecting to other logic analysis systems and logic analyzers in terms of the hardware setup of the involved components.

The U4154A/B Logic Analyzer module is installed in a slot of the Keysight AXIe chassis. The Keysight Logic Analyzer application is hosted on a host PC (a laptop or a desktop) or a remote PC connected to the host PC. The U4154A/B module connects to the host PC through the PCIe interface of the AXIe chassis.

The connection to the U4154A/B Logic Analyzer module can be:

- *Local* – the Keysight Logic Analyzer application is installed and running on the host PC which is connected to the U4154A/B module through the PCIe interface of the AXIe chassis.
- *Remote* – the Keysight Logic Analyzer application is installed and running on a computer other than the host PC and is remotely connected to the host PC.

If the PCIe connection between the host PC and AXIe chassis is made, you are most likely already connected to the U4154A/B Logic Analyzer module installed in the AXIe chassis. (The right-most part of the status bar shows "Local".) However, if you are offline (the right-most part of the status bar shows "Offline") and you want to connect to the U4154A/B module locally or remotely, you can do so from the **File** menu. The procedures for connection are the same as described above.

NOTE

It is recommended that you turn off the standby mode and disable hibernation on the host PC connected to the AXIe chassis via PCIe.

For Windows 7/8, it is also recommended that you disable PCIe power management modes on the host PC.

For more on using the Select System to Use dialog, see:

- To add a logic analysis system to the list (see [page 61](#))
- To delete a logic analysis system from the list (see [page 62](#))
- To refresh the logic analysis system list (see [page 63](#))
- To view logic analysis system details (see [page 64](#))
- To enter your "System In Use" comments (see [page 65](#))
- To select a logic analysis system for auto-connect (see [page 66](#))
- To chat with another logic analysis system user (see [page 67](#))

See Also • [Offline Analysis \(see page 217\)](#)

To add a logic analysis system to the list

- 1 In the Select System to Use dialog (see [page 459](#)) (which appears after choosing **File>Go Online To...**), click **Add**.
- 2 Enter the logic analysis system frame hostname or IP address in the dialog, and click **OK**.

See Also · [Returning to Online Analysis \(see page 59\)](#)

To delete a logic analysis system from the list

- 1 In the Select System to Use dialog (see [page 459](#)) (which appears after choosing **File>Go Online To...**), select the logic analysis system you wish to delete.
- 2 Click **Delete**.

See Also · [Returning to Online Analysis \(see page 59\)](#)

To refresh the logic analysis system list

- 1 In the Select System to Use dialog (see [page 459](#)) (which appears after choosing **File>Go Online To...**), click **Refresh**.

The logic analysis system status and detailed information is updated. Status can be:

Status	Description
Available	
Host offline	The logic analysis system has been powered down or taken off the network.
In use	The "System In Use" comment is displayed in parentheses.
Incompatible remote service version	The version information is displayed in parentheses. The <i>Keysight Logic Analyzer</i> application software on the logic analysis system frame needs to be updated to match the version of software installed on the machine displaying this dialog.

See Also · [Returning to Online Analysis \(see page 59\)](#)

To view logic analysis system details

- 1 In the Select System to Use dialog (see [page 459](#)) (which appears after choosing **File>Go Online To...**), select the logic analysis system whose details you wish to view.
- 2 Click **Details...** (you may have to click **More >>** first).
The logic analysis system details are displayed in the Frame/Module Information dialog (see [page 439](#)).

See Also · [Returning to Online Analysis \(see page 59\)](#)

To enter your "System In Use" comments

- 1 In the Select System to Use dialog (see [page 459](#)) (which appears after choosing **File>Go Online To...**), select the logic analysis system whose details you wish to view.
- 2 Click **Set My Comments** (you may have to click **More >>** first).
- 3 In the dialog that opens, enter your "system in use" comments.
These comments usually contain your contact information.

See Also · [Returning to Online Analysis \(see page 59\)](#)

To select a logic analysis system for auto-connect

- 1 In the Select System to Use dialog (see [page 459](#)) (which appears after choosing **File>Go Online To...**), select the logic analysis system that you wish to auto connect to.
- 2 Click **Set As Auto-Connect** (you may have to click **More >>** first).

See Also · [Returning to Online Analysis \(see page 59\)](#)

To chat with another logic analysis system user

- 1 In the Select System to Use dialog (see [page 459](#)) (which appears after choosing **File>Go Online To...**), select the logic analysis system to which you want to send a chat message.
- 2 Click **Chat** (you may have to click **More >>** first).
- 3 In the Chat Select Destination dialog (see [page 430](#)), select whether you want to chat with the person logged into the logic analysis system or the person connected to the frame; then, click **OK**.
- 4 In the Chat dialog (see [page 429](#)), enter your message; then, click **Send**.
- 5 When you are finished with the chat session, click **Close**.

See Also · [Returning to Online Analysis \(see page 59\)](#)

Setting Up Passwords for Remote Access

The *Keysight Logic Analyzer* application lets you connect to and remotely control logic analysis systems on the network.

If you have a logic analysis system you want to restrict remote frame connections to, you can set up a remote access password. You can password-protect hosted instruments too, by setting a password on the host PC. Because any user on the network with the *Keysight Logic Analyzer* application software installed can force any other user off of a frame, you may wish to protect your local system with a password to prevent this from happening.

NOTE

For multiframe logic analysis systems, the remote access password must be the same on every frame. If this is not the case, you will not be able to connect to the multi-frame set. If a password is not set (blank) on one frame, then you will be able to connect to the entire set of frames so long as the remaining frames have identical passwords or a blank password. In other words, a blank password is a don't care condition.

On the logic analysis systems that you want to restrict remote connections to:

- 1 From the Windows Start menu, choose **Start>Programs>Keysight Logic Analyzer>Utilities>Remote Access Password Utility**.
(You can also run this utility by clicking **Set Local Password** in the Select System to Use dialog (see [page 459](#).)
- 2 In the Remote Access Password Utility dialog, if a password is currently set, click **Clear Password**.



- 3 Enter the new password in the **Enter New Password** and **Re-enter New Password** fields; then, click **Set Password**.
- 4 Click **OK** to make the password changes and close the dialog.
Password changes take effect immediately; however, they do not affect users currently connected to the local frame.
Clicking **Cancel** closes the dialog without making any password changes.

- Notes:
- The remote access password is completely separate from Windows user logon passwords. Setting or clearing remote access passwords does not affect Windows user logon passwords.
 - Remote access passwords are encrypted using a one-way encryption algorithm and securely stored.
 - In order to set or clear remote access passwords using this utility, the currently logged-on user must have administrative credentials.

- Establishing a remote access password does not password-protect access via Windows Remote Desktop.

See Also • [Returning to Online Analysis \(see page 59\)](#)

5 Setting Up the Logic Analyzer

- Configuring Logic Analyzer Modules / 72
- Creating a Logic Analyzer Multi-card Set / 74
- Repartitioning a U4164A Multi-card Set / 75
- Setting Up Probes (in the online help)
- Setting the Logic Analyzer Threshold Voltage / 77
- Defining Buses and Signals / 83
- Choosing the Sampling Mode / 96
- Setting Up Symbols / 132
- Installing Licensed Hardware Upgrades / 139

Configuring Logic Analyzer Modules

A logic analyzer *module* is a logical collection of channels on a single timebase and trigger. A module can be a single *card* (see [page 614](#)) or several cards. Modules give you the flexibility to increase logic analyzer channel count by using more than one card.

Combining cards to increase channel count is done when installing cards into a *frame/chassis* (see [page 615](#) and [page 614](#)). For more information, see the [AXle based Logic and Protocol Analysis – Installation Guide](#).

CAUTION

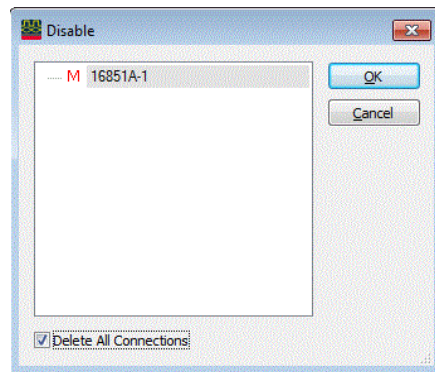
You must power down the AXle chassis before inserting, replacing, or removing the Logic Analyzer module. The enclosure surface of the module may become hot during use. If you need to remove the module, first power down the AXle chassis, wait for at least five minutes to allow the module to cool, and then pull the module out of the chassis.

- See Also
- To disable and enable modules (see [page 72](#))
 - Memory Depth and Channel Count Trade-offs (see [page 363](#))

To disable and enable modules

By default, all modules in a logic analysis system are enabled; however, you can disable modules to stop them from participating in measurements.


- To disable modules
- 1 In the window, choose **Disable...** from the module's menu.
Or, choose **Setup>(Logic Analyzer Module)>Disable...** from the main menu.
 - 2 In the Disable dialog:

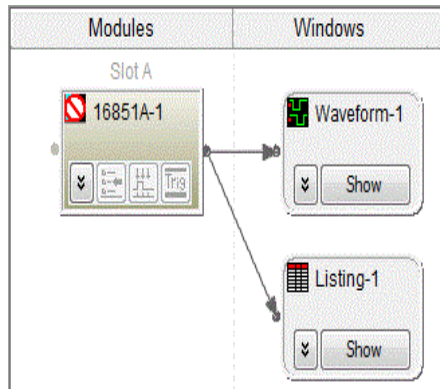


- a Select the modules you want to disable.
- b Check **Delete All Connections** if you want to delete module connections to tools and windows. This causes tools and windows connected only to the disabled modules to be deleted as well.
Uncheck **Delete All Connections** if you want to leave module connections to tools and windows. The disabled modules' buses and signals are hidden from tools and windows, but they will reappear when the modules are re-enabled.
- c Click **OK**.

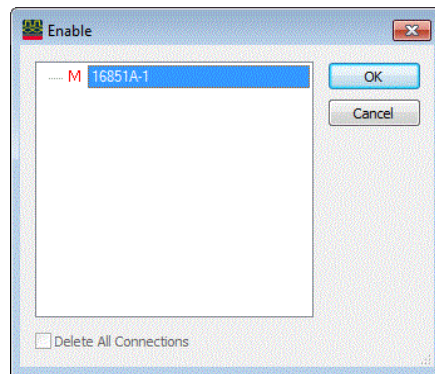
When a module is disabled:

- It no longer runs or sends data to downstream tools or windows, and no data is saved to configuration files.
- The buses and signals defined in the module are hidden from windows and tools.
- You cannot change the setup of the module.

In the window, disabled modules have the  icon:



- To enable modules
- 1 In the window, choose **Enable...** from the module's menu.
Or, choose **Setup>(Logic Analyzer Module)>Enable...** from the main menu.
 - 2 In the Enable dialog:



- a Select the modules you want to enable.
- b Click **OK**.

See Also • [Configuring Logic Analyzer Modules \(see page 72\)](#)

Creating a Logic Analyzer Multi-card Set

You can interconnect multiple modules together and then install these interconnected modules into adjoining slots of an AXIe chassis to make a multi-card set. This multi-card set is represented as a single logical instrument module in the Logic and Protocol Analyzer application. Combining modules in a multi-card set provides an increased channel count required to support various testing scenarios for different memory technologies.

The multi-card set configuration is available only for specific logic analyzer module types, for instance, U4154A, U4154B, and U4164A.

To interconnect U4164A or U4154A/B modules in a multi-card set, use a pair of U4154-61601 flex cables provided in the accessory pouch shipped with each module.



For more information on how to interconnect and install multiple modules in an AXIe chassis, see the AXIe based Logic and Protocol Analysis – Installation Guide.

NOTE

You may physically configure as many modules into a multi-card set as the chassis slots allow. However, the maximum number of interconnected modules that can operate together in a multi-card set is 3.

You may require different numbers of interconnected modules to support different memory technologies testing scenarios. For such situations, you can connect and install logic analyzer modules as a set in the chassis for larger configurations and then reconfigure these interconnected modules into smaller configurations using the Logic and Protocol Analyzer GUI. You need not physically connect/disconnect modules for such reconfigurations. See [“Repartitioning a U4164A Multi-card Set”](#) on page 75 to know more.

Once a multi-card set is configured, you can use the Instrument Details Dialog ([page 441](#)) to view the number of modules installed and the slots in which these are installed and to reconfigure the modules in the set.

See Also

[“Repartitioning a U4164A Multi-card Set”](#) on page 75

Memory Depth and Channel Count Trade-offs

Repartitioning a U4164A Multi-card Set

NOTE

You can repartition modules only in a U4164A multi-card set.

Once you have interconnected the U4164A modules into a multi-card set, you can easily perform the reconfiguration (repartitioning) of this multi-card set using the Logic and Protocol Analyzer software GUI. You need not physically disconnect and reconnect the modules for this repartitioning. This feature is specially useful when you have requirements for different number of interconnected modules for different memory technologies testing scenarios. To support these different configuration requirements, you can connect and install modules as a set in the chassis for larger configurations and then simply adjust the module partitioning boundaries into smaller/different configurations using the software GUI.

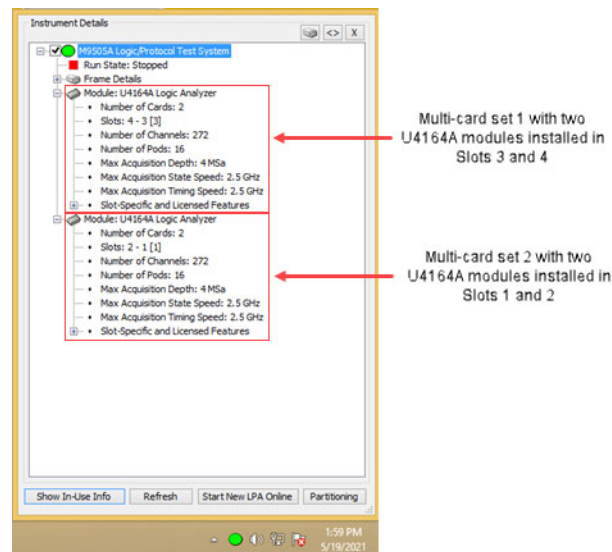
For instance, you may require a multi-card set with three U4164A modules for DDR4 DIMM testing and two multi-card sets with each set having two U4164A modules for a DDR5 DIMM testing. To support both these configurations without reconfiguring the hardware setup and cabling, you can connect four U4164A modules together when installing these in the AXIe chassis. This larger configuration would allow you to subsequently reconfigure into smaller configurations of one set with three modules and then two sets with two modules each using the software GUI.

To repartition a U4164A multi-card set

- 1 Power on the AXIe chassis. Ensure that the Logic analyzer modules are powered on but not in the Run State. The Logic and Protocol Analyzer GUI should not be active.
- 2 Use the Keysight Notification Center icon present on the Windows taskbar (lower, right-hand corner of your desktop).

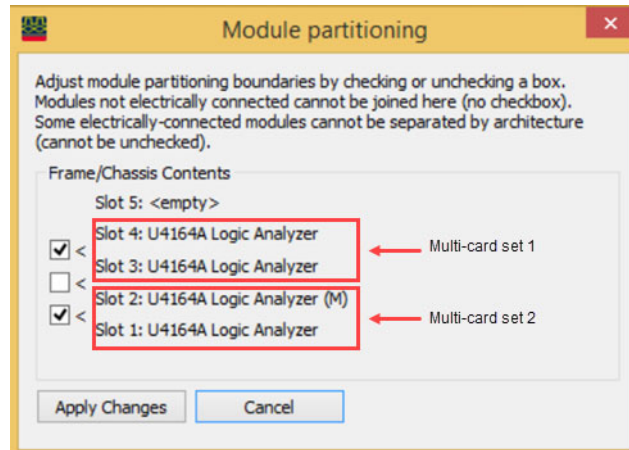


The **Instrument Details** dialog box is displayed. You can review the current configuration of modules in the multi-card set(s).

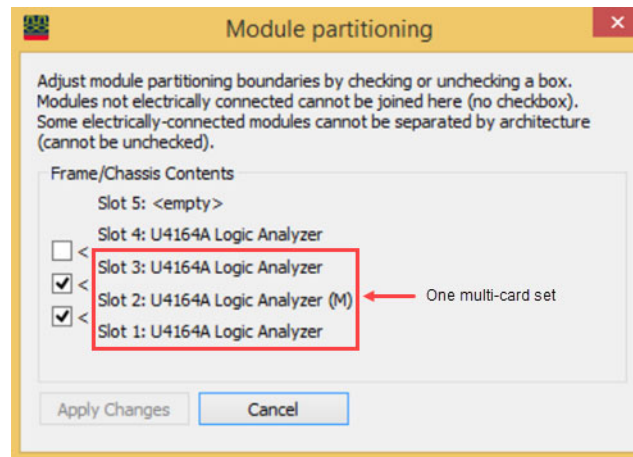


- 3 Click the **Partitioning** button to reconfigure modules in the multi-card set.

The **Module Partitioning** dialog box is displayed. The modules that you physically interconnected using flex cables are displayed with checkboxes indicating that these modules are part of the larger multi-card set and therefore can be reconfigured (included/excluded) to make smaller/alternate configurations. For example, in the screen below, four U4164A modules are interconnected in the chassis. Out of these, two multi-card sets of two modules each are configured to support a DDR5 DIMM testing scenario.



- 4 Select the checkbox displayed for a module to include it in the multi-card set and deselect the checkbox to exclude the module from the multi-card set. For example, the four interconnected modules have been reconfigured in the screen below to create one multi-card set with three U4164A modules to support a DDR4 DIMM testing scenario. The module in Slot 4 has been excluded from the set.



Setting the Logic Analyzer Threshold Voltage

It is very important that you specify a threshold voltage that matches what your device under test is using. Incorrectly specified threshold voltages result in incorrect data.

For a logic analyzer, you can set threshold voltages on a per pod basis and an independent threshold offset for each of the pod channels.

You use the Threshold Settings dialog box to set threshold voltages and offsets.

The threshold offset settings slightly differ based on the logic analyzer model you are using. The following table highlights the threshold offset settings available for each of the supported logic analyzer model.

U4154A/B or 16850-series Logic Analyzers

One threshold offset can be set for each channel in all Timing and State sampling modes.

U4164A Logic Analyzer

In Full Channel and Half Channel Timing Modes One threshold offset can be set for each channel of a pod.

Channel	Threshold	Offset	Effective Threshold
[0]	750 mV	+ 0 V	= 750 mV
[1]	750 mV	+ 0 V	= 750 mV
[2]	750 mV	+ 0 V	= 750 mV

In quarter Channel Timing Mode

Channel	Threshold	Offset	Effective Threshold
[0]	750 mV	+ 0 V	= 750 mV
[2]	750 mV	+ 0 V	= 750 mV
[4]	750 mV	+ 0 V	= 750 mV

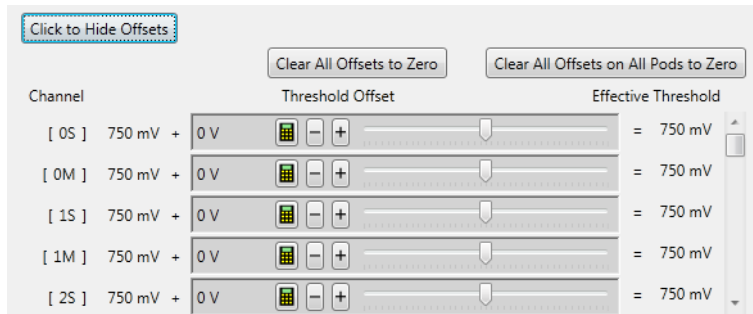
U4154A/B or 16850-series Logic Analyzers

In Master State Sampling Mode One threshold offset can be set for each channel.



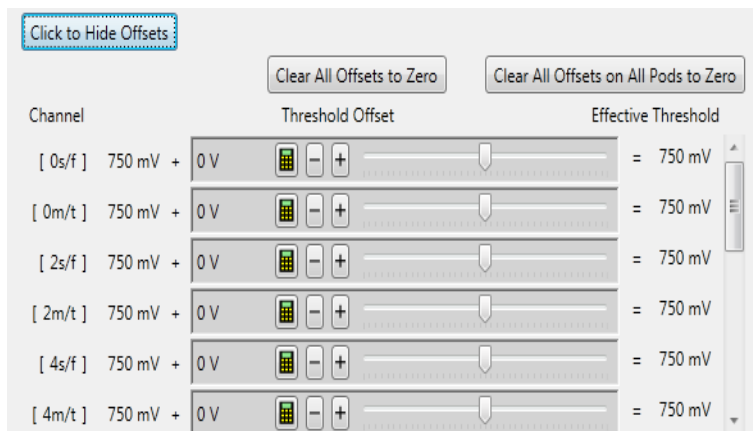
In Dual Sample State Sampling Mode

Two threshold offsets can be set for each of the channels of a pod that you set to Dual sampling. One for the Master sample and other for the Second sample. For pods not set to Dual sampling, one threshold offset can be set for each channel. For differential signals, only one threshold offset is available even in dual sampling.

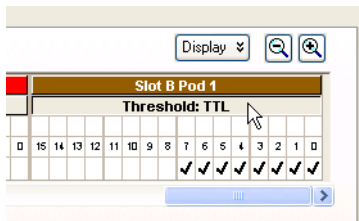


In Quad Sample State Sampling Mode

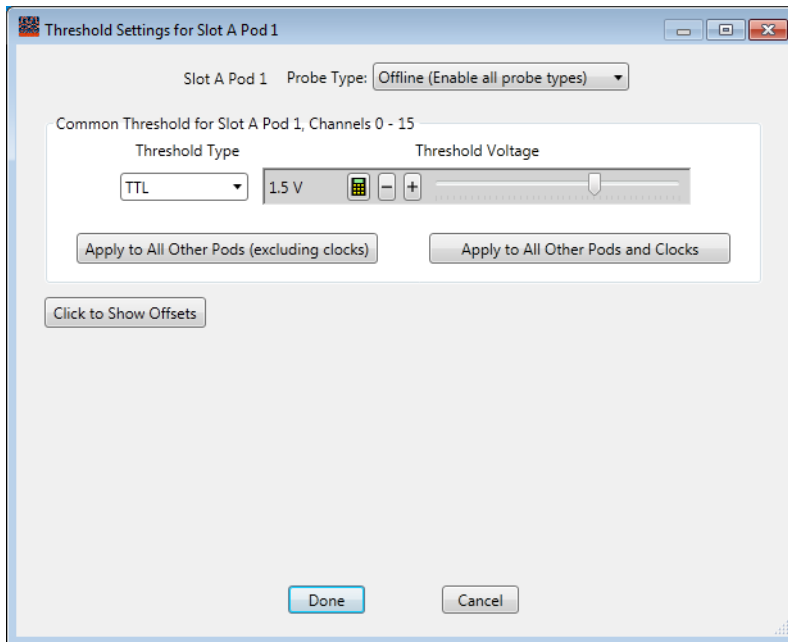
Two threshold offsets can be set for each of the Even channels of a pod that you set to Quad sampling. One for the Master and Third samples and other for the Second and Fourth samples. For pods not set to Quad sampling, one threshold offset can be set for each channel. For differential signals, only one threshold offset is available even in quad sampling.



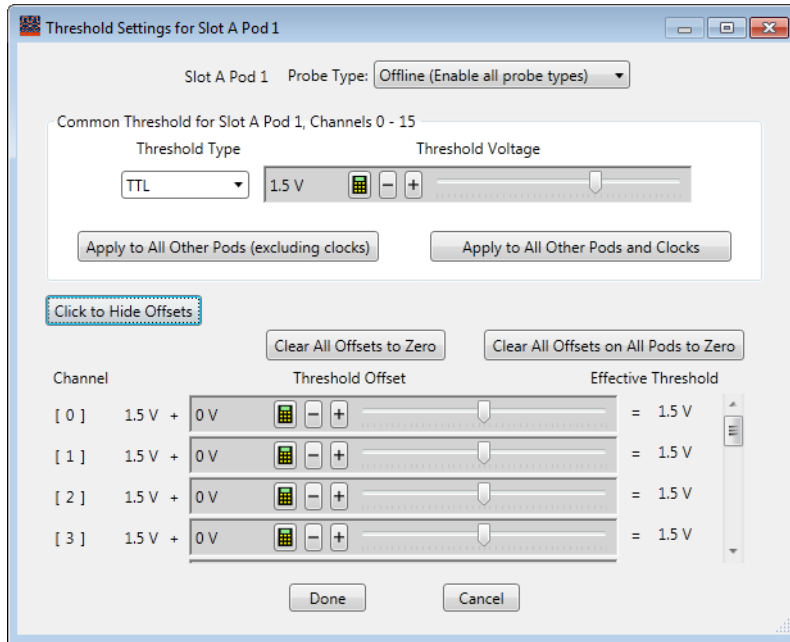
- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal....**
- 2 In the Buses/Signals Setup dialog, click any **Threshold** button. The Threshold buttons are located under the Pod or Clocks label.



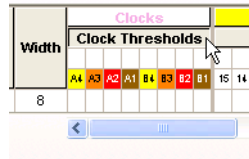
3 In the Threshold Settings dialog:



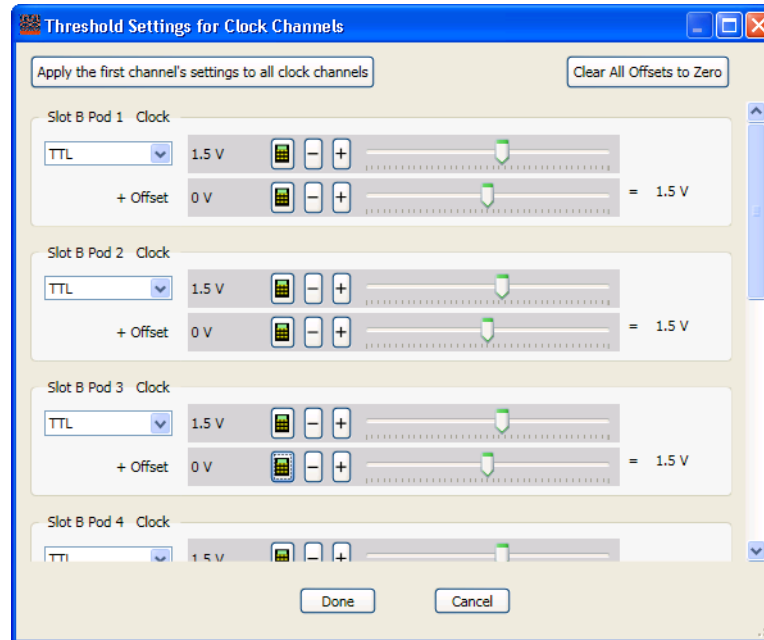
- a If offline, select the Probe Type; this may affect other settings that can be selected. When online, the currently connected probe type (if any) is shown in a read-only field.
- b Select the Common Threshold; you can do this using the Threshold Type drop-down or by dragging the slider to the desired setting. You can enter a value from -3.00 to 5.00 V.
 - Click Apply to All Other Pods (excluding clocks) if you want the common settings to apply to all pods (excluding clock inputs); otherwise, the settings apply only to the selected pod.
 - Click Apply to All Other Pods and Clocks if you want the common settings to apply to all pods and clock inputs.
- 4 If you want to specify individual channel offsets, click Click to Show Offsets, and enter the offsets or drag the sliders.



- The logic analyzer lets you specify threshold voltages for clock channels individually. This may be useful in situations, for example, where the clock channels are probing differential signals while the data channels are probing single-ended signals. Also see clock qualifiers for U4164A to know more about how threshold offsets are set for U4164A clock qualifiers.



In the Threshold Settings for Clock Channels dialog:



- Select the first clock channel's threshold using the threshold type drop-down or by dragging the slider to the desired setting. You can enter a value from -3.00 to 5.00 V.
- If you want to specify individual clock channel threshold offsets, enter the offsets or drag the sliders. (To reset all clock channel offsets, click Clear All Offsets to Zero.)
- Set up thresholds and offsets for the remaining clock channels, or click Apply the first channel's settings to all clock channels.

NOTE


In the state sampling mode, threshold voltages can be adjusted automatically (along with sample positions). See ["To automatically adjust state sampling positions and threshold voltages"](#) on page 128.

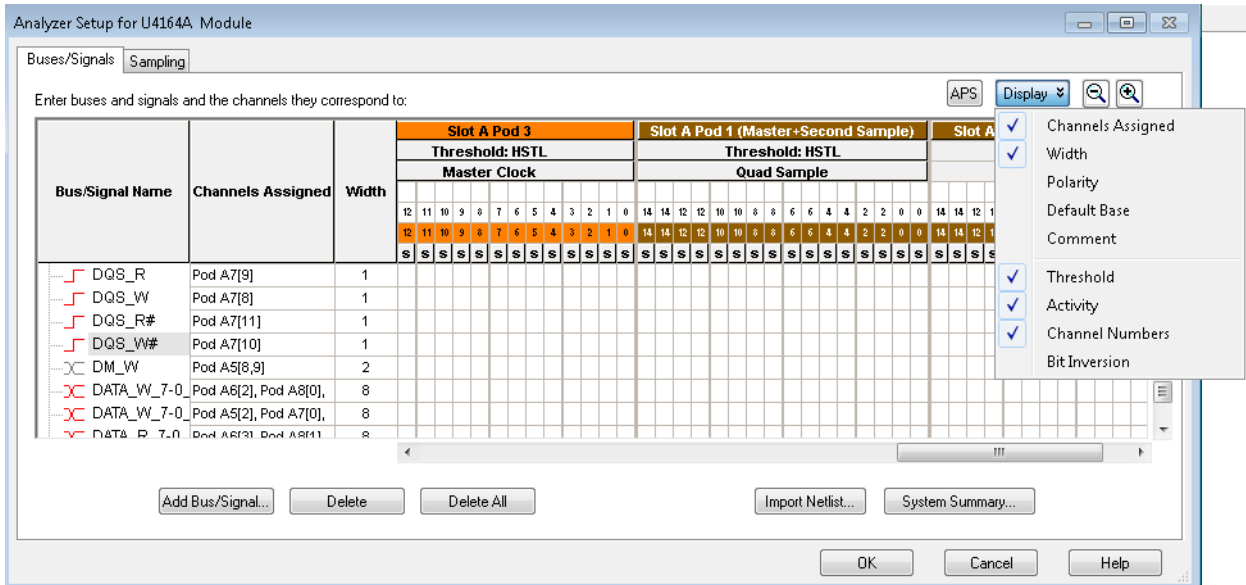
See Also • ["Pod and Channel Naming Conventions"](#) on page 361

Defining Buses and Signals

Before you can use the logic analyzer, you must define buses and signals by:

- 1 Adding bus/signal names.
- 2 Assigning logic analyzer channels to bus/signal names.

Click  in the analyzer setup toolbar, or choose **Setup>(Logic Analyzer Module)>Bus/Signal...** from the menu bar to open the Buses/Signals setup tab (see [page 426](#)).



The following tasks are performed in the Buses/Signals setup tab:

- To add a new bus or signal (see [page 84](#))
- To delete a bus or signal (see [page 84](#))
- To rename a bus or signal (see [page 85](#))
- To rename the bits of a bus (see [page 86](#))
- To assign channels in the default bit order (see [page 88](#))
- To assign channels, selecting the bit order (see [page 89](#))
- To use clock channels as extra data channels (see [page 89](#))
- To define buses and signals by importing netlist files (see [page 90](#))
- To reorder bits by editing the Channels Assigned string (see [page 91](#))
- To set the default number base (see [page 93](#))
- To set polarity (see [page 94](#))
- To add user comments (see [page 94](#))
- To add a folder (see [page 95](#))
- To alias a bus/signal name (see [page 95](#))
- To sort bus/signal names (see [page 95](#))

Through the **Display** button, you can select what bus/signal setup information is displayed (channels assigned, width, polarity, default base, comment, threshold, activity, or channel numbers).

The bus and signal icons in the **Bus/Signal Name** column are normally red, but they turn gray if the bus/signal is locked by an inverse assembler.

NOTE

In previous versions of the *Keysight Logic Analyzer* application, the Buses/Signals setup tab had a **Define Probes...** button; now, probes are defined differently (see "To define probes" (in the online help)).

See Also • Setting the Logic Analyzer Threshold Voltage (see [page 77](#))

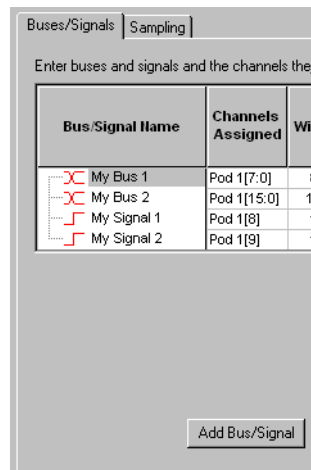
To add a new bus or signal

The add bus/signal feature allows you to add new buses and signals to the configuration. Once added to the configuration, the new bus/signal is automatically inserted into the data displays and also becomes available in any bus/signal insert function.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**
- 2 Select **Add Bus/Signal** to add a new bus or signal.
- 3 The new bus/signal will appear with a system generated default name. Rename (see [page 85](#)) the new bus/signal if desired.

NOTE

Before a new bus/signal can be added to the configuration, at least one channel must be assigned to the bus/signal.



- See Also • To delete a bus or signal (see [page 84](#))
- To rename a bus or signal (see [page 85](#))
 - To assign channels in the default bit order (see [page 88](#))
 - To assign channels, selecting the bit order (see [page 89](#))
 - Defining Buses and Signals (see [page 83](#))

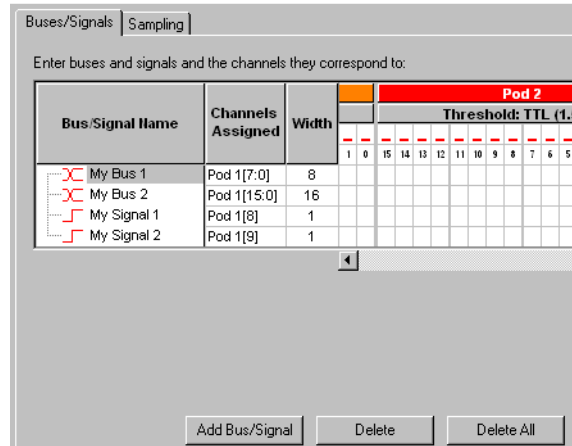
To delete a bus or signal

The delete bus or signal feature allows you to remove buses and signals individually or all at once. The delete bus or signal feature is accessed through the setup menu or the setup toolbar.

- To delete an individual bus or signal (see [page 85](#))
- To delete all buses and signals (see [page 85](#))

To delete an individual bus or signal

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal....**
- 2 Highlight the bus or signal you want to delete.
- 3 Click **Delete**.



To delete all buses and signals

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal....**
- 2 Click **Delete All**.


NOTE

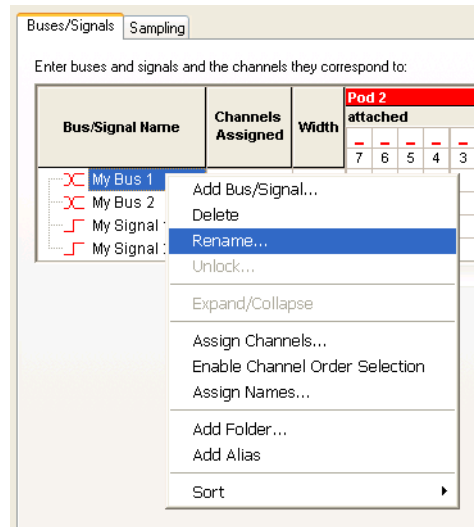
Some tools "lock" buses and signals because they use the bus or signal to produce their own output. Delete and Delete All will not delete these locked buses and signals. A locked bus or signal has a gray icon to the left of the name instead of a red icon.

See Also • Defining Buses and Signals (see [page 83](#))

To rename a bus or signal

The rename bus/signal feature allows you to change bus and signal names. All channel, pod, and clock assignments for the renamed bus/signal remain unchanged.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal....**, or click the  icon in the setup toolbar (see [page 399](#)).
- 2 Right-click the bus or signal name and choose **Rename...**




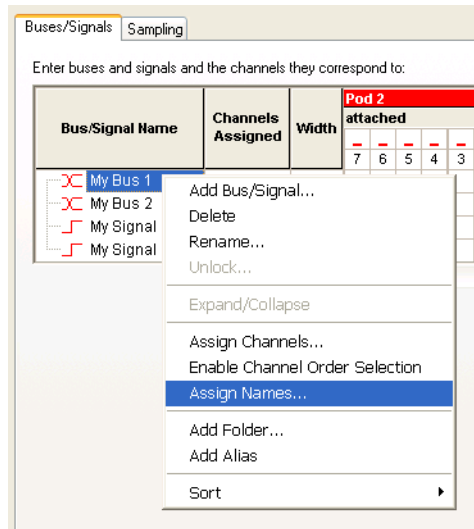
- 3 Enter the new bus or signal name.
- 4 Select **OK**.

- See Also
- To add a new bus or signal (see [page 84](#))
 - To delete a bus or signal (see [page 84](#))
 - To rename the bits of a bus (see [page 86](#))
 - Defining Buses and Signals (see [page 83](#))

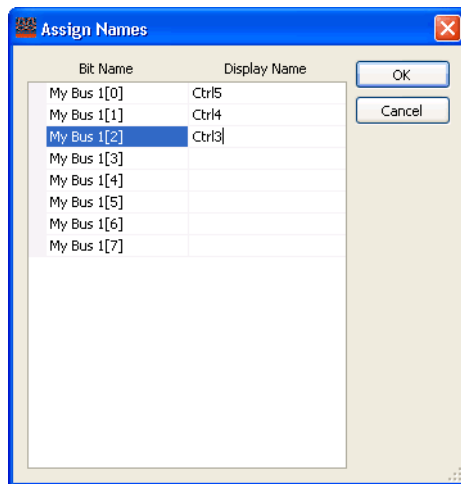
To rename the bits of a bus

When a bus is expanded in the Waveform display window, the names of the signals within the bus are like Bus[0], Bus[1], etc., by default. However, you can assign your own names to the bits within a bus.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**, or click the  icon in the setup toolbar (see [page 399](#)).
- 2 Right-click the bus name and choose **Assign Names...**

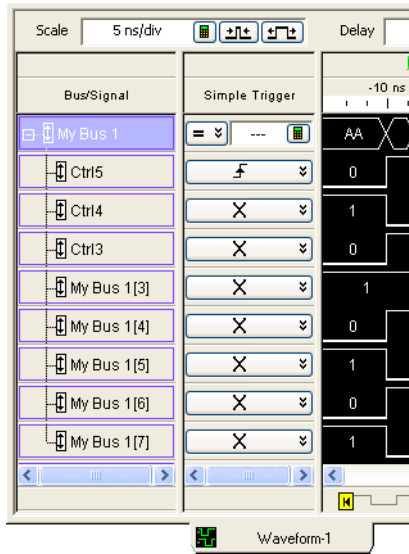


3 In the Assign Names dialog, enter the new names of bits within the bus.



4 Click OK.

Now, you see your names when the bus is expanded in the Waveform window.



- See Also
- To rename a bus or signal (see [page 85](#))
 - To add a new bus or signal (see [page 84](#))
 - Defining Buses and Signals (see [page 83](#))

To assign channels in the default bit order

To make the logic analyzer display match your system's design, assign the physical channels of the logic analyzer to bus and signal names.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal....**
- 2 In the Buses/Signals tab, select squares in the grid to assign channels to bus and signal names. For each signal probed in your device under test, you should have a black check mark mapping the channel to a pod and to a signal name in the interface.

Example: In the picture below, channels 0-7 (pod 1) are mapped to My Bus 1, channels 8-15 (pod 1) are mapped to My Bus 2, and channels 8 and 9 (pod 2) are mapped to My Signal 1 & 2, respectively.

Bus/Signal Name	Channels Assigned	Width	Pod 1	
			TTL (1.50 V)	Threshold: TTL (1.50 V)
My Bus 1	Pod 1[7:0]	8	7 6 5 4 3 2 1 0	15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0
My Bus 2	Pod 1[15:8]	8		✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓
My Signal 1	Pod 2[0]	1	✓	
My Signal 2	Pod 2[1]	1	✓	

TIP

If clock channels are not connected to clock signals, they can be used as extra data channels. Clock channels are grouped together after the last pod in the channel assignment area.

- See Also
- To define buses and signals by importing netlist files (see [page 90](#))
 - To reorder bits by editing the Channels Assigned string (see [page 91](#))
 - Pod and Channel Naming Conventions (see [page 361](#))

To assign channels, selecting the bit order

In cases where buses in the device under test haven't been probed with consecutive logic analyzer channels, you can assign channels to a bus name in a selected bit order.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**
- 2 In the Buses/Signals tab, right-click the bus name, and choose **Enable Channel Order Selection**.
- 3 Start selecting squares in the grid to assign channels from the low order bit of the bus to the high order bit.

The bit numbers are displayed as you select squares.

Bus/Signal Name	Channels Assigned	Width	Pod 2																		
			Threshold: TTL (1.50 V)																		
			1	0	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0	
<input checked="" type="checkbox"/> Default Bit Order	Pod 2[15:12,	8			✓	✓	✓						✓	✓			✓	✓			
<input checked="" type="checkbox"/> Selected Bit Order	Pod 2[1,5,0,4	8										0	1	2	3		6	4		7	5

NOTE

When you select a bit order other than the default, you can only trigger on a sample equal to (=) or not equal (!=) to some value on that bus. You lose the ability to trigger on a sample less than (<), greater than (>), less than or equal to (<=), or greater than or equal to (>=) some value, and you lose the ability to trigger on a sample "in range" or "not in range" of two values.

To reset the default bit order

The default bit order of assigned channels has higher bits on the left and lower on the right (in the Bus/Signal Setup dialog).

Bus/Signal Name	Channels Assigned	Width	Pod 2																		
			Threshold: TTL (1.50 V)																		
			1	0	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0	
<input checked="" type="checkbox"/> Default Order	Pod 2[15:12,	8			7	6	5	4						3	2		1	0			
<input checked="" type="checkbox"/> Reordered Bits	Pod 2[1,5,0,4	8										0	1	2	3		6	4		7	5

To reset to the default bit order:

- Right-click the bus name, and uncheck **Enable Channel Order Selection**.

- See Also
- To define buses and signals by importing netlist files (see [page 90](#))
 - To reorder bits by editing the Channels Assigned string (see [page 91](#))
 - Pod and Channel Naming Conventions (see [page 361](#))

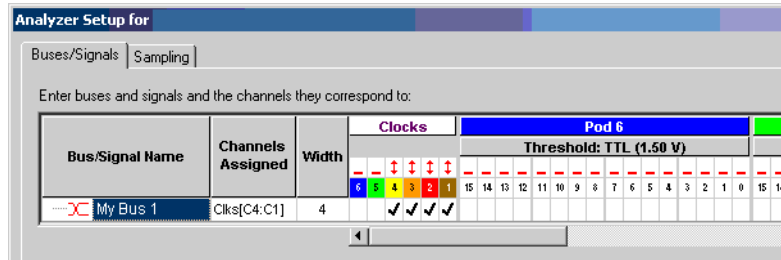
To use clock channels as extra data channels

When clock channels are not used for state mode sampling clock inputs, they can be used as extra data channels and assigned to bus/signal names just like ordinary data channels (see To assign channels in the default bit order (see [page 88](#)) or To assign channels, selecting the bit order (see [page 89](#))).

NOTE

When clock channels are used for state mode sampling clock inputs, it is not useful to assign them to bus/signal names.

Each pod in a logic analyzer module has a clock channel. In the Buses/Signals Setup dialog, all the clock channels are grouped together in a virtual **Clocks** pod. (Only ordinary data channels appear under the columns for each pod.)

**NOTE**

There is no separate physical clock pod. Each pod has a single clock channel.

The clock pod can grow to include as many channels as there are pods in the system. In other words, the clock pod can have more than 16 channels if there are more than 16 pods.

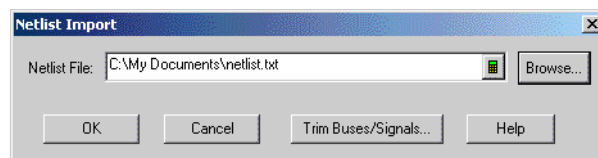
In the **Channels Assigned** string, the clock pod name is "Clks" and the channels are named "C1", "C2", etc.

- See Also
- To reorder bits by editing the Channels Assigned string (see [page 91](#))
 - Pod and Channel Naming Conventions (see [page 361](#))

To define buses and signals by importing netlist files

You can create bus/signal names and assign logic analyzer probe channels by importing netlist files. These netlist files come from the Electronic Design Automation (EDA) tools used to design the device under test, and they contain information about the signals on the connectors built into the device under test for the logic analyzer probes.

- 1 If you haven't already defined probes, choose **Setup>(Logic Analyzer Module)>New Probe>General Purpose Probe Set** and define the probes whose connectors are mapped in the netlist file (see "To define probes" (in the online help)).
- 2 In the Bus/Signal Setup dialog, click **Netlist Import...**
- 3 Then, enter the netlist file to import bus/signal assignments from, and click **OK**.



After importing netlist files, connector pin numbers are displayed in the Bus/Signal Setup dialog.

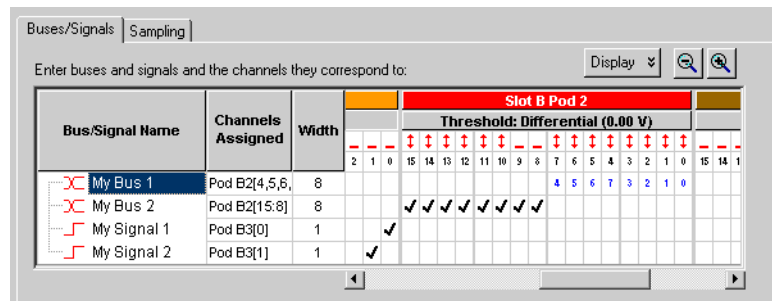
Example	Description
Pod B1[0]	Signal consisting of the first channel in the first pod of slot B.
Pod B1[15:0]	Bus consisting of all sixteen channels in Pod B1 in default order.
Pod B1[7:0], Pod B2[15:8]	Bus with seven channels from first pod in slot B followed by seven channels from second pod in slot B.
Pod B1[7:0,15:8]	Big endian, little endian switch on a 16-bit bus.
Pod B1[0,1,2,3]	Bus with bits in reverse order.
Clks[C2:C1], Pod B1[0], Pod B2[0]	Four bit bus including two clock pod channels.
Clks[C2M:C1S], Pod B1M[0], Pod B1S[0]	Four bit bus with demultiplex.

See Pod and Channel Naming Conventions (see page 361) to understand naming conventions like "Pod B2".

You can easily **Switch Big Endian** <=> **Little Endian** or **Reverse Channel Order** by clicking these buttons.

- 3 Click OK.

Channel numbers are displayed for reordered bits in the Bus/Signal Setup dialog.



NOTE

When bits have been reordered, you can only trigger on a sample equal to (=) or not equal (!=) to some value on that bus. You lose the ability to trigger on a sample less than (<), greater than (>), less than or equal to (<=), or greater than or equal to (>=) some value, and you lose the ability to trigger on a sample "in range" or "not in range" of two values.

To reset the default bit order

The default bit order of assigned channels has higher bits on the left and lower on the right (in the Bus/Signal Setup dialog).

Bus/Signal Name	Channels Assigned	Width	Pod 2																		
			Threshold: TTL (1.50 V)																		
			2	1	0	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
Default Order	Pod 2[15:12]	8				7	6	5	4					3	2			1	0		
Reordered Bits	Pod 2[1,5,0,4]	8								0	1	2	3		6	4			7	5	

To reset to the default bit order:

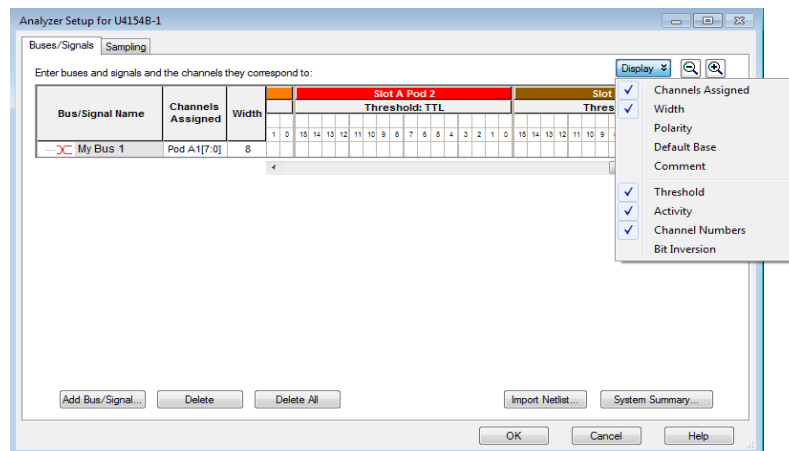
- 1 Click the **Channels Assigned** to the bus name.
- 2 In the Assign Channels dialog, click **Default Channel Order**.
- 3 Click **OK**.

- See Also
- To assign channels in the default bit order (see [page 88](#))
 - To assign channels, selecting the bit order (see [page 89](#))
 - Pod and Channel Naming Conventions (see [page 361](#))

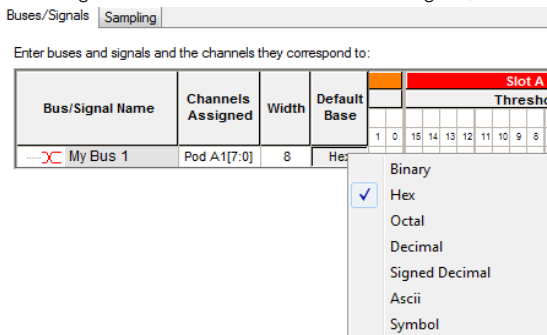
To set the default number base

You can set the default number base for a bus when you create the bus. The default base is used to display bus and signal values in the listing (see [page 241](#)) and waveform (see [page 232](#)) views. Default base only affects new buses and signals; if you change default base for an existing bus or signal you will not see a change unless you add a new copy of the bus or signal to a listing or waveform view.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal....**
- 2 In the bus/signal setup dialog, select **Display**.
- 3 Select **Default Base**.



- 4 To change the default base for a bus or signal, click the default base value.



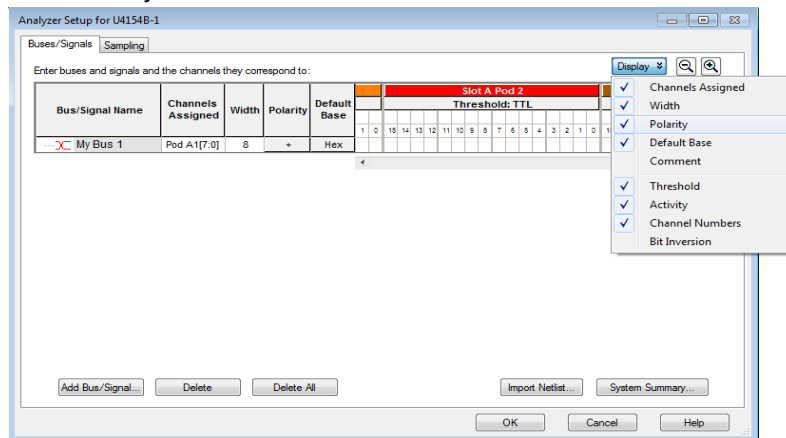
- 5 Select a new value.
- 6 Click **OK** to close the bus/signal dialog.

To set polarity

You can define buses and signals to display with negative or positive polarity. This affects the display of values and waveforms. When a bus or signal is set to negative polarity, an incoming high voltage will be shown with a low waveform and a logical value of 0. The polarity is reflected in all places that use values, such as trigger and symbols.

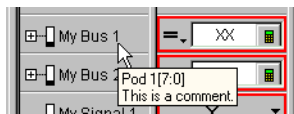
The default polarity is positive (high = 1).

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal....**
- 2 In the bus/signal setup dialog, select **Display**.
- 3 Select **Polarity**.



- 4 In the polarity column that appears, toggle between + (positive) and – (negative).

To add user comments



You can attach comments to buses and signals. The comments show up in the tool tip when you hover the mouse over a bus or signal name in both the waveform and listing windows.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal....**
- 2 In the Buses/Signals setup tab that appears, select **Display**.
- 3 Select **Comment**. A new column labeled Comment appears.
- 4 In the Comment column, type your comment for the bus or signal.
- 5 Click **OK** to close the Analyzer Setup dialog.

NOTE

Comments are intended as a descriptor to embellish a bus/signal name and not as a notepad. Comments can be up to 64 character in length.

To add a folder

The **Add Folder...** feature adds a windows style folder to the bus/signal list. Use folders to help organize bus and signal names when using many bus/signal names with inverse assemblers.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**
- 2 Right-click on a bus/signal name, then select **Add Folder**.
- 3 The new folder appears directly below the highlighted name. By default, the new folder has a system generated default name. If desired, rename (see [page 85](#)) the new folder in the same way you would a bus/signal name.

See Also · To alias a bus/signal name (see [page 95](#))

To alias a bus/signal name

The **Add Alias...** feature adds an exact duplicate bus or signal name (same channel, polarity, etc. assignments). Use alias names along with folders (see [page 95](#)) to help organize the many bus and signal names with inverse assembly.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**
- 2 Right-click on the desired bus/signal name, then select **Add Alias**.
- 3 The new alias name appears directly below the highlighted name. The new alias name can be renamed (see [page 85](#)), however, the new name will also be applied to the original name.

See Also · To add a folder (see [page 95](#))

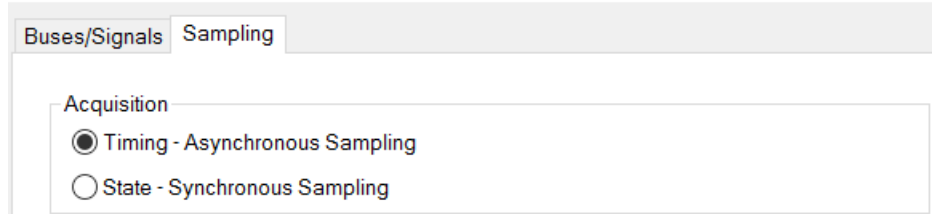
To sort bus/signal names

You can sort bus/signal names and folder names to help organize them.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**
- 2 Right-click on one of the bus/signal or folder names to be sorted; then, select either **Sort>Ascending** or **Sort>Descending**.

See Also · To add a folder (see [page 95](#))

Choosing the Sampling Mode



The Sampling tab is accessed through the menu bar's **Setup > (Logic Analyzer Module) > Timing/State (Sampling)...** item. The Sampling setup tab is used to select and configure the acquisition mode.

In the **Timing - Asynchronous Sampling** acquisition mode, you set the sampling option and the sampling period. The device under test is sampled at regular intervals (the sampling period).

In the **State - Synchronous Sampling** acquisition mode, you set the sampling option, you set up the clocking signal(s) from the device under test that tells the logic analyzer when to sample data, and you adjust sampling positions on each channel relative to the sampling clock to make sure data is sampled when it is valid.

In both state and timing mode, you can set the acquisition (memory) depth and the position of the trigger event within the acquisition memory.

Some logic analyzers have the timing zoom feature which collects additional high-speed timing data around the logic analyzer trigger.

The following tasks are performed in the Sampling setup tab.

- Selecting the Timing Mode (Asynchronous Sampling) (see [page 96](#))
 - To select the timing acquisition mode (see [page 97](#))
 - To select the timing sampling option (see [page 97](#))
 - To set the timing mode sampling period (see [page 98](#))
- Selecting the State Mode (Synchronous Sampling) (see [page 98](#))
 - To select the state acquisition mode (see [page 98](#))
 - To select the state sampling option (see [page 98](#))
 - Selecting the State Sampling Clock Mode (see [page 99](#))
 - To set up the state sampling clock (see [page 124](#))
 - To automatically adjust state sampling positions and threshold voltages (see [page 128](#))
 - To manually adjust state sampling positions (see [page 128](#))
- In Either Timing Mode or State Mode (see [page 128](#))
 - To specify the trigger position (see [page 128](#))
 - To set acquisition memory depth (see [page 129](#))
- Using Timing Zoom (see [page 129](#))
 - To turn timing zoom on or off (see [page 131](#))
 - To specify the timing zoom sample period (on some logic analyzers) (see [page 131](#))
 - To specify the timing zoom trigger position (see [page 131](#))


See Also • [Memory Depth and Channel Count Trade-offs \(see page 363\)](#)

Selecting the Timing Mode (Asynchronous Sampling)

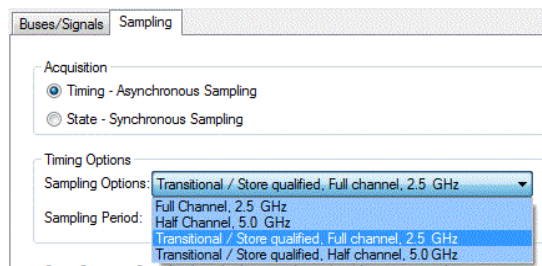
In *timing mode*, the logic analyzer samples asynchronously, based on an internally-generated sampling clock.

- To select the timing acquisition mode (see [page 97](#))
- To select the timing sampling option (see [page 97](#))
- To set the timing mode sampling period (see [page 98](#))

To select the timing acquisition mode

- 1 From the menu bar select **Setup>(Logic Analyzer Module)>Timing/State (Sampling)...**, or click the  icon from the setup toolbar (see [page 399](#)).
- 2 In the Acquisition area of the Sampling setup dialog, select the **Timing - Asynchronous Sampling** option.

To select the timing sampling option



In the timing (asynchronous) sampling mode, you can:

- Trade-off channel width for faster sampling. That is, if you want a smaller sampling period, you can set up the logic analyzer to use half of the maximum channels available.
- Get the most out of acquisition memory and measure an overall greater amount of time by choosing to store only transitions or other store-qualified patterns.

NOTE

Changing the sampling option will affect the sampling period and may affect bus assignments.

To select the timing mode sampling option:


- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>State/Timing (Sampling)...**
- 2 Select **Timing** acquisition mode. Timing Options become selectable.
- 3 Select the sampling option you prefer. Your channel count may be different depending on the logic analyzer model.

Mode	Description
<i>Full Channel Timing Mode</i>	Default. All channels are available.
<i>Half Channel Timing Mode</i>	Uses one pod from each pod pair.
<i>Quarter Channel Timing Mode</i>	Available only if you have installed the 01G or 02G Speed license option of the U4164A module. Only odd pod of each pod pair is available that is, Pod 1, 3, 5, and 7. Only even channels (0, 2, 4, 6, 8, 10, 12, 14) for each odd pod are available. Two clock channels are available per U4164A module.
<i>Transitional / Store Qualified Timing Mode</i> (see page 536)	Provides maximum duration of acquisition because data is only stored when a change from the last value is detected. See transitional timing (see page 365).

See Also • Logic Analyzer Notes, Timing Mode Sampling Options/Period (see [page 536](#))

To set the timing mode sampling period

In timing mode, a logic analyzer takes a sample of the device under test's activity once per sample period. You can set this sample period in the Sampling Setup tab.

- 1 From the menu bar select **Setup>(Logic Analyzer Module)>Timing/State (Sampling)...**, or click the  icon from the setup toolbar (see [page 399](#)).
- 2 Select **Timing - Asynchronous Sampling**.
- 3 In the Timing Mode area of the Sampling setup dialog, increase or decrease the Sample Period.

NOTE

To capture signal level changes reliably, the sample period should be less than half of the period of the fastest signal you want to measure. Time interval measurements are made by counting the number of samples in the desired waveform area. These measurements are made to a +/- one sample error, so measurement accuracy is improved if the number of samples is maximized.


See Also • Logic Analyzer Notes, Timing Mode Sampling Options/Period (see [page 536](#))

Selecting the State Mode (Synchronous Sampling)

In *state mode*, the logic analyzer samples synchronously, based on a sampling clock signal from the device under test. Typically, the signal used for sampling in state mode is a state machine or microprocessor clock signal.

- To select the state acquisition mode (see [page 98](#))
- To select the state sampling option (see [page 98](#))
- Selecting the State Sampling Clock Mode (see [page 99](#))
- To set up the state sampling clock (see [page 124](#))
- To automatically adjust state sampling positions and threshold voltages (see [page 128](#))
- To manually adjust state sampling positions (see [page 128](#))

To select the state acquisition mode

- 1 From the menu bar select **Setup>(Logic Analyzer Module)>Timing/State (Sampling)...**, or click the  icon from the setup toolbar (see [page 399](#)).
- 2 In the Acquisition area of the Sampling setup dialog, select the **State - Synchronous Sampling** option.

To select the state sampling option

NOTE

The state sampling options are only available for the U4164A logic analyzer and 16860-series logic analyzers.

In the state (synchronous) sampling mode, the sampling option specifies the speed up to which the state mode sampling clock will match input clock edges from the device under test. You can trade-off triggering and clocking capability to allow faster state mode sampling speeds.

To select the state mode sampling option:

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>State/Timing (Sampling)...**
- 2 Select the **State** acquisition mode. State Sampling Options becomes selectable.

- 3 Select the sampling option you prefer.

Mode	Description
Single Clock	Faster state mode sampling speeds are supported, but some triggering, advanced clocking features, and multiple clocks are not available.
Multiple Clocks	Advanced clocking capabilities and multiple clocks are supported, but at a slower state sampling speed.

For details and comparison between these sampling options for a particular logic analyzer model, see:

- Selecting the State Sampling Option for a U4164A Logic Analyzer
- Selecting the State Sampling Option for a 16860-Series Logic Analyzer

See Also • ["To set up the state sampling clock"](#) on page 124

Selecting the State Sampling Clock Mode

The state sampling clock inputs let signals from the device under test specify when data should be captured.

State Options - Specify when the logic analyzer should acquire samples

Sampling Options: Single Clock, Full Channel, 350 MHz to 12.5 MSps Eye Scan: Sample Positions and Thresholds...

Clock Mode: Master AND OR Clock Qualifiers

Pod:	Pod 1.4	Pod 1.3	Pod 1.2	Pod 1.1	Clock Description
Clock:	Clk4	Clk3	Clk2	Clk1	
Activity:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Master:	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	Clk1 AND (Clk2=1 OR Clk3=0)

The state sampling clock mode specifies how the clock inputs are used for sampling. The availability of these state sampling clock modes depends on the logic analyzer model and the state sampling option that you select.

- Master (see [page 100](#)) – all pods sampled on one master clock.
- Dual Sample (see [page 106](#)) – one pod of pair dual sampled on one master clock with different delays. Different (two) thresholds are used for two samples in case of the U4164A and 16860-series logic analyzers.
- Quad Sample (see [page 110](#)) – odd pod of pair sampled on one master clock but with two thresholds and independent sampling positions. Two samples for each threshold making a total of four samples. (Available only for the U4164A and 16860-series logic analyzers)
- Master/Slave (see [page 101](#)) – Master pod is sampled on master clock and slave pod is sampled on slave clock but the captured data of both slave and master clocks is saved together when the master clock occurs.
- Demux (see [page 103](#)) – Data being probed by one pod is demultiplexed into the logic analyzer memory that is normally used for two pods.

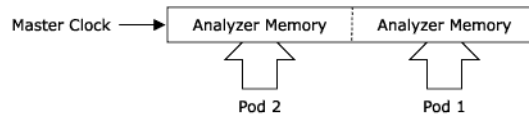
For instructions on setting up these state sampling clock modes, see:

- ["To set up the master only sampling clock mode"](#) on page 114
- ["To set up the dual sample sampling clock mode"](#) on page 116
- ["To set up the quad sample sampling clock mode"](#) on page 119
- ["To set up the master/slave sampling clock mode"](#) on page 114
- ["To set up the demultiplex sampling clock mode"](#) on page 115

- See Also
- ["To set up the state sampling clock"](#) on page 124
 - ["Pod and Channel Naming Conventions"](#) on page 361

Master Only Sampling Clock Mode

In the Master only state sampling clock mode, there is one sampling clock signal. When a clock edge occurs, data is captured and saved into one sample of logic analyzer memory.



NOTE

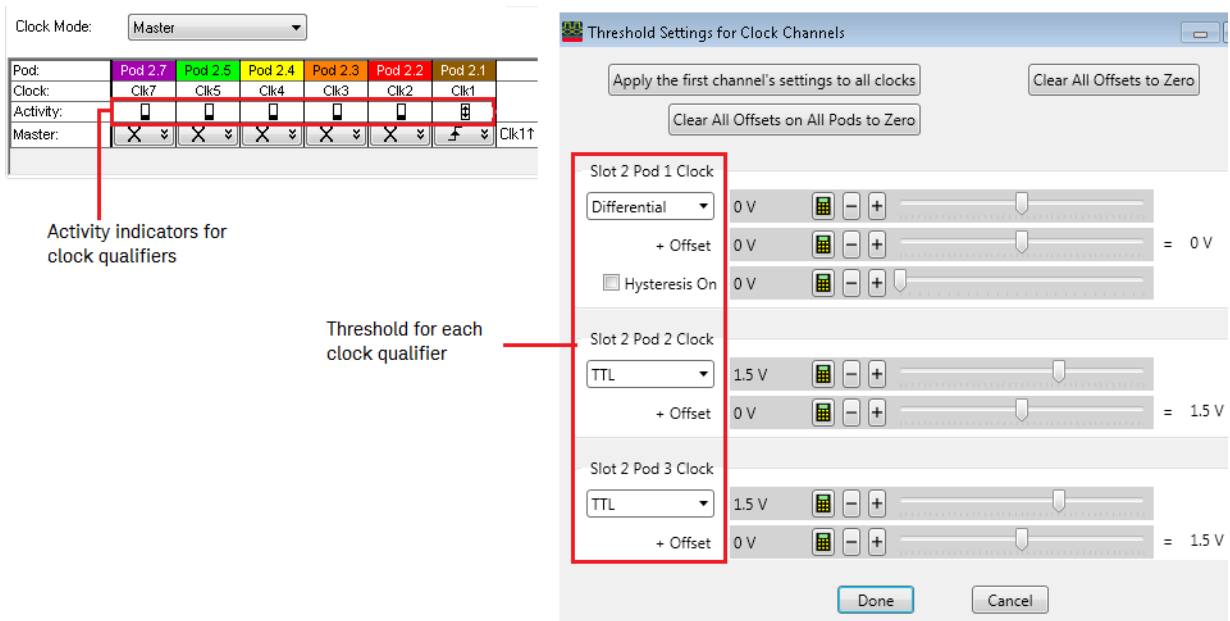
This clock mode is available in the Single Clock as well as Multiple Clocks sampling options of all currently supported logic analyzers.

When using the Master clock mode:

- You can set one master clock in the single clock sampling option and up to four master clocks (in Ored combination) in the multiple clocks sampling option.
- All pods are set to the Master clock mode.
- All logic analyzer pods and channels are available in this clock mode.
- For each channel, you can set one threshold offset.
- One sample position can be set for each channel.
- Clock channel for all pods are available and one threshold can be set for each clock channel.

Activity Indication and Thresholds for Clock Qualifiers in Master Clock Mode

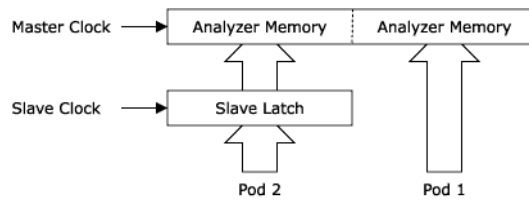
All the supported clock qualifiers are available and you can view the activity individually on these qualifiers as High or Low. You can also set individual clock thresholds for each of these clock qualifiers.



See Also · To set up the master only sampling clock mode (see [page 114](#))

Master/Slave Sampling Clock Mode

In the Master/Slave state sampling clock mode, you can save data captured on different clock edges (Master and Slave) into the same sample of logic analyzer memory.



When the slave clock occurs, data captured on the pods that use the slave clock is saved in a slave latch. Then, when the master clock occurs, data captured on the pods that use the master clock, as well as the slave latch data, are saved into logic analyzer memory. The captured data of both slave and master clocks is saved together when the master clock occurs. Therefore, the data captured by slave is not presented until the master clock occurs.

If multiple slave clocks occur before the next master clock, only the most recently acquired slave data is saved into logic analyzer memory.

NOTE

This clock mode is only available for the Multiple Clocks sampling option in the U4164A and 16860-series logic analyzers.

Example - Master/Slave Clock Mode

State Options - Specify when the logic analyzer should acquire samples

Sampling Options: Multiple Clocks, Full Channel, 350 MSps to 0 MSps

Clock Mode: Master/Slave/Demux Advanced Clocking

Pod:	Pod 1.4	Pod 1.3	Pod 1.2	Pod 1.1	
Clock:	Clk4	Clk3	Clk2	Clk1	
Activity:					
Master:		1	X	X	[(Clk4↑) AND (Clk3=1)]
Slave:	X	1	X		[(Clk1↓) AND (Clk3=1)]

Slot 1 Pod 2										Slot 1 Pod 1																					
Threshold: User 368 mV										Threshold: GTLPlus																					
Master Clock										Slave Clock																					
↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↓	↓	↓	↓	↓	↓	↓	↓	↓	↓												
15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0

In this example:

- the Master clock is set to occur on both edges of clock 4.
- The Slave clock is set to occur on falling edge of clock 1.
- These master and slave clocks have been qualified using the clock qualifier on clock 2.
- Pod 1 is set to slave clock and Pod 2 is set to Master clock.

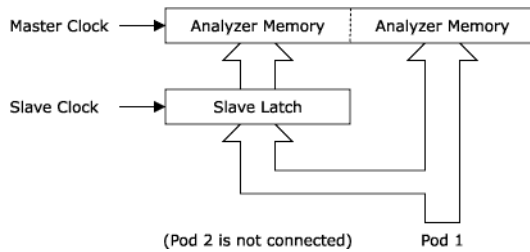
With the above settings, the data on pod 1 is captured according to the slave clock (on falling edge of clock 1). The master clock occurs on both edges of clock 4 and then data is captured on Pod 2. The master clock then saves the data captured by it as well as the slave clock to the logic analyzer memory.

When using the Master/Slave clock mode

- Up to four Ored master clocks and four Ored slave clocks can be set separately. This number may differ based on the 16860-series model you have purchased.
- You can choose the specific pod(s) of logic analyzer that you want to set as master or slave. Master/slave sampling can be combined with Demultiplex sampling on different pod-pairs.
- From the pod-pair that you set to Master and Slave sampling, both the pods in the pair are used and these pods capture data separately as per their clocks.

Slot 1 Pod 2										Slot 1 Pod 1																					
Threshold: GTLPlus										Threshold: GTLPlus																					
Master Clock										Slave Clock																					
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---												
15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0

- You can set one threshold offset for each channel.
- Clock channels for all pods (Master or Slave) are available and one threshold can be set for each clock channel.



When the slave clock occurs, data captured on the pod is saved into the slave latch for the other pod in the pod pair. Then, when the master clock occurs, data captured on the pod, as well as the slave latch data, are saved in logic analyzer memory. If multiple slave clocks occur before the next master clock, only the most recently acquired slave data is saved into logic analyzer memory.

NOTE

This clock mode is only available for the Multiple Clocks sampling option in the U4164A logic analyzer and the 16860-series logic analyzer.

Example - Demultiplex Clock Mode

State Options - Specify when the logic analyzer should acquire samples

Sampling Options: Multiple Clocks, Full Channel, 350 MSps to 0 MSps Eye Scan: Sample Position

Clock Mode: Master/Slave/Demux Advanced Clocking

Pod:	Pod 1.4	Pod 1.3	Pod 1.2	Pod 1.1	Clock Description
Clock:	Clk4	Clk3	Clk2	Clk1	
Activity:					
Master:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	$[(Clk1\phi) \text{ AND } (Clk2=0)]$
Slave:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	$[(Clk1j) \text{ AND } (Clk2=0)]$

Clocks	Slot 1 Pod 3 (Master Clock)															Slot 1 Pod 3 (Slave Clock)																
Clock Th	Threshold: Differential																															
	Demultiplex																															
	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0

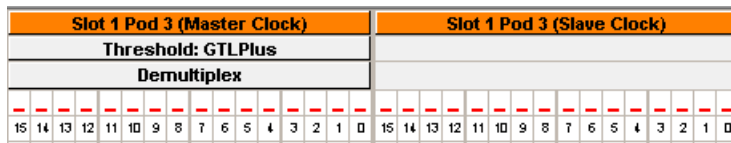
In this example:

- The Master clock is set to occur on the falling edge of clock 1.
- The Slave clock is set to occur on the rising edge of clock 1.
- These master and slave clocks have been qualified using the clock qualifier on clock 2.
- Pod 3 has been demultiplexed.

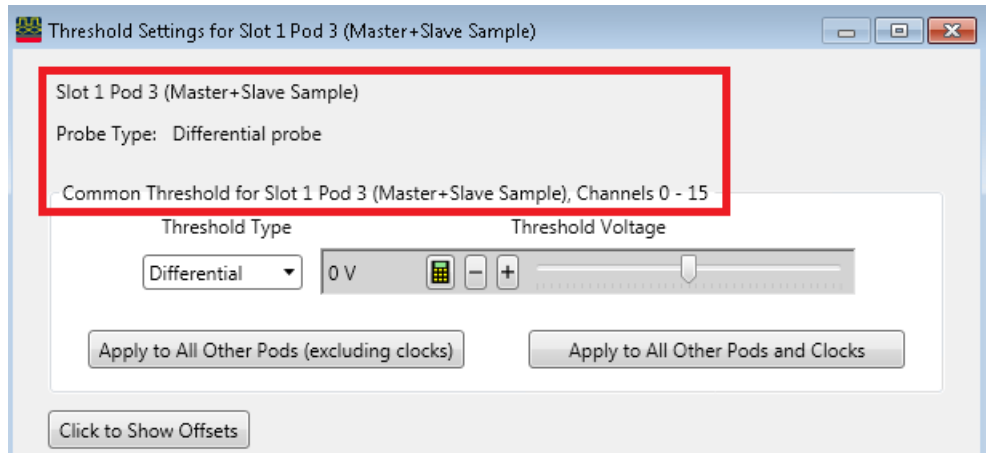
With the above settings, one set of data is captured on Pod 3 with the rising edge of the clock 1 (slave clock). This data is saved into the slave latch for the other pod (pod 4) in the pod pair. A different set of data is then captured on pod 3 with the falling edge of the clock 1 (master clock). Both these sets of data are then saved into the logic analyzer memory together. So, the pod 3 is demultiplexed by allowing 16 bits of signals to be used to capture two different sets of data on different clocks.

When using the Demultiplex clock mode

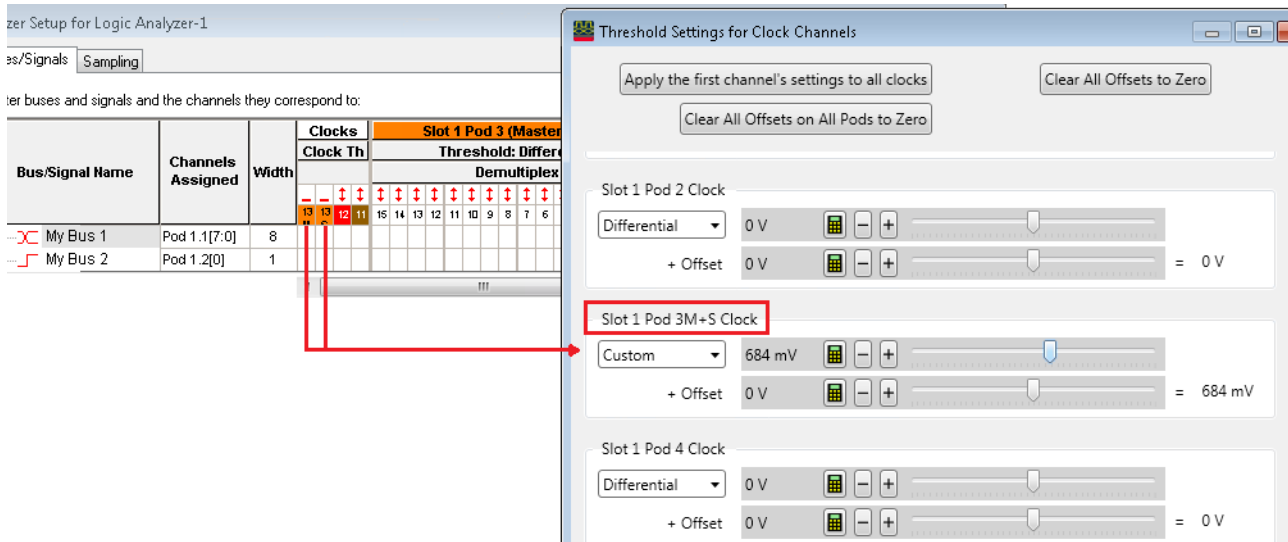
- Up to four Ored master clocks and four Ored slave clocks can be set separately. This number may differ based on the 16860-series model you have purchased.
- You can choose the specific pod pair(s) of logic analyzer that you want to set to the Demultiplex mode. From this pod-pair, the Even pod is NOT used. The Demultiplex mode allows the usage of only the Odd pod from the pod pair.



- The Demultiplex sampling can be combined with Master/Slave sampling on different pod-pairs.
- You can set one threshold offset for each data channel of the Demultiplexed Odd pod from the pod pair. The same threshold and its offsets are applied to the channels of the Even pod in this pod-pair.



- For the pod-pair that you set to Demultiplex, only the clock channel of the Odd pod is displayed and is used for both master as well as slave in the Demultiplexed pod pair. The master and slave share the commonly set clock threshold for this clock channel on the Odd pod.



Activity Indication and Thresholds for Clock Qualifiers in Demux Clock Mode

When you set a pod-pair to the Demux mode,

- the clock channel on the even pod of this pair becomes unavailable as a channel in the Buses/Signals tab but is still available as a clock qualifier and you can view the activity individually on this qualifier as High or Low in the Sampling tab. The clock threshold that you set for this clock channel on the Even pod is still used for its clock qualifier.
- the clock channel on the odd pod of this pair is available as a channel in the Buses/Signals tab and also as a clock qualifier. You can view the activity individually on this qualifier as High or Low in the Sampling tab. The clock threshold that you set for this clock channel on the Odd pod is used for the master as well as slave clock.

Clocks	Slot 1 Pod 3 (Master Clock)								Slot 1 Pod 3 (Slave Clock)																					
Clock Th	Threshold: GTLPlus																													
	Demultiplex																													
	13	12	11	10	9	8	7	6	5	4	3	2	1	0	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0

State Options - Specify when the logic analyzer should acquire samples

Sampling Options: Multiple Clocks, Full Channel, 350 MSps to 0 MSps

Clock Mode: Master/Slave/Demux Advanced Clocking

Pod:	Pod 1.4	Pod 1.3	Pod 1.2	Pod 1.1
Clock:	Clk4	Clk3	Clk2	Clk1
Activity:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Master:	0	X	X	1
Slave:	1	X	X	0

Activity indicators for the Demultiplexed pod pair clock channels

See Also • To set up the demultiplex sampling clock mode (see [page 115](#))

Dual Sample Sampling Clock Mode

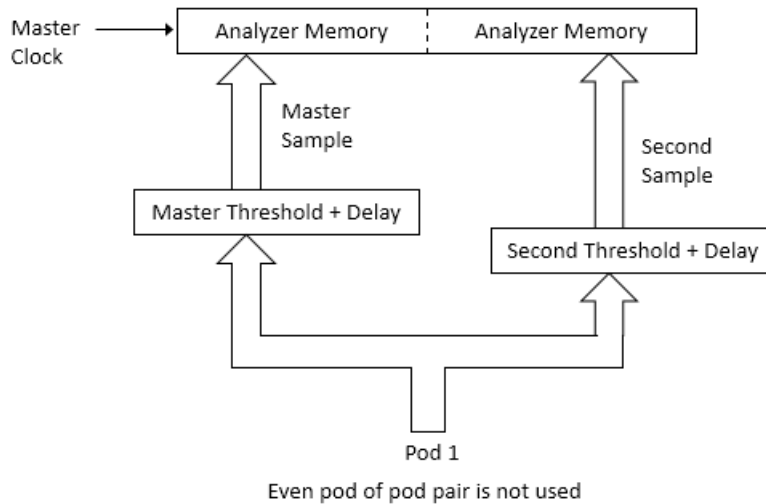
NOTE

This clock mode is available only for the Single Clock sampling option.

The topic describes and illustrates dual sampling mode as supported in different models of logic analyzers.

Dual Sample in U4164A and 16860-series Logic Analyzers

In the Dual Sample clock mode, you can capture two samples per clock edge with two different threshold offsets and separate sampling positions. These separate threshold offsets and sampling positions allows you to set independent thresholds and sampling positions for Read and Write in DDR/LPDDR captures and for Rising and Falling edge in general-purpose data captures. One threshold offset and sampling position is used for the Master sample and the other for the Second sample.

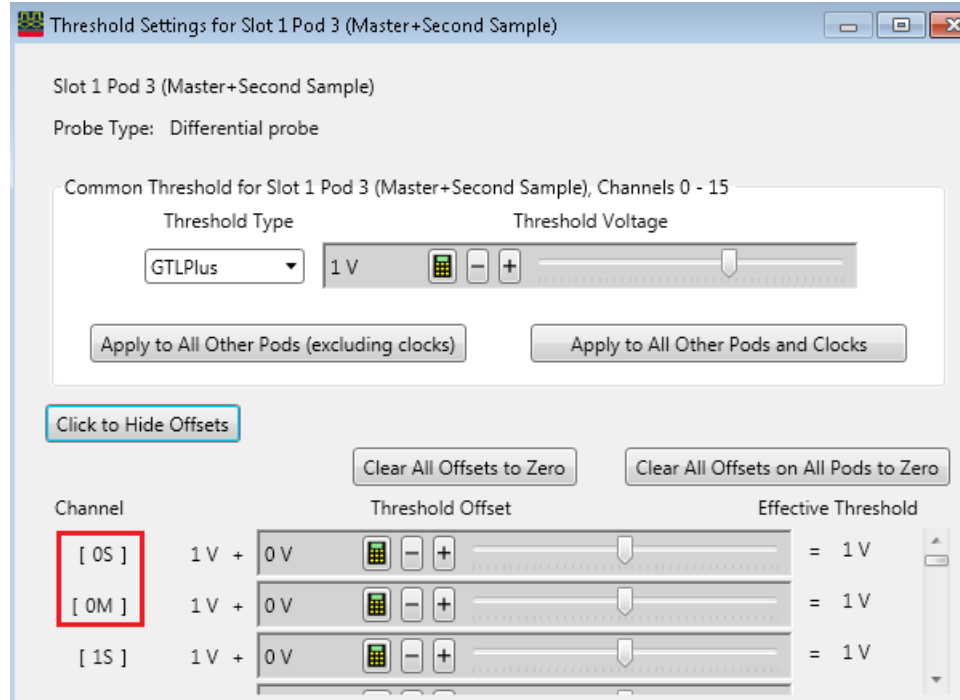


When using the Dual Sample clock mode in U4164A or 16860-series logic analyzers:

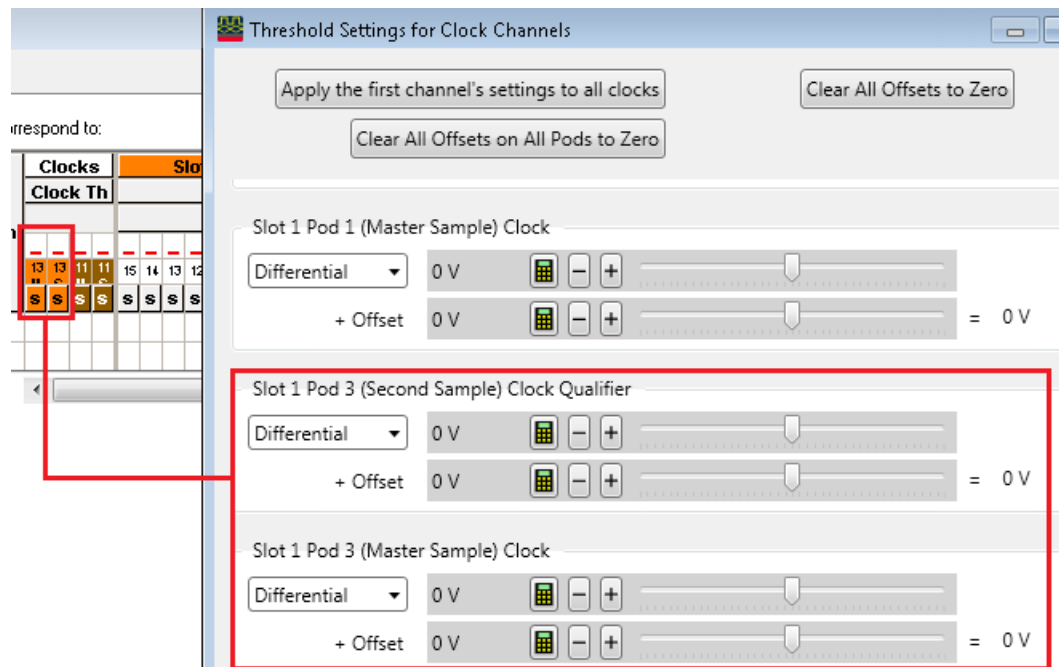
- One master clock can be set. The state sampling clock input is always from Pod1 of the logic analyzer.
- You can choose the specific pod-pair(s) of logic analyzer that you want to set as dual-sampled. Dual sampling is set for individual pod-pairs and can be combined with Master only pod-pairs.
- From the pod-pair that you set to Dual sampling, only the Odd pod of logic analyzer is used. The Even pod is not used.

Slot 1 Pod 3 (Master Sample)																Slot 1 Pod 3 (Second Sample)															
Threshold: GTLPlus																															
Dual Sample																															
15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	

- For each single-ended dual-sampled channel, you can set two separate threshold offsets - one for the Master sample and the other for the second sample. For differential signals, you can set only one threshold offset.



- For the pod-pair that you set to dual-sample, only the clock channel of the Odd pod is displayed and is dual-sampled. You can set two threshold offsets for a single-ended dual-sampled clock channel.



- For each dual-sampled channel, you can set independent sampling positions for Master and Second sample.

Activity Indication and Thresholds for Clock Qualifiers in Dual Sample Clock Mode (Single Clock only)

When you enable dual sampling on Pod-pair 1 and 2, the Pod 2 clock qualifier becomes unavailable as a channel and its activity cannot be seen in the Sampling tab. Similarly, when you enable dual sampling on Pod-pair 3 and 4, the Pod 4 clock qualifier becomes unavailable as a channel and its activity cannot be seen in the Sampling tab. You cannot then set the clock thresholds for these two clock qualifiers. In such a situation, the clock threshold that you set for Pod 3 clock qualifier is then used automatically for Pod 2 and Pod 4 clock qualifiers. This way Pod 2 and Pod 4 clock qualifiers can still be used as clock qualifiers even though the activity on these qualifiers cannot be seen and threshold cannot be set individually for these.

State Options - Specify when the logic analyzer should acquire samples

Sampling Options: Single Clock, Full Channel, 350 MHz to 12.5 MSps

Clock Mode: Dual Sample

Pod:	Pod 1.4	Pod 1.3	Pod 1.2	Pod 1.1
Clock:	Clk4	Clk3	Clk2	Clk1
Activity:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Master:	X	X	X	f Clk1j

No activity indication on Pod 2 and Pod 4 clock qualifiers when pods are set to dual sampling

- Clock threshold for Pod 3 clock qualifier.
- Also used for Pod 2 and Pod 4 clock qualifiers when these pods are set to dual sampling.

Threshold Settings for Clock Channels

Apply the first channel's settings to all clocks

Clear All Offsets to Zero

Clear All Offsets on All Pods to Zero

Slot 1 Pod 1 (Master Sample) Clock

Differential 0 V

+ Offset 0 V = 0 V

Slot 1 Pod 3 (Second Sample) Clock Qualifier

Differential 0 V

+ Offset 0 V = 0 V

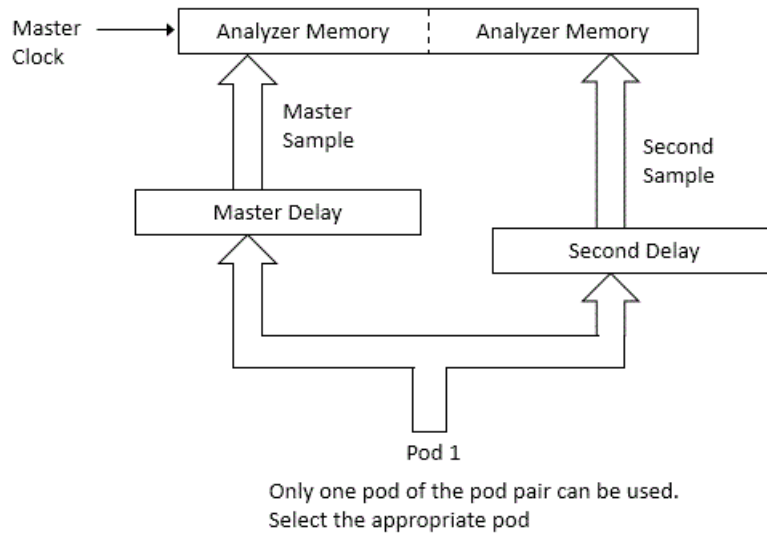
Slot 1 Pod 3 (Master Sample) Clock

Differential 0 V

+ Offset 0 V = 0 V

Dual Sample in U4154A/B and 16850-series Logic Analyzers

In the Dual Sample state sampling clock mode, you can save data captured using the master clock at each of two different sample times into the same sample of analyzer memory. Unlike U4164A, two threshold offsets are not supported for a dual-sampled channel. You can set only a single threshold offset for each of dual-sampled channels. When the master clock occurs, data on the pod is sampled twice using two independent sampling positions. Each of the two sample positions can be set using the Eye-Scan - Thresholds and Sample Positions dialog.



Dual sampling in the U4154A/B or 16850-series logic analyzers allows you to set either odd or even pod from the pod-pair as the dual-sampled pod. The other pod in the pair is not used.

The Dual Sample mode is often used to capture DDR memory bus activity using the common bus clock as the master clock. One sample position is used to capture write data and another is used to capture read data.

See Also · [“To set up the dual sample sampling clock mode”](#) on page 116

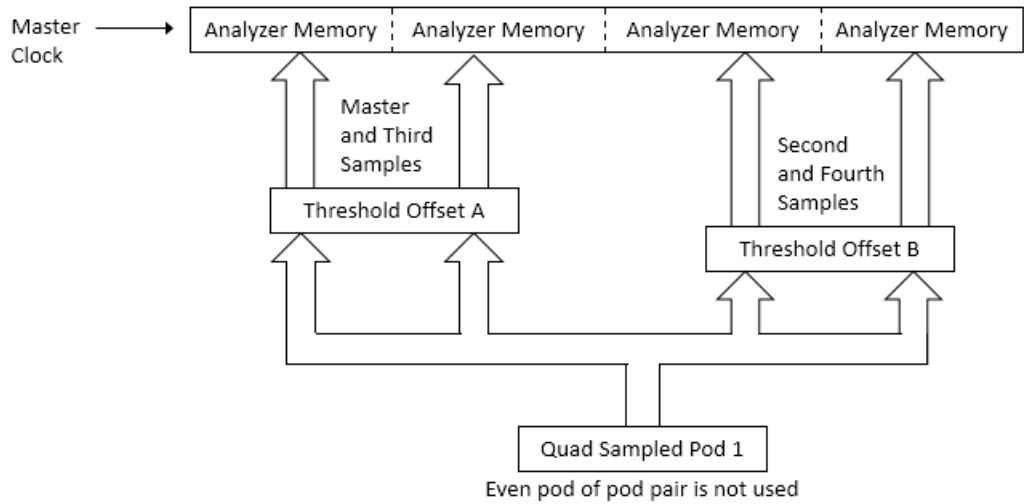
Quad Sample Sampling Clock Mode

NOTE

This mode is only available for the U4164A module with O2G license installed and state sampling set to Single Clock.

In the Quad Sample state sampling clock mode, you can capture the following four samples per clock edge with two different threshold offsets (one threshold offset for the Master and Third samples and the other for the Second and Fourth samples).

- Read rising clock, Read falling clock
- Write rising clock, Write falling clock



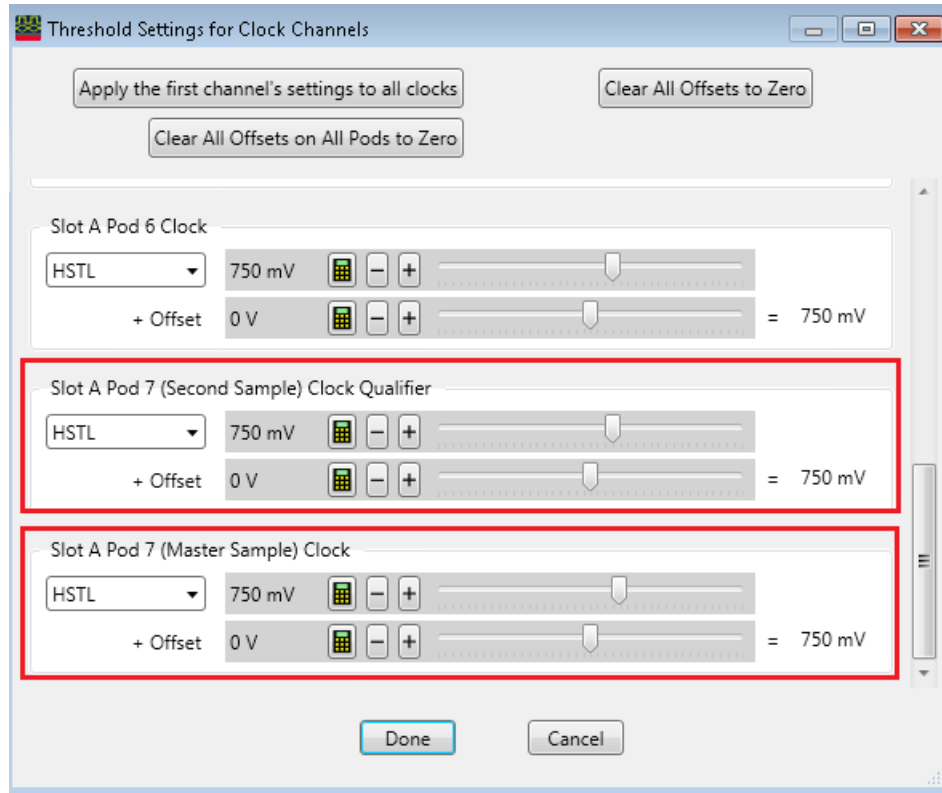
The Quad Sample mode is used to provide separate read rising, read falling, write rising, and write falling edge captures for data rates up to 4 Gb/s from a single probe point. This mode is suitable for DDR/LPDDR data captures with data rates greater than 2500 Mb/s from a single probe load placed on each DQ.

When using the Quad Sample clock mode

- You can set up a single clock. The state sampling clock input is always from Pod1 of the logic analyzer.
- You can choose the specific pod-pair(s) of logic analyzer that you want to set as quad-sampled. Quad sampling is set for individual pod-pairs and can be combined with Master or Quad-sampled pod-pairs.
- From the pod-pair that you set to Quad sampling, only the Odd pod of logic analyzer is used. Even pod is not used. Only even channels (0, 2, 4, 6, 8, 10, 12, 14) for each quad-sampled pod are available.

Slot A Pod 1 (Master+Second Sample)														Slot A Pod 1 (Third+Fourth Sample)																	
Threshold: HSTL																															
Quad Sample																															
14	14	12	12	10	10	8	8	6	6	4	4	2	2	0	0	14	14	12	12	10	10	8	8	6	6	4	4	2	2	0	0
14	14	12	12	10	10	8	8	6	6	4	4	2	2	0	0	14	14	12	12	10	10	8	8	6	6	4	4	2	2	0	0

You can set two separate threshold offsets - one for the Read (Read rising and Read falling) and the other for Write (Write rising and Write falling). This makes two samples per threshold that you can set.



- You can set independent sampling positions for Read rising, Read falling, Write rising, and Write falling edge captures.

Activity Indication and Thresholds for Clock Qualifiers in Quad Sample Clock Mode

In Quad Sample clock modes, when you enable quad sampling on Pod-pair 1 and 2, the Pod 2 clock qualifier becomes unavailable as a channel and its activity cannot be seen in the Sampling tab. Similarly, when you enable quad sampling on Pod-pair 3 and 4, the Pod 4 clock qualifier becomes unavailable as a channel and its activity cannot be seen in the Sampling tab. You cannot then set the clock thresholds for these two clock qualifiers. In such a situation, the clock threshold that you set for Pod 3 clock qualifier is then used automatically for Pod 2 and Pod 4 clock qualifiers. This way Pod 2 and Pod 4 clock qualifiers can still be used as clock qualifiers even though the activity on these qualifiers cannot be seen and threshold cannot be set individually for these.

Clock Mode: Quad Sample

Pod:	Pod 2.7	Pod 2.5	Pod 2.4	Pod 2.3	Pod 2.2	Pod 2.1
Clock:	Clk7	Clk5	Clk4	Clk3	Clk2	Clk1
Activity:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Master:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

No activity indicator for Pod 2 and Pod 4 clock qualifiers when pods are set to Quad sampling

Clock Threshold for Pod3 clock qualifier. Also used for Pod 2 and Pod 4 clock qualifiers when these pods are set to Quad sampling.

Threshold Settings for Clock Channels

Apply the first channel's settings to all clocks

Clear All Offsets to Zero

Clear All Offsets on All Pods to Zero

Slot 2 Pod 1 (Master/Second Sample) State Sampling Clock

Differential 0 V

+ Offset 0 V = 0 V

Hysteresis On 0 V

Slot 2 Pod 3 (Second Sample) Clock Qualifier

TTL 1.5 V

+ Offset 0 V = 1.5 V

Slot 2 Pod 3 (Master Sample) Clock

TTL 1.5 V

+ Offset 0 V = 1.5 V

Slot 2 Pod 5 Clock

TTL 1.5 V

+ Offset 0 V = 1.5 V

See Also ["To set up the quad sample sampling clock mode"](#) on page 119

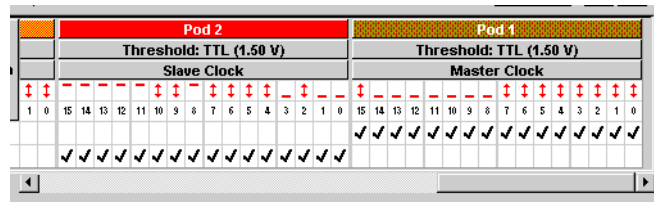
To set up the master only sampling clock mode 1

- 1 From the menu bar, select **Setup**>(Logic Analyzer Module)>Timing/State (Sampling)....
- 2 Select **State**.
- 3 In the State Options, change Clock Mode to **Master**.
- 4 Set up your master clock (see To set up the state sampling clock (see [page 124](#))).

See Also • Master Only Sampling Clock Mode (see [page 100](#))

To set up the master/slave sampling clock mode

- 1 From the menu bar, select **Setup**>(Logic Analyzer Module)>Timing/State (Sampling)....
- 2 Select **State**.
- 3 For U4164A/16860-series, select the **Multiple Clocks** option from the **Sampling Option** listbox. This listbox is displayed only for the U4164A and 16860-series logic analyzers as these support multiple clocks. The Master/slave clock mode is available only for the Multiple clocks sampling option.
- 4 From the **Clock Mode** listbox, select **Master/Slave/Demux**.
- 5 Set up your master and slave clocks (see To set up the state sampling clock (see [page 124](#))).
- 6 Select the **Buses/Signals** tab.
- 7 For each pod, click the clock button under the pod heading and choose either **Master Clock** or **Slave Clock**.



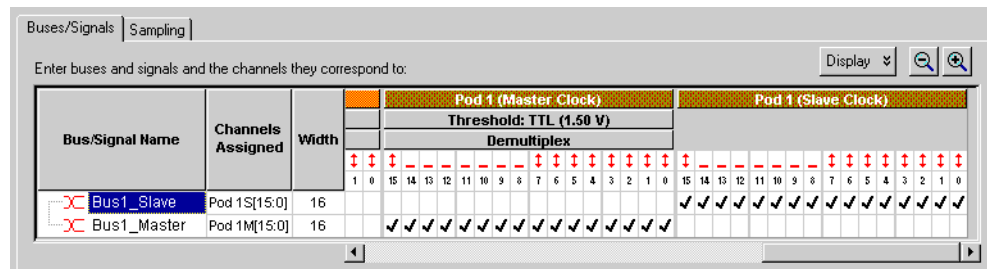
See Also • Master/Slave Sampling Clock Mode (see page 101)

To set up the demultiplex sampling clock mode

NOTE

To capture demultiplexed data, use only one pod of a pod pair.

- 1 From the menu bar, select **Setup>(Logic Analyzer Module)>Timing/State (Sampling)....**
- 2 Select the **State** acquisition mode.
- 3 For U4164A/16860-series, select the **Multiple Clocks** option from the **Sampling Option** listbox. This listbox is displayed only for the U4164A and 16860-series logic analyzers as these support multiple clocks. The demultiplex clock mode is available only for the Multiple clocks sampling option.
- 4 From the **Clock Mode** listbox, select **Master/Slave/Demux**.
- 5 Set up your master and slave clocks (see To set up the state sampling clock (see page 124)).
- 6 Select the **Buses/Signals** tab.
- 7 Click the clock button under the pod heading and choose **Demultiplex**.



The display of the pod and the other pod in the pair changes. For example, if you set Pod 1 to demultiplex, Pod 2 goes away and you see two Pod 1 columns. The first Pod 1 column is labeled Pod 1 (Master Clock), and the second column is Pod 1 (Slave Clock).

Note that you can adjust sampling positions relative to the master clock and to the slave clock by assigning master and slave bus/signal names. (Note the "M" and "S" suffixes on pods in the **Channels Assigned** column to indicate "master clock" and "slave clock".) For more information on adjusting sampling positions, see To automatically adjust state sampling positions and threshold voltages (see page 128) or To manually adjust state sampling positions (see page 128).

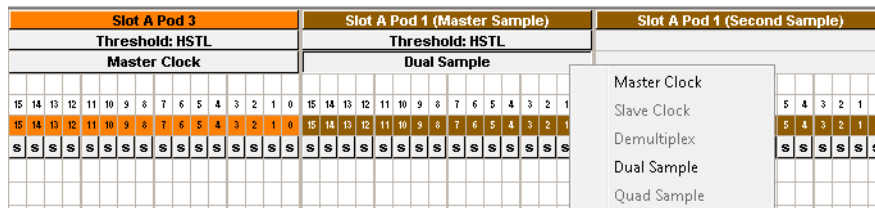
See Also • Demultiplex Sampling Clock Mode (see page 103)

NOTE**To set up the dual sample sampling clock mode**

The Dual Sample clock mode for the U4164A and 16860-series logic analyzers has some differences when compared to this mode for other logic analyzer models such as U4154A/B or 16850-series logic analyzer.

In this topic, the setting up of the dual sample clock mode is described along with these differences based on logic analyzer models.

- 1 Set the clock mode to Dual Sample
 - a From the menu bar, select Setup>(Logic Analyzer Module)>Timing/State (Sampling)....
 - b Select State - Synchronous Sampling.
 - c For U4164A/16860-series, select the Single Clock option from the Sampling Option listbox.
This listbox is displayed only for the U4164A and 16860-series logic analyzers as these support single clock as well as multiple clocks.
The Dual sample clock mode is available only for the single clock sampling option.
 - d From the Clock Mode listbox, select Dual Sample.
 - e Set up your master clock and qualifiers (see To set up the state sampling clock and State Sampling Clock and Qualifiers for U4164A).
- 2 Set the logic analyzer pod(s) as Dual-sampled on which dual sampling is required
 - a Select the Buses/Signals tab.
 - b Click the clock button displayed under the pod which you want to set as Dual-sampled and choose Dual Sample from the displayed menu. In the Dual Sample mode, a combination of dual sample and master can be set on a pod-pair basis. You can choose which pod pairs you want to set as Dual-sampled or Master.



- For U4154A/B or 16850-series logic analyzers - You can set either Odd or Even pod from a pod pair for dual sampling and accordingly the display of the pod and the other pod in the pair changes. For example, if you set Pod 2 to dual sample, Pod 1 goes away and you see two Pod 2 columns. One Pod 2 column is labeled Pod 2 (Master Sample), and the other column is Pod 2 (Second Sample).
- For U4164A or 16860-series logic analyzers - On selecting Dual Sample for any of the two pods in a pod-pair, the Odd pod out of the two pods is used for dual sampling. Only Odd pods are used for dual sampling. For example, if you set Pod 3 to dual sample, Pod 4 goes away and you see two Pod 3 columns. The first Pod 3 column is labeled Pod 3 (Master Sample), and the second column is Pod 3 (Second Sample).

NOTE

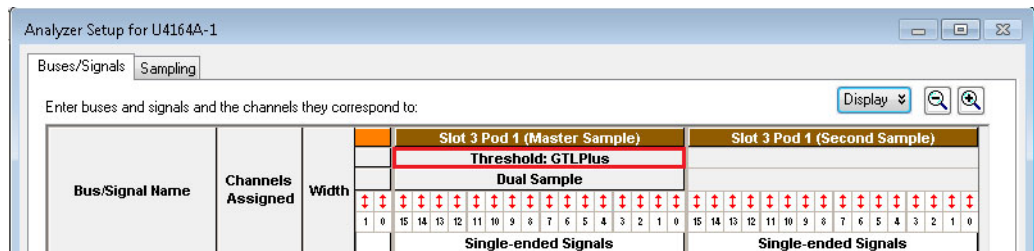
Note that timing zoom data is only valid for the selected pod.

- 3 Assign appropriate bus/signal names to the channels in the two samples - master and second

You can give names to clearly reflect Read and Write samples for DDR/LPDDR captures or Rising and Falling edge samples for general-purpose data captures. Note the "M" and "S" suffixes on dual-sampled pods in the Channels Assigned column to indicate "master sample" and "second sample" respectively.

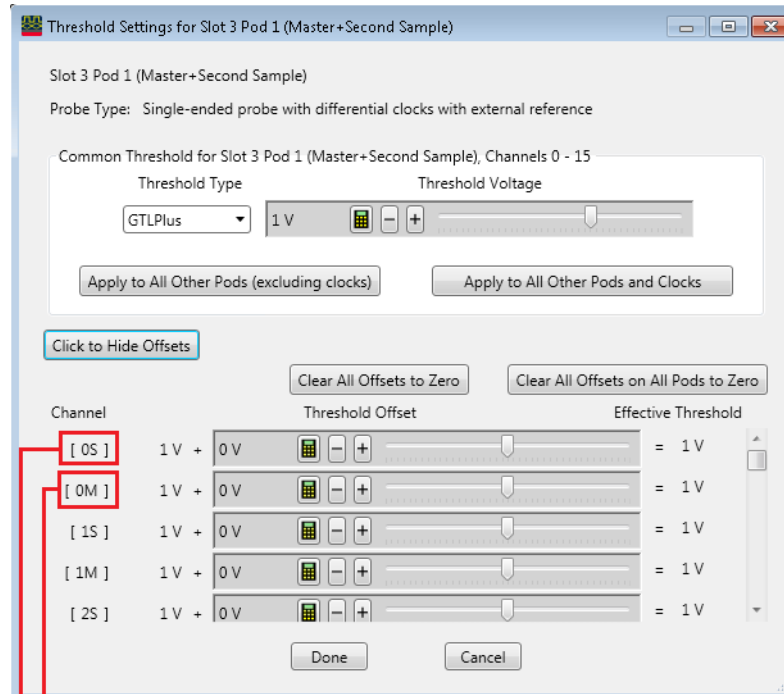
Bus/Signal Name	Channels Assigned	Width
ALERT#	Pod A1S[12]	1
C	Pod A3S[14], Pod A1S[13], Pod A3S[12]	3
ODT	Pod A3S[15]	1
STAT	Pod A3S[4], Clks[C3S], Pod A1S[13], Pod A3S[12], Pod A1S[15:14,11]	14
DATA_R	Pod A6[3], Pod A7M[1], Pod A6[1], Pod A7M[3], Pod A6[5], Pod A7M[16
DATA_W	Pod A6[6,2], Pod A7M[0], Pod A6[0], Pod A7M[2], Pod A6[4], Pod A7	16

- 4 Set the threshold offsets for the dual-sampled channels
 (This step is applicable only for dual sampling in the U4164A or 16860-series logic analyzers.)
 In U4164A/16860-series, you can set two separate threshold offsets for each dual-sampled channel - one for the Master sample and the other for the second sample. This enables you to have separate thresholds for Read and Write in DDR/LPDDR captures and separate thresholds for Rising and Falling edge in general-purpose data captures. Other logic analyzer modules such as U4154A/B and 16850-series analyzers provide only one threshold and offset per dual-sampled channel.
 You can either use the Eye Scan - Sample Position and Threshold Settings dialog box or the Threshold Settings dialog box to set the thresholds. (see To automatically adjust state sampling positions and threshold voltages or To manually adjust state sampling positions).
 - a Click the Threshold button displayed under the pod that you set as Dual-sampled.



The Threshold Settings dialog box is displayed. Click the Click to Show Offsets button. Two threshold offset setting options are then displayed for each channel of the dual-sampled pod.

NOTE The differential signals for which you have set the clock mode to Dual sample have only one threshold offset instead of two threshold offsets as available for single-ended signals.



Threshold offset for master sample on Channel 0 of Pod 1

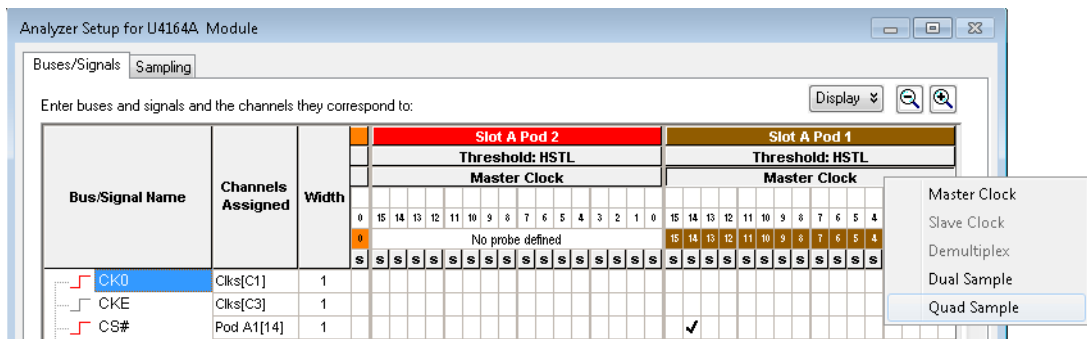
Threshold offset for second sample on Channel 0 of Pod 1

- b For dual-sampled channels, the Threshold Voltage range available in the above dialog box is from -4 to +4 V unlike the threshold voltage range of -5 to +5 V for Master sampled channels.
 - c Set the threshold offsets for the channels and click Done.
- 5 Adjust the state sampling positions independently for the two samples
- This allows you to set independent sampling positions for Read and Write in DDR/LPDDR captures and for Rising and Falling edge in general-purpose data captures. You can set separate sampling positions for the two sampled.
- You use the Eye Scan - Sample Position and Threshold Settings dialog box to set the sampling positions. (see To automatically adjust state sampling positions and threshold voltages or To manually adjust state sampling positions).
- 6 Set threshold offsets for clock channels of dual-sampled pods
- For U4154A/B and 16850-series logic analyzers
- You can set threshold offsets for clock channels of all the pods - dual sampled or not. Each clock channel has a single threshold offset.
- For U4164A and 16860-series logic analyzers
- The pods that you set as dual-sampled have their associated clock channels set to dual thresholds (dual-sampled). Only the clock channel of the odd pod in a dual-sampled pod pair is available. You can set two threshold offsets for this clock channel.
- Notice that in the following screen, the clock channel of pod1 is available from the dual-sampled pod pair (pod 1 and pod 2) and this clock channel is dual-sampled.

- 1 Set the clock mode to Quad Sample
 - a From the menu bar, select Setup > (Logic Analyzer Module) > Timing/State (Sampling)....
 - b Select State - Synchronous Sampling.
 - c Select the Single Clock option from the Sampling Option listbox.

This listbox is displayed only for the U4164A and 16860-series logic analyzers as these support single clock as well as multiple clocks.

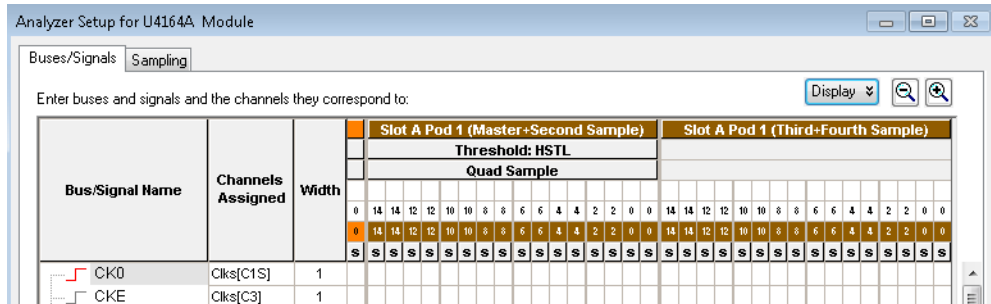
The Quad sample clock mode is available only for the single clock sampling option.
 - d From the Clock Mode listbox, select Quad Sample.
 - e Set up your master clock and qualifiers (see topics To set up the state sampling clock and State Sampling Clock and Qualifiers for U4164A).
- 2 Set the logic analyzer pod(s) as Quad-sampled on which quad sampling is required
 - a Select the Buses/Signals tab.
 - b Click the clock button displayed under the pod pair which you want to set as Quad-sampled and choose Quad Sample from the displayed menu. In the Quad Sample mode, a combination of quad sample, dual sample, and master can be set on a pod-pair basis. You can choose which pod pairs you want to set as Quad-sampled, Dual-sampled, or Master.



On selecting Quad Sample for a pod, the display of the pod and the other pod in the pair changes. For example, if you set Pod 1 to quad sample, Pod 2 goes away and you see two Pod 1 columns. The first Pod 1 column is labeled Pod 1 (Master + Second Sample), and the second column is Pod 1 (Third + Fourth Sample). Only even channels (0, 2, 4, 6, 8, 10, 12, 14) for each quad-sampled pod are available.

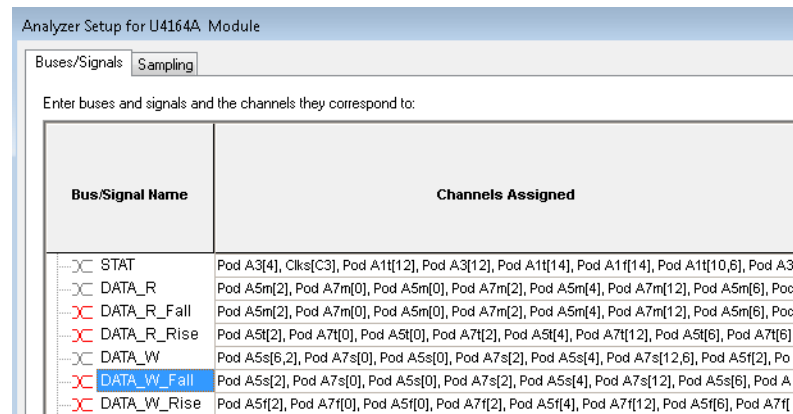
NOTE

Irrespective of the pod that you select from a pod pair for quad sampling, only the Odd pod from the pair is used. Therefore, only pod 1, 3, 5, and 7 are available for quad sampling.



- 3 Assign appropriate bus/signal names to the channels in the four samples - master, second, third, and fourth.

You can give names to clearly reflect Read rising, Read falling, Write rising, and Write falling edge captures for data signals. Note the "m" "s", "t", and "f" suffixes on quad-sampled pods in the Channels Assigned column to indicate "master sample", "second sample", "third sample" and "fourth sample" respectively.

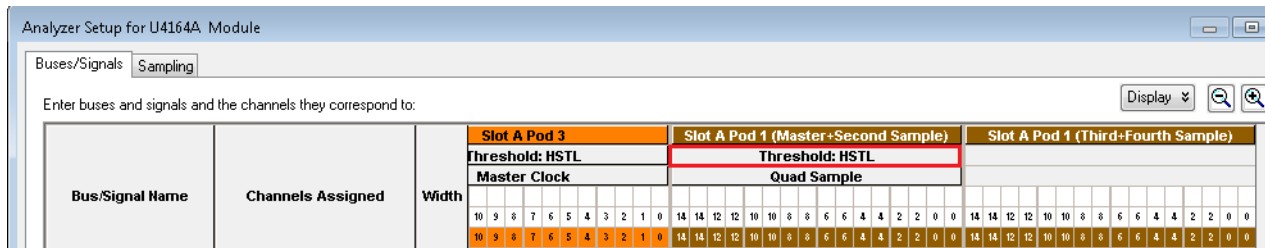


- 4 Set the threshold offsets for the quad-sampled channels

For each quad-sampled channel, you can set two separate threshold offsets - one for the Master and Third samples and the other for the second and fourth samples. This enables you to have separate thresholds for Read and Write with one threshold for Read Rising and Read Falling and the other threshold for Write Rising and Write Falling.

You can either use the Eye Scan - Sample Position and Threshold Settings dialog box or the Threshold Settings dialog box to set the thresholds. (see To automatically adjust state sampling positions and threshold voltages or To manually adjust state sampling positions).

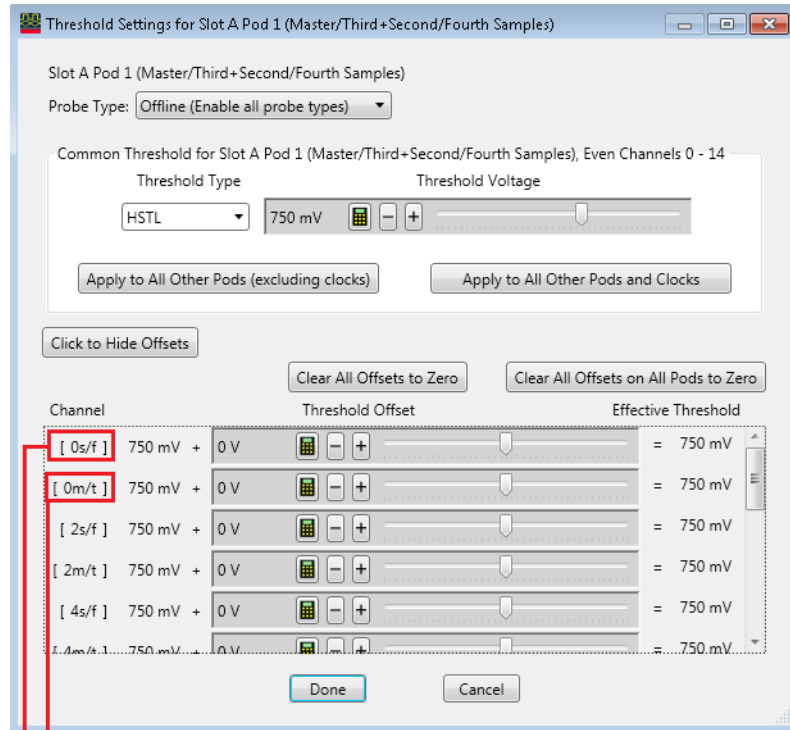
- a Click the Threshold button displayed under the pod that you set as Quad-sampled.



- b The Threshold Settings dialog box is displayed. Click the Click to Show Offsets button. Two threshold offset setting options are then displayed for each even channel of the quad-sampled pod.

NOTE

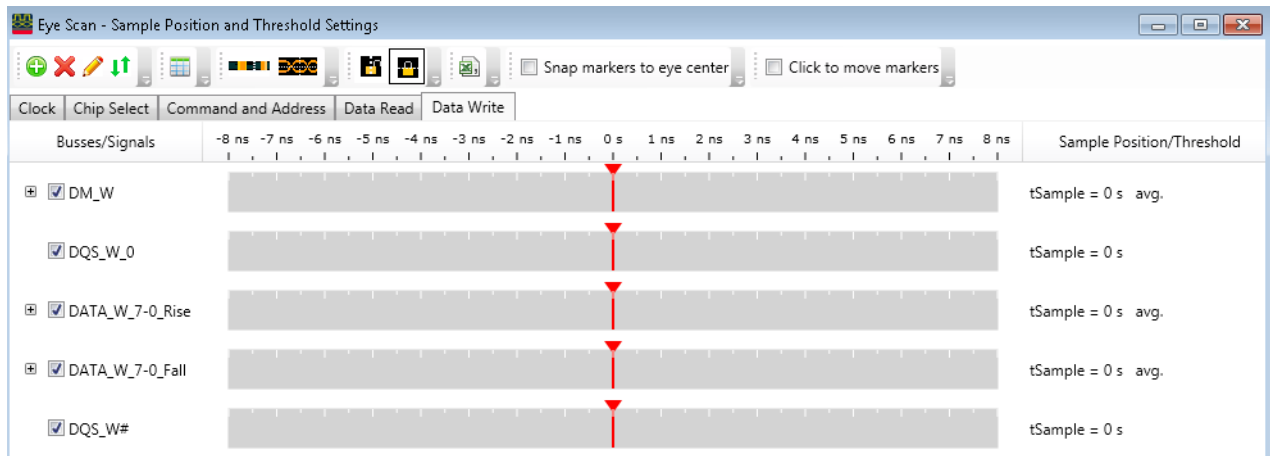
The differential signals for which you have set the clock mode to Quad sample have only one threshold offset instead of two threshold offsets as available for single-ended signals.



Threshold offset for master and third samples on Channel 0 of Pod 1
 Threshold offset for second/fourth samples on Channel 0 of Pod 1

For Quad-sampled channels, the Threshold Voltage range available in the above dialog box is from -4 to +4 V unlike the threshold voltage range of -5 to +5 V for Master sampled channels.

- c. Set the threshold offsets for the channels and click Done.
- 5 Adjust the state sampling positions independently for the four samples
 This allows you to set independent sampling positions for Read rising, read falling, write rising, and write falling edge captures.
 You use the Eye Scan - Sample Position and Threshold Settings dialog box to set the sampling positions. (see To automatically adjust state sampling positions and threshold voltages or To manually adjust state sampling positions).



- 6 Set threshold offsets for clock channels of quad-sampled pods

The pods that you set as quad-sampled have their associated clock channel set to dual thresholds (dual-sampled). Only the clock bit of the odd pod in a quad-sampled pod pair is available. You can set two threshold offsets for this clock channel. Notice that in the following screen, the clock channel of pod1 is available from the quad-sampled pod pair (pod 1 and pod 2) and this clock channel is dual-sampled.

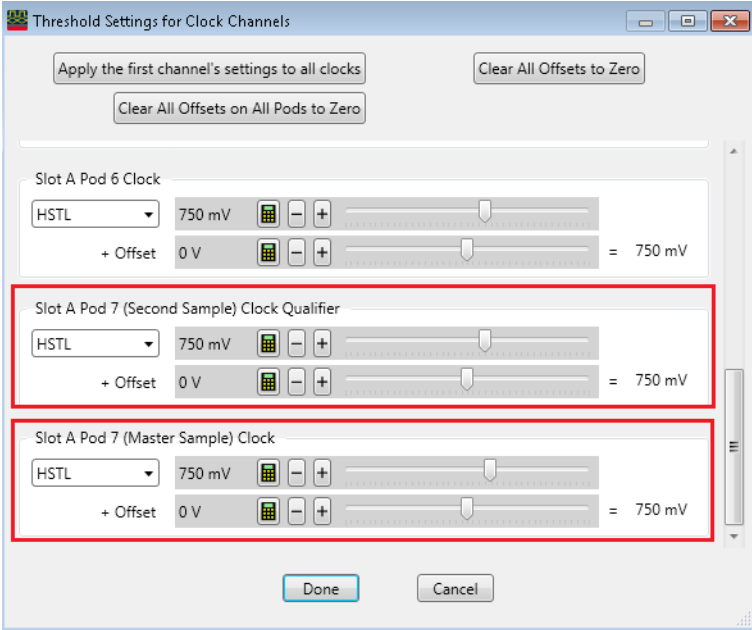
Clocks		Slot A Pod 8																Slot A Pod 7																					
Clock Thresholds		Threshold: HSTL																Threshold: HSTL																					
		Master Clock																Master Clock																					
A6	A5	A4	A3	A2	A1	A1	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0	
No probe defined																																							
S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S

NOTE

There are a few exceptions to the availability of two samples and two clock thresholds for quad-sampled clock channels. These exceptions are:

- In case of a multi-card set, the second and third U4164A modules support four samples per clock bit when all four pods are set to Quad Sample.
- If you set a quad-sampled clock channel as Differential, then this clock channel provides only one clock threshold instead of two thresholds. Differential signals do not have Dual Thresholds.

- a Click the Clock Thresholds button displayed under the Clocks column in the Buses/Signals tab.
- b The Threshold Settings for Clock Channels dialog box is displayed. For a quad-sample clock channel, you can set two clock thresholds. Note that in some cases the clock bit can be used as a clock qualifier input. The indication of Clock Qualifier tells you which of the clock thresholds will be associated with the clock qualifier input.



- c Click Done to confirm threshold settings.

- See Also
- “Quad Sample Sampling Clock Mode” on page 110
 - “State Mode Sampling Option” on page 540

To set up the state sampling clock

The state clock should be set to match the clock signal on your device under test. Some logic analyzer models support only a single clock and some support multiple clocks as well. Clocks can be as simple as a single rising edge, or a complicated combination of edges and highs or lows. The logic analyzer can handle clock signals comprised of up to four lines in case of multiple clocks.

Clock Mode: Master

Pod:	Pod 1.7	Pod 1.5	Pod 1.4	Pod 1.3	Pod 1.2	Pod 1.1	Clock Description
Clock:	Clk7	Clk5	Clk4	Clk3	Clk2	Clk1	
Activity:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Master:	0 ▾	X ▾	X ▾	1 ▾	X ▾	f ▾	Clk1↑ AND Clk3=1 AND Clk7=0

The following sections provide details on a single clock and multiple clocks for logic analyzers.

Single Clock

- In case you select the Single Clock sampling option or the logic analyzer supports only a single clock, the state sampling clock input is always from Pod1 of the logic analyzer. You cannot use the signals from other pods clock inputs as the sampling clock inputs.
- You can choose the sampling to occur on rising, falling, or both edges of the input clock signal.

State Options - Specify when the logic analyzer should acquire samples

Sampling Options: Single Clock, Full Channel, 350 MHz to 12.5 MSps ▾ Eye Scan: Sample Positions and Thresholds...

Clock Mode: Master ▾ AND OR Clock Qualifiers

Pod:	Pod 1.4	Pod 1.3	Pod 1.2	Pod 1.1	Clock Description
Clock:	Clk4	Clk3	Clk2	Clk1	
Activity:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Master:	X ▾	0 ▾	1 ▾	f ▾	Clk1↑ AND (Clk2=1 OR Clk3=0)

- The clock qualifiers available in a single clock setup can be set with the "AND"/ "OR" conditional operators with the state clock qualifiers to add conditions and completely describe the clock as a combination of edges and highs or lows.

State Options - Specify when the logic analyzer should acquire samples

Sampling Options: Single Clock, Full Channel, 350 MHz to 12.5 MSps ▾ Eye Scan: Sample Positions and Thresholds...

Clock Mode: Master ▾ AND OR Clock Qualifiers

Pod:	Pod 1.4	Pod 1.3	Pod 1.2	Pod 1.1	Clock Description
Clock:	Clk4	Clk3	Clk2	Clk1	
Activity:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Master:	1 ▾	0 ▾	1 ▾	f ▾	Clk1↑ AND (Clk2=1 OR Clk3=0 OR Clk4=1)

Multiple Clocks (applicable to U4164A and 16860-series logic analyzers only)

- If your logic analyzer model supports multiple clocks, you can select the Multiple Clocks sampling option to set multiple clocks.
- When setting multiple clocks, you can use the signals from multiple pods' clock inputs (Pod 1, 2, 3, and 4) as the sampling clock inputs. It is not restricted to pod 1 only.
- Depending on model, your logic analyzer may have more pods. However, only the clock lines on the first 4 pods can be used to generate the state clock signal. Clock lines on extra pods can be used like normal data lines.
- You can choose the sampling to occur on rising, falling, or both edges of these multiple clock input signals.
- The multiple clocks that you set are used in an Ored combination. For a master/slave clock description, an Ored master and Ored slave combination is used.

State Options - Specify when the logic analyzer should acquire samples

Sampling Options: **Multiple Clocks, Full Channel, 350 MHz to 0 MSps** Eye Scan: Sample Positions and Thresholds...

Clock Mode: **Master/Slave/Demux** Advanced Clocking

Pod:	Pod 1.4	Pod 1.3	Pod 1.2	Pod 1.1	Clock Description
Clock:	Clk4	Clk3	Clk2	Clk1	
Activity:					
Master:					Clk2↑ OR Clk4↑
Slave:					[(Clk1↓ AND (Clk2=0))]

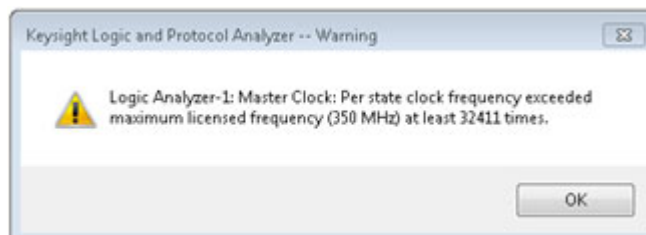
Slave clock
Multiple master clocks

NOTE

Note on Overlapping Multiple Clocks

When setting up multiple clocks, do not define overlapping ORed clocks, that is the clocks occurring on top of each other. An example of overlapping clocks is to set a clock on clock 1 on both edges and on clock 2 on both edges where the edges of clock 1 and clock 2 occur at or near the same instant in time. In such a situation, an error is displayed for such a clocking setup.

- If the input clock edges from your DUT are faster than the highest speed allowed in multiple clocks, you will get an error or a warning message. In such a situation, measurements may be available but there is a risk of some missing data. You can use the Single Clock option in such a situation.









- You can set a maximum of two qualifiers for each clock in a multiple clock description.

- You can set up complex clock description using the Advanced Clocking feature available with multiple clocks. For instance, you can use a specific clock channel both as an edge and a qualifier in the same clock description or set different qualifiers for different clocks.

To set up state sampling clock

- Attach logic analyzer pods to your device under test. Clock signals must be connected to the clock lines on pods 1 through 4.
- From the menu bar, select **Setup > (Logic Analyzer Module) > Timing/State (Sampling)...**
- Select **State - Synchronous Sampling**. The State Options area becomes active.
- Select **Single Clock** or **Multiple Clocks** from the Sampling Option listbox. Only the U4164A and 16860-series logic analyzers have this listbox as these analyzers support multiple clocks.
- Select the state clock mode (see [page 99](#)). Most measurements use only the master clock.
- Set up a clock description to match the clock(s) in your device under test. A clock description must have at least one edge.

Symbols	Description
	Don't care. Clock line not used in this clock.
	Rising edge.
	Falling edge.
	Both edges.
	Qualifier - high.
	Qualifier - low.

NOTE

Note on adjusting the clock and qualifier's setup and hold time

In Synchronous sampling, the input logic signals need to be stable for a specific period of time before the clock qualifier event and for a specific period of time after the clock qualifier event. These time periods are called clock qualifier setup time and clock qualifier hold time respectively and vary based on the model of your logic analyzer. These time period requirements need to be met to ensure that the logic level is properly interpreted. When these time requirements are not met, it can result in missed clocks and measurements are not captured.

For instance, when the clock and clock qualifier transition at the same time, there may not be enough setup time for the clock qualifier resulting in missing out some samples that should have been captured on the occurrence of the clock event.

In such a situation, you can manually adjust the delay value for the clock qualifier channel to provide enough setup time as per your logic analyzer model. You can use the Signal Deskew tool to do this. This tool is accessible by selecting **Tools > External Applications > Signal Deskew Tool** from the Logic and Protocol Analyzer GUI's menubar.

To know about the clock and clock qualifier setup and hold time requirements for your model of logic analyzer, refer to your logic analyzer's specifications.

- See Also
- ["Setting up the State Sampling Options for a U4164A Logic Analyzer"](#) on page 542
 - ["Setting up the State Sampling Options in U4154A Logic Analyzer"](#) on page 547
 - ["Setting up the State Sampling Options in U4154B Logic Analyzer"](#) on page 551

- "Selecting the State Sampling Clock Mode" on page 99
- "To automatically adjust state sampling positions and threshold voltages" on page 128

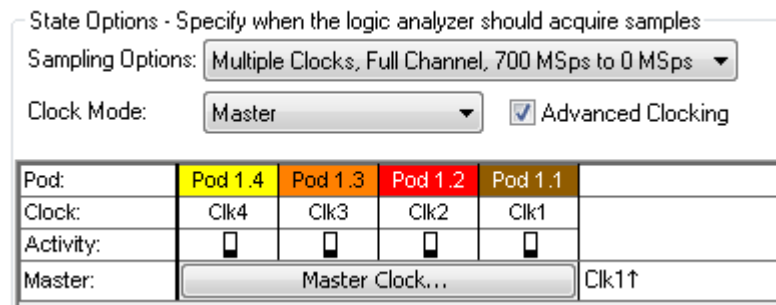
To set up advanced clocking

If you want to specify more complex clock setups than you can with the normal Master or Slave selections (for example, if you want to use a specific clock channel both as an edge and a qualifier in the same clock description), you need to use advanced clocking.

NOTE

The advanced clocking feature is only available for logic analyzers that support multiple clocks, that is, the U4164A and 16860-series logic analyzers.

- 1 From the menu bar, select **Setup**>(Logic Analyzer Module)>**State/Timing (Sampling)**...
- 2 Select **State** mode. The state options become selectable.
- 3 Select the Multiple Clocks option from the Sampling Option listbox.
- 4 Next to the clock mode, the Advanced Clocking checkbox becomes visible. Select this checkbox. The clock controls are replaced by a Master Clock button. In case of Master/slave clock mode, the clock controls are replaced by two buttons - Master Clock and Slave Clock.



- 5 Select Master Clock... or Slave Clock... button as appropriate to set advanced clocking options for master clock(s) or slave clock(s).
- 6 In the Advanced Clocking Setup Dialog (see [page 422](#)), choose settings as appropriate for clock edges and their qualifiers.
 - Clock channels can be used both as primary clocks and as clock qualifiers.
 - Clock qualifiers can be set with "And" or "Or" conditional operators.
 - Different qualifiers can be set for different clocks.
- 7 Click OK to close the dialog. The clock description in the Analyzer Setup window updates as per the clock definition in the Advanced Clocking Setup dialog.

NOTE

If you un-check the Advanced clocking checkbox, all qualifiers that you set are erased from the clock description.

See Also • Pod and Channel Naming Conventions (see [page 361](#))

To automatically adjust state sampling positions and threshold voltages

When adjusting the state mode sampling position with eyescan, the logic analyzer looks at signals from the device under test, figures out the threshold voltage that results in the widest possible data valid window, then figures out the location of the data valid window in relation to the sampling clock, and automatically sets the threshold voltage and sampling position.

Because eyescan automatically sets the sample position on individual channels, it can correct for the small skew effects caused by probe cables and circuit board traces. This makes the logic analyzer's setup/hold window smaller and lets you accurately capture data at higher clock speeds.

NOTE

You use the Eyescan feature with the U4154A/B Logic Analyzer and 16850-series logic analysis system to automatically adjust sampling positions and threshold voltages on individual channels. Refer to the topic [“Setting up and Running Eyescans in Logic Analyzers”](#) on page 567 to know more about the eyescan feature.

To manually adjust state sampling positions

For U4154A/B Logic Analyzer and 16850 series of Logic Analyzers you use the Eye Scan - Sample Positions and Thresholds Settings dialog to manually adjust state sampling positions.

In the Eye Scan - Sample Positions and Thresholds Settings dialog, you can manually adjust state mode sampling positions without running eye scan (which automatically adjusts state sampling positions).

- 1 Select the state (synchronous sampling) mode (see [To select the state acquisition mode](#)).
- 2 In the Sampling Setup dialog, click **Eye Scan: Sample Positions and Thresholds....** button.
- 3 In the Eye Scan - Sample Positions and Thresholds Settings dialog, drag the blue sampling position bars to the proper locations.

You can expand or collapse the channels in a bus.

Sampling positions are saved with the logic analyzer configuration (see [To save a configuration file](#) (see [page 198](#))).

- See Also
- [Understanding State Mode Sampling Positions](#) (see [page 367](#))
 - [To automatically adjust state sampling positions and threshold voltages](#) (see [page 128](#))

In Either Timing Mode or State Mode


- [To specify the trigger position](#) (see [page 128](#))
- [To set acquisition memory depth](#) (see [page 129](#))

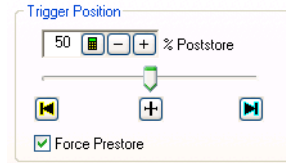
To specify the trigger position

The trigger position specifies the amount of trace memory used for samples captured after the trigger. For example, when 10% poststore is selected, 90% of trace memory is used for samples captured before the trigger. When 90% poststore is selected, 10 % of trace memory is used for samples captured before the trigger.

When **Force Prestore** is checked, the amount of pre-trigger and post-trigger memory is always what you expect because, after a run, the logic analyzer fills pre-trigger memory before it starts looking for a trigger. If the event you want to trigger on occurs while pre-trigger memory is being filled, the logic analyzer does not trigger.

When **Force Prestore** is unchecked, the trigger position may not end up being where you expect because the logic analyzer starts looking for a trigger immediately after a run (it does not wait for pre-trigger memory to be filled). For example, if you set the trigger position to 50%, but the logic analyzer finds the trigger right away, the amount of pre-trigger memory is less than you expect.

- 1 From the menu bar select **Setup>(Logic Analyzer Module)>Timing/State (Sampling)...**, or click the  icon from the setup toolbar (see [page 399](#)).
- 2 Select the trigger position, and check or uncheck **Force Prestore** as desired.

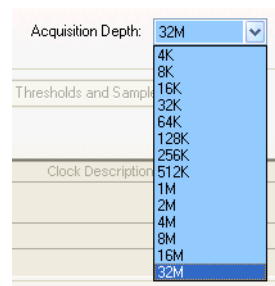


- See Also
- Understanding Logic Analyzer Triggering, The Conveyor Belt Analogy (see [page 369](#))
 - To set the acquisition memory depth (see [page 129](#))

To set acquisition memory depth

The acquisition depth control lets you set the amount of memory that is filled with data on an acquisition. The choices available depend on the maximum memory depth available in the analyzer that is being used.

- 1 From the menu bar select **Setup>(Logic Analyzer Module)>Timing/State (Sampling)...**
- 2 Set the acquisition mode and any state or timing options. These will affect the available memory choices.
- 3 In the Options box to the right, set Acquisition Depth.



- See Also
- Memory Depth and Channel Count Trade-offs (see [page 363](#))
 - Logic Analyzer Notes, Channels and Memory Depth (see [page 536](#))
 - Understanding Logic Analyzer Triggering, The Conveyor Belt Analogy (see [page 369](#))
 - To specify the trigger position (see [page 128](#))

Using Timing Zoom

Timing zoom collects additional high-speed timing data around the trigger of the logic analyzer.

The timing zoom settings are accessed through the TimingZoom box in the Sampling tab.

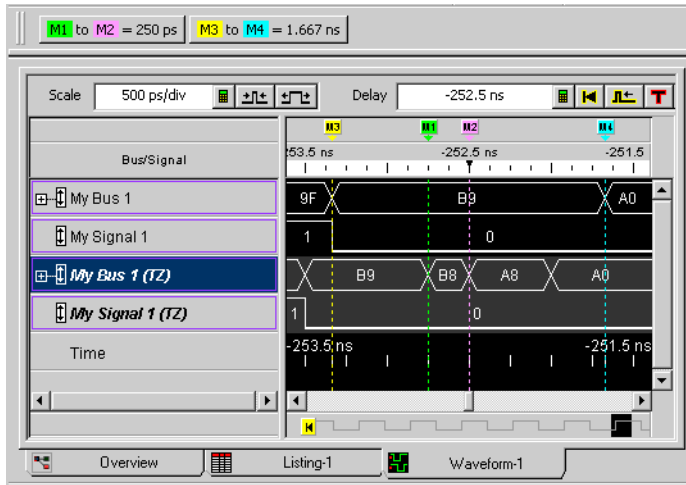
- To turn timing zoom on or off (see [page 131](#))
- To specify the timing zoom sample period (on some logic analyzers) (see [page 131](#))

- To specify the timing zoom trigger position (see [page 131](#))

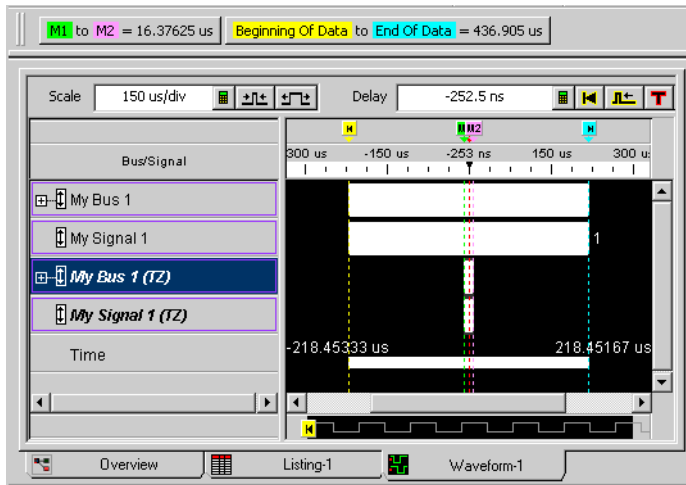
What is Timing Zoom?

Timing zoom collects a window of additional high-speed timing data around the trigger of the logic analyzer.

Because of timing zoom's faster sample rate, you get a higher-resolution view of transitions than with the normal timing mode. Timing zoom data appears in rows with "(TZ)" after bus/signal names.



Because of the faster sample rate and the relatively smaller amount of memory for samples, the overall window of time captured by timing zoom is smaller. Timing zoom data is captured around the trigger.



See Also • [Logic Analyzer Notes, Timing Zoom \(see page 536\)](#)

To turn timing zoom on or off

If you are not interested in the timing zoom data for a measurement, you can improve logic analyzer performance by turning off timing zoom.

- 1 In the Sampling tab, check or uncheck **Enable** in the TimingZoom box.

To specify the timing zoom sample period (on some logic analyzers)

With some logic analyzers (see Logic Analyzer Notes, Timing Zoom (see [page 536](#))), you can change the sampling period to see more or less sampling resolution around the trigger.

- 1 In the Sampling tab, check **Enable** in the TimingZoom box.
- 2 Click **Setup...** in the TimingZoom box.
- 3 In the TimingZoom Setup dialog (see [page 476](#)), select the **Sampling Period**.

See Also • Logic Analyzer Notes, Timing Zoom (see [page 536](#))

To specify the timing zoom trigger position

- 1 In the Sampling tab, check **Enable** in the TimingZoom box.
- 2 Click **Setup...** in the TimingZoom box.
- 3 In the TimingZoom Setup dialog (see [page 476](#)), drag the **Trigger Position** slider bar to the desired setting.

The trigger position specifies the amount of timing zoom memory used for samples captured after the trigger. For example when "10% poststore" is selected, 90% of timing zoom memory is used for samples captured before the trigger.

See Also • Logic Analyzer Notes, Timing Zoom (see [page 536](#))

Setting Up Symbols

You can use symbol names in place of bus/signal data values when:

- Setting up triggers.
- Displaying captured data.
- Searching for bus/signal values in the display windows.
- Setting up the Filter/Colorize tool.

Symbol names can be: variable names, procedure names, function names, source file line numbers, etc.

You can enter user-defined symbol names, or you can load symbol name definitions from a program's object file or from a general-purpose ASCII format symbol file.

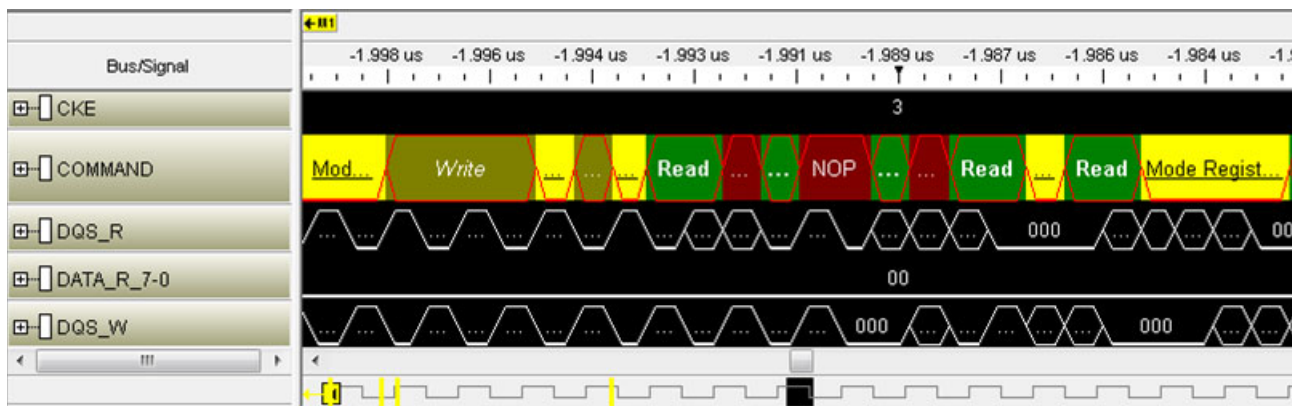
- To create and edit user-defined symbols (see [page 132](#))
- To load symbols from a file (see [page 134](#))
- To run the symbol reader outside the application (see [page 135](#))
- To create an ASCII symbol file (see [page 135](#))
- To change symbol reader options (see [page 136](#))

Multiple user-defined symbols can have the same name and different values. Symbol value lookups are based on the name and the value.

Multiple symbols with the same name are not allowed when loading symbols from a file. When a symbol file has multiple symbols with the same name, the first is accepted and the rest are ignored.

When two or more symbols have the same value, the first symbol name matching the value is used (even though you may have selected one of the others).

The following figure shows an example of symbols used for data presented in Waveform Viewer. Notice that different colors have been assigned to symbols for differentiation in the presented data.



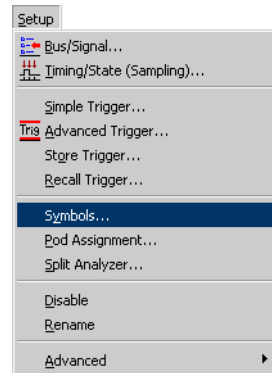
See Also • To enter symbolic bus/signal values (see [page 138](#))

To create and edit user-defined symbols

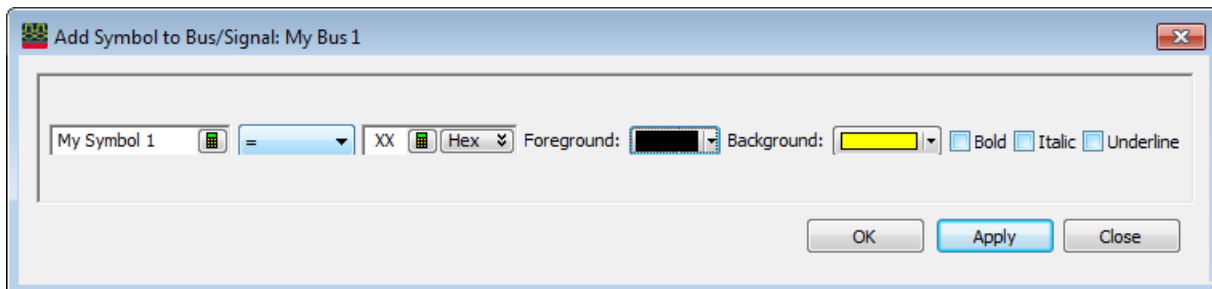
You can create and edit user-defined symbols for bus/signal values.

To add a user-defined symbol

- 1 Select **Setup>(Logic Analyzer Module)>Symbols...**



- 2 In the Symbols dialog (see [page 471](#)), select the bus or signal for which the new symbol should be displayed.
Each symbol is defined for a particular bus/signal.
- 3 Click **Add...**
- 4 In the Add Symbol dialog, define a value or range of values.



There are no restrictions on the characters you can use in the name of a symbol. You can also colorize the symbol by setting its background and foreground colors. These colors will then be used for this symbol in the data presented in Listing and Waveform Viewers. By default, white foreground and black background is used for symbols.

Many identical symbols for a bus/signal can be entered, all with unique or identical values. During symbol lookup by a window or tool, the first symbol that matches the pattern is used. This is why the Symbol dialog has Move Up and Move Down buttons for reordering symbols.

- 5 Click **Apply**.

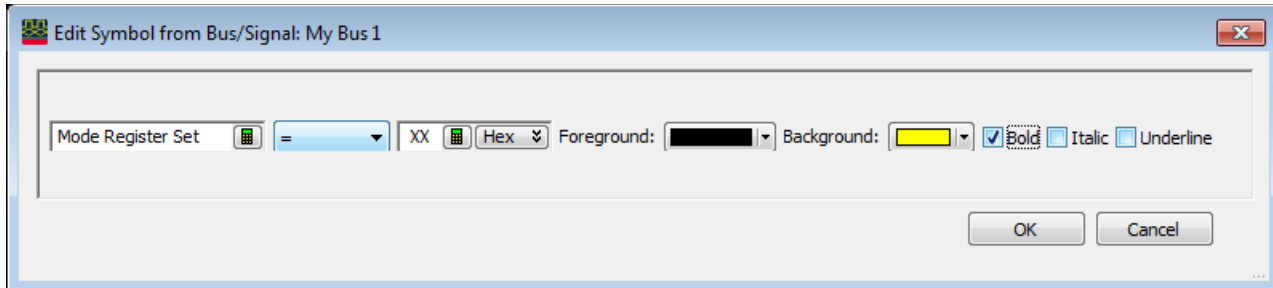
To see the symbols in the listing or waveform display, click OK in the Symbols dialog and change the base (see [page 241](#)) for the bus/signal to Symbols.

Inverse Assemblers (IA) can also create Symbols. These symbols are generated by the IA on run or on .ala load. For such IA generated symbols, the Add and Edit buttons are disabled in the Symbols dialog.

NOTE

Because XML format logic analyzer configuration files save and load user-defined symbols, you can also add symbols by (1) using text processing tools to re-format symbol information from software development tools, (2) inserting them into an XML format configuration file, and (3) loading the configuration file into the *Keysight Logic Analyzer* application (see "XML Format" (in the online help)).

- To edit a user-defined symbol
- 1 Select **Setup>(Logic Analyzer Module)>Symbols...**
 - 2 Select the symbol you want to edit.
 - 3 Click Edit and modify the details such as name or background/foreground colors used for the symbol.



- To delete a user-defined symbol
- 1 Select **Setup>(Logic Analyzer Module)>Symbols...**
 - 2 Select the symbol you want to delete.
 - 3 Click **Delete**.

- To save symbols
- Save symbols as part of a configuration file (see [page 198](#)). Symbols are saved in the configuration whether or not you select **Setup only** in the Save As dialog.
- You can move user-defined symbols from one bus/signal to another by saving to an XML format configuration file, editing, then reloading the file.

See Also • [Displaying Names \(Symbols\) for Bus/Signal Values \(see page 243\)](#)

To load symbols from a file

You can load symbols from object files, which are created by your compiler/linker or other software development tools, or you can load symbols from a general-purpose ASCII (GPA) format symbol files.

- 1 Create the symbol file:
 - Generate an object file with symbolic information using your software development tools (see [Object File Formats Supported by the Symbol Reader \(see page 513\)](#)).
 - Generate a Keysight Symbol Reader ".sym" file by running the symbol reader outside of the *Keysight Logic Analyzer* application (see [page 135](#)); loading symbols from ".sym" files is faster than loading them from object files.
 - If your language tools cannot generate object file formats that are supported by the logic analyzer, create an ASCII symbol file (see [page 135](#)).
- 2 From the *Keysight Logic Analyzer* application's main menu bar, choose **Setup>(Logic Analyzer Module)>Symbols...**
- 3 In the Symbols dialog (see [page 471](#)), select the bus/signal name you want to load object file symbols for.

In most cases, you will select the bus/signal representing the address bus of the processor you are analyzing.
- 4 Click **Load...**
- 5 In the Select Symbol File dialog, select the file from which you want to load symbols.
- 6 Click **Open**.

The name of the symbol file is saved when a configuration file is saved. The symbol file will be reloaded when the configuration is loaded.

- To reload symbols from a file
- 1 Choose **Setup>(Logic Analyzer Module)>Symbols...**
 - 2 In the Symbols dialog (see [page 471](#)), select the symbol file whose symbols you want to reload.
 - 3 Click **Load...**

The values of the symbols being used in the trigger sequence are updated automatically each time a symbol file is reloaded.

- To delete a symbol file
- 1 Choose **Setup>(Logic Analyzer Module)>Symbols...**
 - 2 In the Symbols dialog (see [page 471](#)), select the symbol file whose symbols you want to delete.
 - 3 Click **Delete**.

See Also • Object File Formats Supported by the Symbol Reader (see [page 513](#))

To run the symbol reader outside the application

You can run the symbol reader outside the *Keysight Logic Analyzer* application to create an Keysight Symbol Reader ".sym" file that loads faster than the object file.

- 1 Open a Command Prompt window.
- 2 Run the command:

```
agSymbolBuild.exe [-r <readers.ini>] <object_file> <dest_file>.sym
```

For example:

```
agSymbolBuild.exe q.elf q.sym
```

The **agSymbolBuild.exe** symbol reader program is located in the directory:

```
<Drive letter>:\<Install directory>\SymbolReaders\
```

For example:

```
C:\Program Files\Keysight Technologies\Logic Analyzer\SymbolReaders\
```

To change the symbol reader options, copy the readers.ini file from the SymbolReaders directory, edit it, and use the `-r <readers.ini>` option when running the **agSymbolBuild.exe** program.

For more information on the symbol reader program, see the README.txt file in the SymbolReaders directory.

See Also • To load symbols from a file (see [page 134](#))
• To change symbol reader options (see [page 136](#))

To create an ASCII symbol file

General-purpose ASCII (GPA) symbol files are created by converting object file symbols to a GPA format symbol files and/or by using text editing/processing tools.

To convert object file symbols to GPA format symbol files

When you need to apply different offsets to different symbols or sections of code, you can convert object file symbols to general-purpose ASCII (GPA) format symbol files. Then, you can use text editing/processing tools to adjust the symbol or section offset values in the GPA format file before loading the file into the *Keysight Logic Analyzer* application.

- 1 Open a Command Prompt window.
- 2 Run the command:

```
agSymbolQuery.exe -a <object_file> <dest_file>.sym > GPA_file
```

For example:

```
agSymbolQuery.exe -a q.elf q.sym > q.gpa
```

The `agSymbolQuery.exe` program is located in the directory:

```
<Drive letter>:\<Install directory>\SymbolReaders\
```

For example:

```
C:\Program Files\Keysight Technologies\Logic Analyzer\SymbolReaders\
```

For more information on the `agSymbolQuery.exe` program, see the README.txt file in the SymbolReaders directory.

See Also • General-Purpose ASCII (GPA) Symbol File Format (see [page 514](#))

To change symbol reader options

You can change how ELF/Stabs, Ticoff, or Coff/Stabs symbol files are processed by editing the readers.ini file.

1 Make a backup copy of the readers.ini file.

The readers.ini file is located in the directory:

```
<Drive letter>:\<Install directory>\SymbolReaders\
```

For example:

```
C:\Program Files\Keysight Technologies\Logic Analyzer\SymbolReaders\
```

2 Edit the readers.ini file.

For more information on the symbol reader options, see the comments in the readers.ini file.

Reader Options

SectionReloc Use the following options to specify the relocation. Replace <sectionname> with the name of your section. Replace <hex_relocation_value> with the hex relocation amount (32-bit max). You can set the relocation for section to an absolute value or you can add a relative relocation amount. The relocation value will be calculated using unsigned 32-bit math.

```
[SectionReloc]
```

```
Place this before all relocation options.
```

```
AddReloc_AllSections=<hex_relocation_value>
```

```
Relocates all sections by an amount specified.
```

```
If this command is used with subsequent relocation commands  
the subsequent commands will override this operation.
```

```
AddReloc_<sectionname>=<hex_relocation_value>
```

```
Adds a relative relocation value.
```

```
SetReloc_<sectionname>=<hex_relocation_value>
```

```
Relocates the section to the absolute address specified.
```

```
NonReloc_<sectionname>=TRUE
```

```
Inhibits a section from being relocated.
```

```
It is only useful when it follows a AddRloc+AllSections.
```

C++Demangle 1= Turn on C++ Demangling (Default)

0= Turn off C++ Demangling

C++DemOptions 803= Standard Demangling

203= GNU Demangling (Default Elf/Stabs)

403= Lucid Demangling

800= Standard Demangling without function parameters

	200= GNU Demangling without function parameters
	400= Lucid Demangling without function parameters
MaxSymbolWidth	80= Column width max of a function or variable symbol Wider symbols names will be truncated. (Default 80 columns)
OutSectionSymbol Valid	0= Symbols whose addresses aren't within the defined sections are invalid (Default) 1= Symbols whose addresses aren't within the defined sections are valid
	This option must be specified in the Nsr section of the Readers.ini file:
	[Nsr] OutSectionSymbolValid=1
ReadElfSection	2= Process all globals from ELF section (Default) Get size information of local variables 1= Get size information of global and local variables Symbols for functions will not be read, and only supplemental information for those symbols in the Dwarf or stabs section will be read. 0= Do not read the Elf Section
	If a file only has an ELF section this will have no effect and the ELF section will be read completely. This can occur if the file was created without a "generate debugger information" flag (usually -g). Using the -g will create a Dwarf or Stabs debug section in addition to the ELF section.
StabsType	StabsType=0 Reader will determine stabs type (Default) StabsType=1 Older style stabs (Older style stabs have individual symbol tables for each file that was linked into the target executable, the indexes of each symbol table restart at 0 for each file.) StabsType=2 Newer style stabs (New style stabs have a single symbol table where all symbols are merged into a large symbol array).
ReadOnlyTicoffPage e	ReadOnlyTicoffPage tells the ticoff reader to read only the symbols associated with the specified page (as an example 'ReadOnlyTicoffPage=0' reads only page 0 symbols). A value of -1 tells the ticoff readers to read symbols associated with all pages. ReadOnlyTicoffPage=-1 Read all symbols associated will all ticoff pages (Default) ReadOnlyTicoffPage=p Read only symbols associated with page 'p' (where p is any integer between 0 and n the last page of the object file).
AppendTicoffPage	AppendTicoffPage tells the ticoff reader to append the page number to the symbol value. This assumes that the symbol value is 16-bits wide and that page number is a low positive number which can be ORed into the upper 16 bits of an address to create a new 32-bit symbol address. For example, if the page is 10 decimal and the symbol address is 0xF100 then the new symbol address will be 0xAF100. AppendTicoffPage=1 Append the ticoff page to the symbol address

AppendTicoffPage=0 Do not append the ticoff page to the symbol address (Default)

Examples

Example for
Elf/Stabs

```
[ReadersElf]
C++Demangle=0
C++DemOptions=203
MaxSymbolWidth=60
StabsType=2
```

Example for
Coff/Stabs (using
Ticoff reader)

```
[ReadersTicoff]
C++Demangle=0
C++DemOptions=203
MaxSymbolWidth=60
StabsType=2
```

Example for Ticoff

```
[ReadersTicoff]
C++Demangle=0
C++DemOptions=203
MaxSymbolWidth=60
ReadOnlyTicoffPage=4
AppendTicoffPage=1
```

To enter symbolic bus/signal values

When entering bus/signal values while setting up triggers, searching display windows, or setting up the Filter/Colorize tool:

- 1 Select the desired operator for the bus/signal value.
- 2 Select the **Symbol** number base.
- 3 Click the value button.
- 4 In the Select Symbol dialog (see [page 458](#)), select the symbol you want to use.
All of the symbols for the current bus/signal, regardless of type, are available in the dialog.
- 5 Click **OK**.

See Also · [Select Symbol Dialog \(see page 458\)](#)

Installing Licensed Hardware Upgrades

Some of the newer logic analysis system cards (like the 16850-series logic analyzers) have hardware features (like state speed and memory depth) that can be upgraded by purchasing a license.

NOTE

When installing licensed hardware upgrades, you must run the *Hardware Update Utility* program on the frame that contains the cards you want to upgrade. In other words:

- In a multiframe logic analysis system, you must run the *Hardware Update Utility* program on each frame that has cards to be upgraded.
- You cannot install module upgrades over a remote connection (including remote connections via Remote Desktop, NetOp, or RealVNC).

To install a licensed hardware upgrade:

- 1 After you have ordered the hardware upgrade product/option and have received your license file, copy the license file to the directory:

C:\Program Files (x86)\Agilent Technologies\Logic Analyzer\License

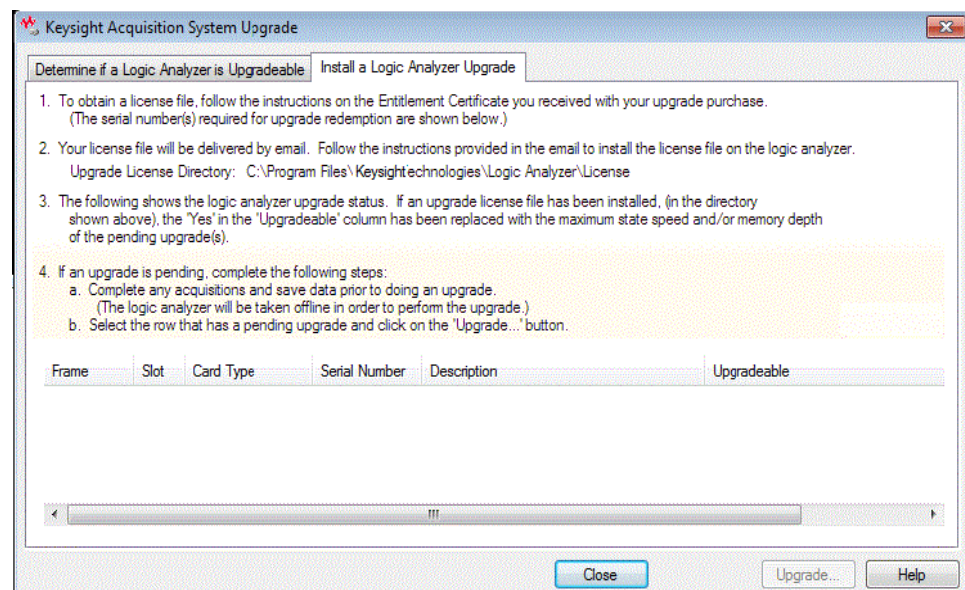
If you have installed the *Keysight Logic Analyzer* application in a different directory, copy the license file to the directory:

<Drive letter>:\<Install directory>\License\

If upgrade options were ordered for several cards at the same time, there will be one license file for all submitted serial numbers.

License file names must have the ".lic" extension in order to work.

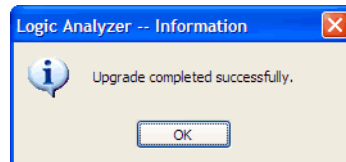
- 2 In the *Keysight Logic Analyzer* application, choose **Help>Logic Analyzer Upgrade....**
Or, from the Windows Start bar, click **Start>All Programs>Keysight Logic Analyzer>Utilities>Hardware Update Utility.**
- 3 In the *Keysight Logic Analyzer Upgrade* dialog's "Install a Logic Analyzer Upgrade" tab, select the card that has an upgrade pending.



CAUTION

The *Keysight Logic Analyzer* application will be taken offline in order to perform the upgrade. Therefore, complete any acquisitions and save data before performing the upgrade.

- 4 Click **Upgrade...**
- 5 When the upgrade completed information dialog appears, click **OK**.



The Keysight Logic Analyzer Upgrade dialog shows the upgraded hardware.

- 6 Click **Close** to close the Keysight Logic Analyzer Upgrade dialog.

Once you complete the hardware upgrade, the hardware will retain its new settings and can be moved to any 16850-series logic analysis system frame.

6 Capturing Data from the Device Under Test

After you have probed the device under test (see [page 57](#)) and set up the logic analyzer (see [page 71](#)) (by defining buses and signals (see [page 83](#)) and choosing the sampling mode (see [page 96](#))), you are ready to tell the logic analyzer when to capture/acquire data (in other words, set up a trigger) and run the measurement.

You can set up:

- *Quick Triggers* by drawing a box in a display window around the data to trigger on,
- *Simple Triggers* in a display window by specifying bus/signal values to trigger on, or
- *Advanced Triggers* by opening a dialog box and choosing from collections of predefined *trigger functions* (see [page 478](#)).

Advanced triggers let you trigger the logic analyzer after a sequence of events occur in the device under test.

Once you have set up a trigger, you can run the measurement. When the measurement completes, you can view the captured data (see [page 207](#)) and save it (along with the logic analyzer setup).

- **Setting Up Quick (Draw Box) Triggers** (see [page 143](#))
 - To set a Quick Trigger in the Waveform window (see [page 143](#))
 - To set a Quick Trigger in the Listing window (see [page 144](#))
 - To set a Quick Trigger in the Source window (see [page 145](#))
- **Specifying Simple Triggers** (see [page 146](#))
 - To specify bus patterns in a simple trigger (see [page 147](#))
 - To specify signal levels in a simple trigger (see [page 148](#))
 - To set a bus/signal edge in a simple trigger (see [page 148](#))
- **Specifying Advanced Triggers** (see [page 152](#))
 - To replace or insert trigger functions into trigger sequence steps (see [page 158](#))
 - To specify bus/signal patterns (see [page 159](#))
 - To set a bus/signal edge in an advanced trigger (see [page 160](#))
 - To specify packet events (in "Find a packet" trigger function) (see [page 164](#))
 - To specify a trigger sequence step's goto or trigger action (see [page 166](#))
 - To specify default storage (see [page 167](#))
 - To insert or delete events (see [page 168](#))
 - Using burst patterns (see [page 170](#))
 - To negate events (see [page 177](#))
 - To change the evaluation order of AND/OR'ed events (see [page 177](#))
 - To choose between a duration or occurrence count for events (timing mode) (see [page 178](#))
 - To insert or delete actions (in a trigger sequence step) (see [page 179](#))
 - To cut, copy, and paste trigger sequence steps (see [page 182](#))
 - To delete trigger sequence steps (see [page 182](#))
 - To show a trigger sequence step as Advanced If/Then trigger functions (see [page 182](#))

- To convert a trigger sequence step to Advanced If/Then trigger functions (see [page 183](#))
- To display or hide "If" clause comments (see [page 183](#))
- To clear the trigger sequence (see [page 185](#))
- **Triggering From, and Sending Triggers To, Other Modules/Instruments** (see [page 186](#))
- **Storing and Recalling Triggers** (see [page 194](#))
 - To store a trigger (see [page 194](#))
 - To recall a trigger (see [page 194](#))
 - To set the trigger history depth (see [page 195](#))
- **Running/Stopping Measurements** (see [page 196](#))
- **Saving Captured Data (and Logic Analyzer Setups)** (see [page 198](#))
 - To save a configuration file (see [page 198](#))
 - To export data to standard CSV format files (see [page 199](#))
 - To export data to module CSV format files (see [page 201](#))
 - To export data to module binary (ALB) format files (see [page 203](#))
- **Extending Capture Capability with COM/DCOM** (see [page 206](#))

- See Also
- Probing the Device Under Test (see [page 57](#))
 - Setting Up the Logic Analyzer (see [page 71](#))
 - Defining Buses and Signals (see [page 83](#))
 - Choosing the Sampling Mode (see [page 96](#))
 - Analyzing the Captured Data (see [page 207](#))

Setting Up Quick (Draw Box) Triggers

Within the Waveform, Listing, and Source windows, you can quickly set up a simple trigger by drawing a rectangle with the mouse or right-clicking on a source line.

After the simple trigger has been defined, and the analyzer is run, the trigger saved in the most recently used triggers list and can be recalled (see [page 194](#)) at any time.

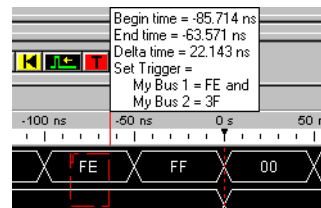
- To set a Quick Trigger in the Waveform window (see [page 143](#))
- To set a Quick Trigger in the Listing window (see [page 144](#))
- To set a Quick Trigger in the Source window (see [page 145](#))

To set a Quick Trigger in the Waveform window

In the Waveform window, you can quickly set up a simple trigger by drawing a rectangle with the mouse.

- 1 Make sure the Waveform window's Fast Zoom In (see [page 230](#)) option is not selected.
- 2 Using the mouse, point to the upper-left corner of your desired trigger rectangle.
- 3 While holding down the mouse button, drag the mouse pointer to the lower-right corner of your desired rectangle, then release the mouse button.

As you draw the rectangle, you can monitor the trigger as it is set with the tool tip readout that appears.



As you move the mouse left-to-right and top-to-bottom, the signal edge/level or bus value in contact with the **left of the rectangle** becomes the trigger.

Only one edge can be set.

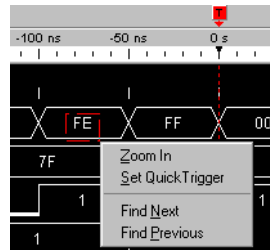
If a bus is expanded into its separate signals, three conditions apply:

- a If drawing starts on a bus, none of its expanded signals can be included.
- b If drawing starts on a signal, the bus cannot be included.
- c Edges and levels are mutually exclusive. That is, either one edge can be set, or all levels can be set, but not both at the same time.

NOTE

In the Waveform display window, it may be necessary to redraw the rectangle if you do not get your desired trigger points dictated by the left-side line of the rectangle. You could also try drawing the rectangle backwards leaving the left-side rectangle line set last.

- 4 Select **Set Quick Trigger**.



General Guidelines

- Any bus/signals with overlapping bits are not included within the trigger specification.

Example: Bus_1 has channels 0 through 7 of pod 1 assigned and Bus_2 has channels 3 through 6 of pod 1 assigned. At this point, you have the same probed signals (channels 3 through 6 of pod 1) assigned in both Bus_1 and Bus_2. Now you draw the rectangle over both bus_1 and bus_2. Since Bus_1 channels 3 through 6 are repeated (overlapped) on Bus_2, they will not be included in the trigger specification.
- Only a single sequence step can be defined by a drawn rectangle.
- As you draw the rectangle, a tool tip is displayed showing the current trigger specification that would be set.

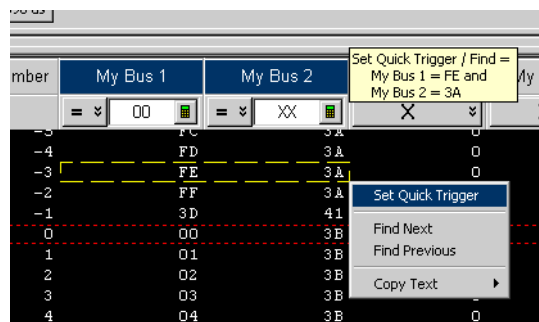
To set a Quick Trigger in the Listing window

In the Listing window, you can quickly set up a simple trigger by drawing a rectangle with the mouse.

- Using the mouse, point to the sample that you want to use as the quick trigger.
- While holding down the mouse button, drag the mouse pointer horizontally to draw a rectangle around the buses/signals you want to include in the trigger; then, release the mouse button.

As you draw the rectangle, a tool tip shows the trigger that will be set.

Dragging the mouse pointer vertically does not affect the sample used for the quick trigger; the sample used is always the one from which the drawn rectangle originates.
- Select **Set Quick Trigger**.



General Guidelines

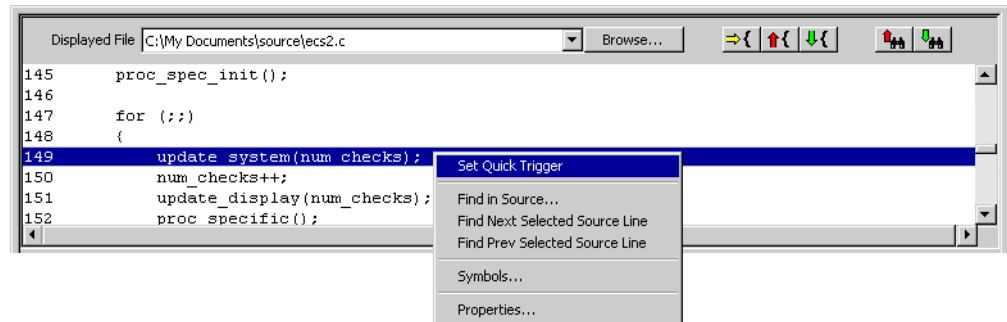
- Any bus/signals with overlapping bits are not included within the trigger specification.

Example: Bus_1 has channels 0 through 7 of pod 1 assigned and Bus_2 has channels 3 through 6 of pod 1 assigned. At this point, you have the same probed signals (channels 3 through 6 of pod 1) assigned in both Bus_1 and Bus_2. Now you draw the rectangle over both bus_1 and bus_2. Since Bus_1 channels 3 through 6 are repeated (overlapped) on Bus_2, they will not be included in the trigger specification.
- Only a single sequence step can be defined by a drawn rectangle.

- As you draw the rectangle, a tool tip is displayed showing the current trigger specification that would be set.

To set a Quick Trigger in the Source window

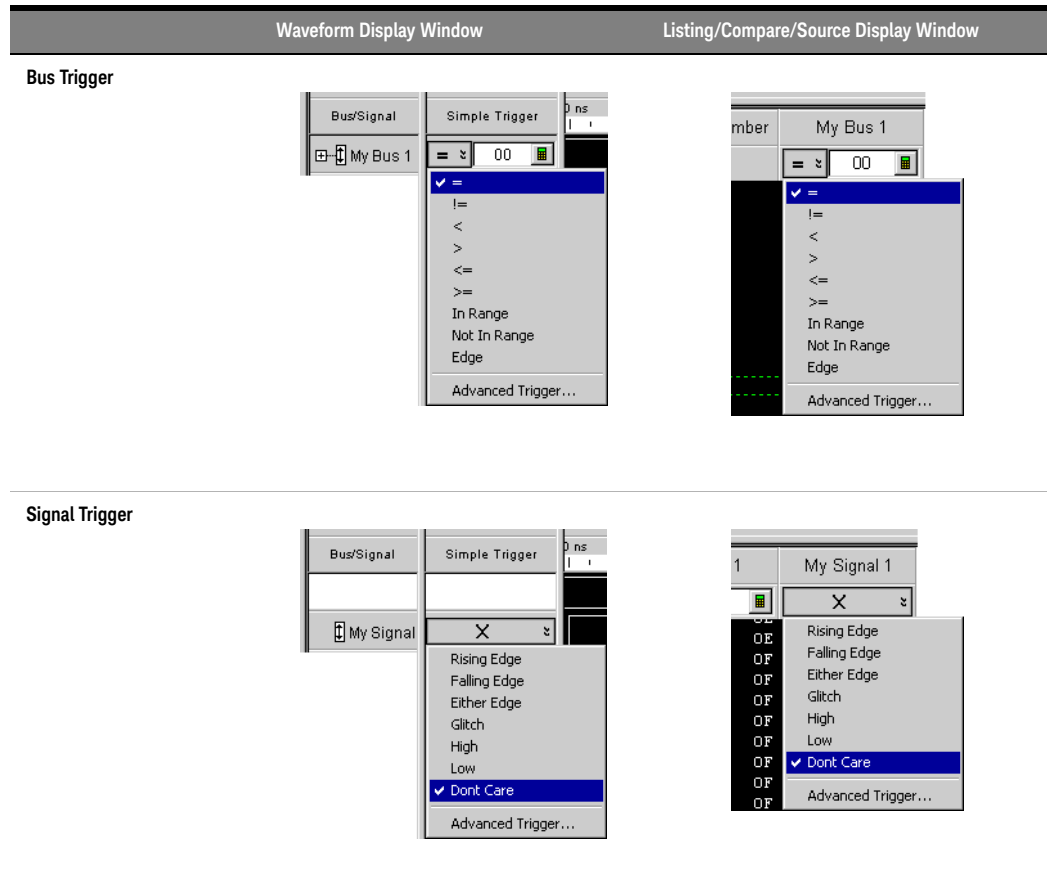
- 1 In the Source window's source pane, right-click the source line you want to set a Quick Trigger on, and choose **Set Quick Trigger**.



- See Also
- To change the "Set Quick Trigger" alignment (see [page 282](#))
 - Running/Stopping Measurements (see [page 196](#))
 - Viewing Source Code Associated with Captured Data (see [page 277](#))

Specifying Simple Triggers

Simple triggers let you quickly set up triggers on edges and bus/signal patterns from within display windows.



Buses are compared to entered pattern values using a relational operator (=, !=, <, >, <=, >=, **In Range**, **Not In Range**) or to one of multiple edges.

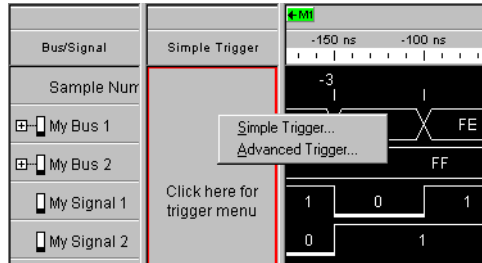
Signals are compared to edges (**Rising Edge**, **Falling Edge**, **Either Edge**, or a logic level pattern (**High**, **Low**). Edge options are available in timing mode only. For 16850 series and U4154A/B Logic Analyzer modules, edge options are also available in the state mode.

You can specify multiple bus/signal pattern values and one edge, all of which must evaluate to true for a sample to trigger the logic analyzer. (If you try to specify multiple edges, the last edge specified has priority, and the previously specified edge is changed to don't care.)

When buses/signals overlap (that is, the same logic analyzer channels are assigned to multiple buses/signals), the last change has highest priority. For example, if you specify a pattern on Bus A and then specify a rising edge on Signal B, which is bit 0 on Bus A, the previously specified pattern is erased.

When the desired trigger condition requires more than a simple AND expression (for example, one pattern OR another pattern on a bus, patterns in a sequence of samples, testing timer or counter values, etc.), you can choose **Advanced Trigger...** to specify an advanced trigger (see [page 152](#)). When an advanced trigger surpasses the functional limits of a simple trigger, the simple trigger fields

go away; to restore them, you must either change the advanced trigger so that it doesn't surpass the limits of a simple trigger, or click [Click here for trigger menu](#) and choose **Simple Trigger...** to reset the trigger.

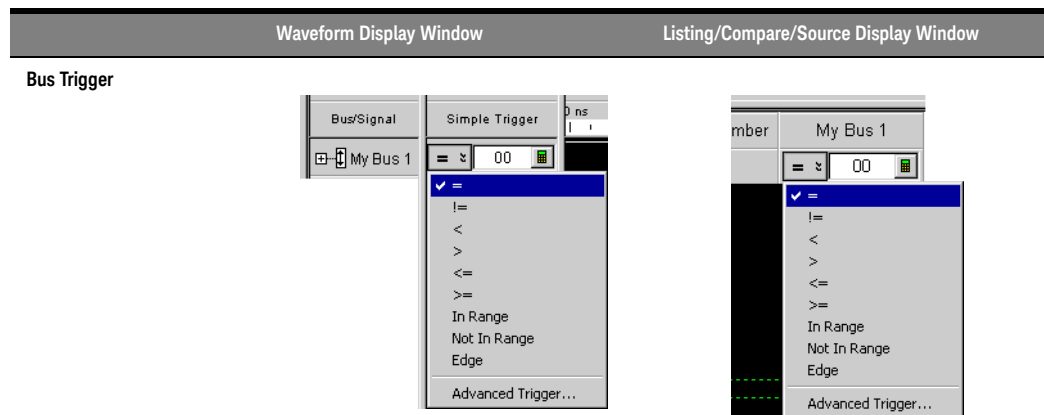


- To specify bus patterns in a simple trigger (see [page 147](#))
- To specify signal levels in a simple trigger (see [page 148](#))
- To set a bus/signal edge in a simple trigger (see [page 148](#))


- See Also
- Setting Up Quick Triggers (see [page 143](#))
 - Specifying Advanced Triggers (see [page 152](#))
 - To store a trigger (see [page 194](#))
 - To recall a trigger (see [page 453](#))

To specify bus patterns in a simple trigger

When specifying simple triggers (see [page 146](#)), you can specify bus patterns to trigger on.




To specify a bus pattern

- 1 In the Simple Trigger field for a bus, click the  operator button; then, choose from one of the following operators:
 - = (Equal To)
 - != (Not Equal To)
 - < (Less Than)
 - > (Greater Than)
 - <= (Less Than Or Equal To)
 - >= (Greater Than Or Equal To)

- In Range
- Not In Range

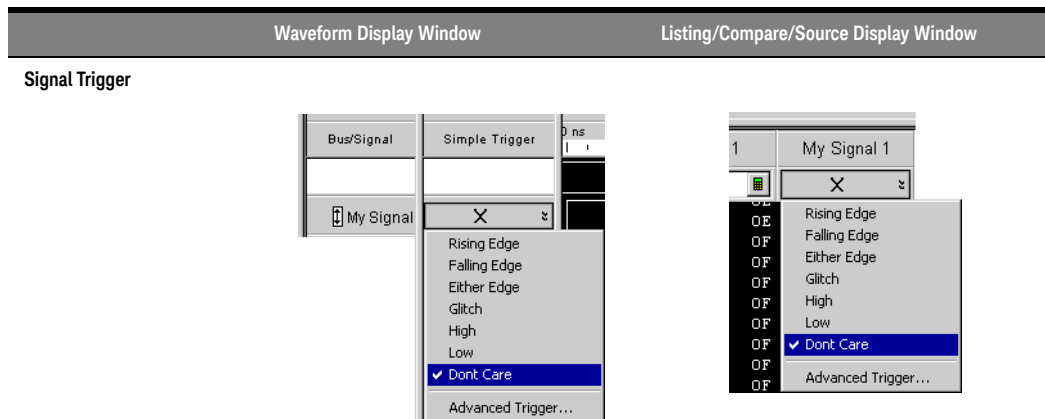
NOTE

The $<$, $>$, $<=$, $>=$, **In Range**, and **Not In Range** operators are not available when a bus with reordered bits has been selected. Also, these operators cannot be used when the selected bus contains clock bits that span pod pairs. The **In Range** and **Not In Range** operators are limited to buses that span 2 or fewer pod pairs (up to 64 bits wide).


- 2 In the text entry field , enter the bus pattern value to compare.

To specify signal levels in a simple trigger

When specifying simple triggers (see [page 146](#)), you can specify signal levels to trigger on.



To specify a signal level

- 1 In the Simple Trigger field for a signal, click the  edge/level button; then, choose from:
 - High
 - Low

See Also · To specify bus patterns in a simple trigger (see [page 147](#))

To set a bus/signal edge in a simple trigger

You can set a trigger on Rising, Falling, or Either Edge when the logic analyzer is configured in the Timing - Asynchronous sampling mode.

NOTE

For 16850-series and U4154A/B logic analyzer modules, you can set a trigger on edge in the Timing - Asynchronous sampling mode as well as State - Synchronous sampling mode.

Trigger on edge options are available in a simple trigger, an advanced trigger, and an eyescan trigger.

NOTE

The trigger event occurs between the previous state clock and the state clock at which the trigger point is marked.

If there is a rising edge and a falling edge on the same signal between the previous state clock and the state clock at which the trigger point is marked, such that the actual state of the signal is the same at both the previous state clock and the state clock at which the trigger point is marked, then the trigger will not occur.

You can configure trigger on edge in a simple trigger from the Waveform or Listing display window. The trigger settings that you configure in one of these windows is reflected automatically in the other window.

From Waveform Viewer

For a simple trigger created from Waveform Viewer, you can set a trigger on edge at an individual bit/signal level or at bus level.

At the bus level, you can set trigger on edge on one or more signals within the bus. If you set trigger on edge on multiple signals within a bus, then trigger on edge for these signals is ORed.

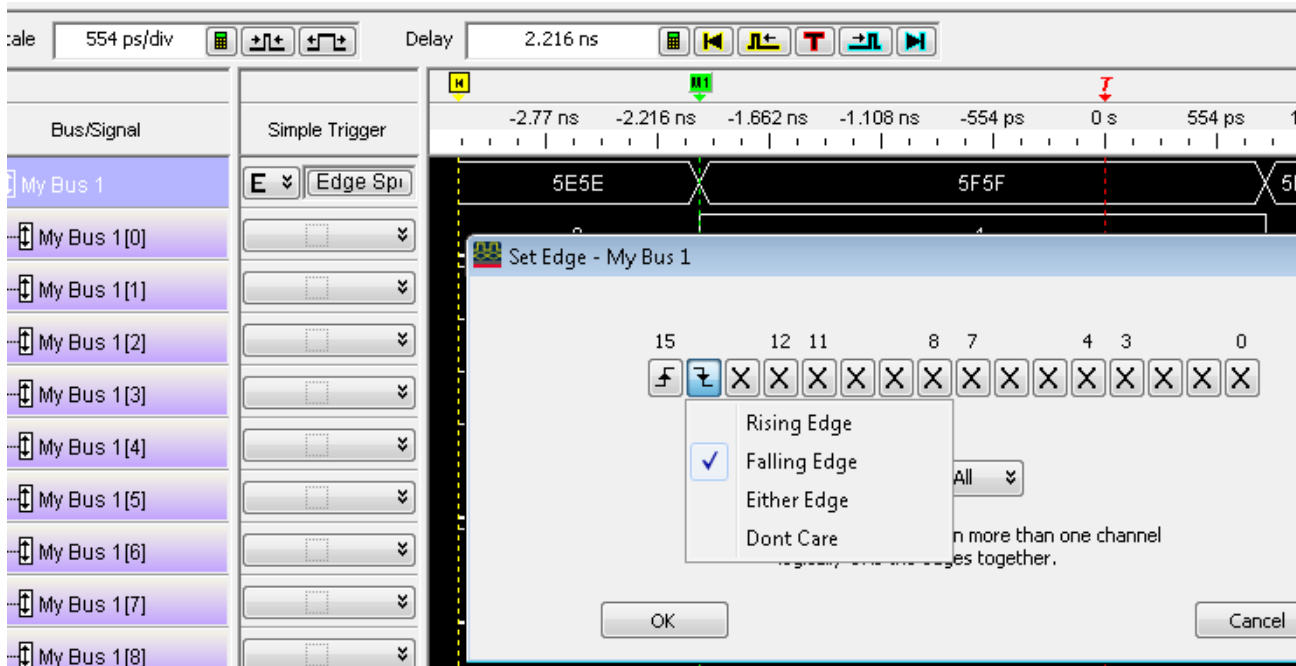


Figure 1 Figure - Trigger on edge set from Waveform viewer for a bus

At the individual signal level, you can set trigger on edge for one signal at a time. If you try to specify multiple edges, the last edge specified has priority, and the previously specified edge is changed to "Don't Care".

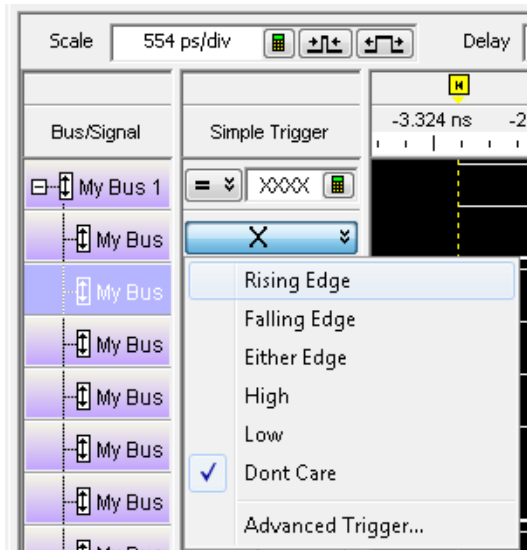


Figure 2 Trigger on edge set from Waveform viewer for a signal

From Listing Viewer

For a simple trigger created from Listing Viewer, you can set a trigger on edge on one or more signals within a single bus. For multiple signals, trigger on edge is ORed.

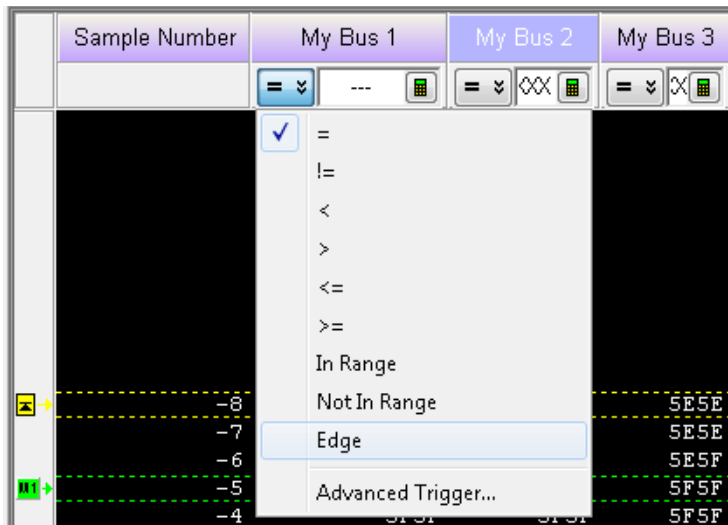


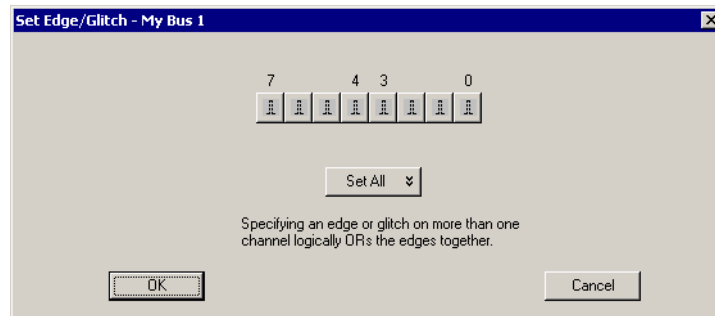
Figure 3 Trigger on edge set from Listing viewer for a bus

To specify edges on a bus

- 1 In the Simple Trigger field for a bus, click the **=** operator button; then, choose **Edge**.
- 2 Click **Edge Spec...**




- 3 In the Set Edge dialog, specify edges you are looking for; use the **Set All** button to make a selection for all signals in the bus.



- 4 Click **OK** to close the Set Edge dialog.

To specify a signal edge

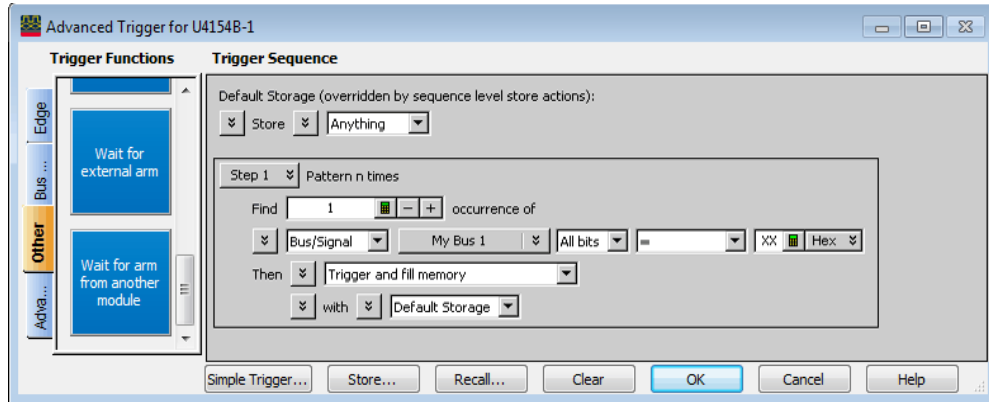
- 1 In the Simple Trigger field for a signal, click the  edge/level button; then, choose from:
 - **Rising Edge**
 - **Falling Edge**
 - **Either Edge**

- See Also
- To specify signal levels in a simple trigger (see [page 148](#))
 - Understanding Logic Analyzer Triggering, Edges (see [page 369](#))

Specifying Advanced Triggers

When you need to set up triggers that are more complex than just finding particular bus/signal values (for example, when you need to trigger on a sequence of events in the device under test), you set up advanced triggers.

To open the Advanced Trigger dialog, click **Trig** in the analyzer setup toolbar, or choose **Setup>(Logic Analyzer Module)>Advanced Trigger...** from the menu bar.



Advanced triggers are specified by dragging-and-dropping predefined *trigger functions* (see [page 478](#)) into trigger sequence steps. If the trigger function you need doesn't exist, start with a trigger function that is close, convert the trigger sequence step to advanced If/Then trigger functions, and edit the If/Then trigger functions.

Each step in the trigger sequence looks for *events* (see [page 153](#)) in data samples captured from the device under test (or in logic analyzer timers, counters, or flags), and when those events are found, takes some *action* (see [page 153](#)) (like triggering or going to another step in the sequence). You can also insert actions for timers, counters, or flags.

Default storage lets you ignore the question, in individual trigger sequence steps, of which captured samples should be stored in logic analyzer memory. However, you can insert *storage control* actions in individual trigger sequence steps to specify whether samples should be stored or to specify whether default storage should be turned on or off. Sequence step storage control actions override the default storage specification.

- To replace or insert trigger functions into trigger sequence steps (see [page 158](#))
- To specify bus/signal patterns (see [page 159](#))
- To set a bus/signal edge in an advanced trigger (see [page 160](#))
- To specify packet events (in "Find a packet" trigger function) (see [page 164](#))
- To specify a trigger sequence step's goto or trigger action (see [page 166](#))
- To specify default storage (see [page 167](#))
- To insert or delete events (see [page 168](#))
 - To insert a timer event (see [page 169](#)) (see also Using Timers (see [page 153](#)))
 - To insert a counter event (see [page 169](#)) (see also Using Counters (see [page 154](#)))
 - To insert a flag event (see [page 170](#)) (see also Using Flags (see [page 157](#)))
 - To insert an "Arm in from" event (see [page 170](#))
- To negate events (see [page 177](#))
- To change the evaluation order of AND/OR'ed events (see [page 177](#))

- To choose between a duration or occurrence count for events (timing mode) (see [page 178](#))
- To insert or delete actions (in a trigger sequence step) (see [page 179](#))
 - To insert a timer action (see [page 179](#)) (see also Using Timers (see [page 153](#)))
 - To insert a counter action (see [page 180](#)) (see also Using Counters (see [page 154](#)))
 - To insert a reset occurrence counter action (see [page 180](#))
 - To insert a flag action (see [page 180](#)) (see also Using Flags (see [page 157](#)))
 - To insert a storage control action (see [page 181](#))
- To cut, copy, and paste trigger sequence steps (see [page 182](#))
- To delete trigger sequence steps (see [page 182](#))
- To show a trigger sequence step as Advanced If/Then trigger functions (see [page 182](#))
- To convert a trigger sequence step to Advanced If/Then trigger functions (see [page 183](#))
- To display or hide "If" clause comments (see [page 183](#))
- To clear the trigger sequence (see [page 185](#))

- See Also
- Understanding Logic Analyzer Triggering (see [page 369](#))
 - State Mode Trigger Functions (see [page 490](#))
 - Timing Mode Trigger Functions (see [page 478](#))
 - Specifying Simple Triggers (see [page 146](#))

Reading Event and Action Statements

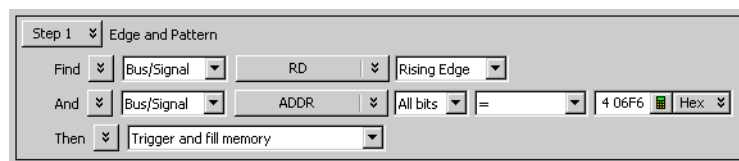
Event and action statements in trigger sequence steps read from left-to-right like:

Find an event in the device under test; when the event is found, take some action.

Or:

If an event is found in the device under test; then, take an action.

For example, suppose you want to see what happens after a read from the address 406F6H. To do this, set up the trigger to look for a rising edge on the RD (memory read) signal and an address bus pattern of 406F6H (hexadecimal):



As you set up the trigger, think of it as constructing a sentence that reads left-to-right. For example:

Find a **Signal** named **RD** with a **Rising Edge** And a **Bus** named **ADDR** with **All bits= (equal)** to the pattern **406F6 Hex**. When found, then **Trigger and fill memory**.

- See Also
- Specifying Advanced Triggers (see [page 152](#))
 - Understanding Logic Analyzer Triggering, Sequence Steps (see [page 371](#))

Using Timers

Timers are like stopwatches. Use timers to create either a user-defined delay or a time standard which valid data duration is evaluated against. The timer can **Start** from reset, **Stop** and reset, **Pause**, or **Resume**.

Timer considerations:

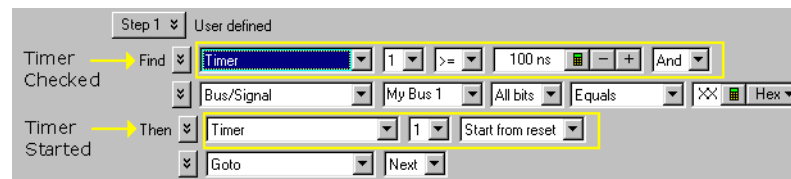
- It takes a certain amount of time for timers to reset; this is called the *timer reset latency*. To find the timer reset latency for your logic analyzer, see the description of its characteristics (see [page 587](#)).
- The number of timers available in a module depends on the selected acquisition mode and sampling option:

Acquisition Mode	Sampling Option	# Timers Available
State mode (synchronous sampling)	Single Clock	1
	Multiple clocks	3
Timing mode (asynchronous sampling)	all	1

Refer to your logic analyzer characteristics (see [page 587](#)) for the actual number of timers available. For information on when pod pairs are reserved for time tag storage, see *Why Are Pods Missing?* (see [page 362](#)).

- Timers are checked in event statements, and started in action statements.
- Timers must be started before they can be checked. This is done by either including the timer start action with the timer check event within the same trigger step or starting the timer in a preceding trigger step.

The following example shows the timer start action and check event within the same trigger step.



- Once a timer event is configured, you can reuse the timer by selecting its identification number. The same timer must always be checked against the same value. To check for different durations, use different timers.

See Also

- To insert a timer action (see [page 179](#))
- To insert a timer event (see [page 169](#))
- Understanding Logic Analyzer Triggering, Timers (see [page 374](#))

Using Counters

Counters are used to count eventual occurrences during an acquisition. Like other events, these counters evaluate to either true or false. You can use counters to create a user-defined delay, or to create a standard against which valid data duration is evaluated.

There are three types of counters available for use within trigger sequence steps. Based on your logic analyzer model and the sampling option you select, the availability of counters changes.

The following table lists and compares these three types of counters.

	Event counter	Global counter	Occurrence counter
Available in	Single Clock state sampling option	All timing sampling options	Multiple Clocks state sampling option only All sampling options
Number of counters	2 per trigger sequence	2 per trigger sequence	1 per trigger sequence step
Usage	To increment/reset the counter's value on the occurrence of an event and observe the counter's value in the Status button during acquisition.	To guide the trigger sequence by testing the counter's value in the "If" clause.	To count eventual occurrences in a sequence step.
Used in	Only the Action statements of a trigger sequence (Cannot be tested in an If clause of a step.)	Event and Action statements of a trigger sequence	(Can be tested in an If clause of a step.) Steps that contain the "occurs" phrase
Counter Attributes	Increment Reset	Increment Decrement Reset	Reset occurrence counter
Reuse	Once configured, the counter persists throughout the steps of a trigger sequence and you can reuse it by selecting its identification number.	Once configured, the counter persists throughout the steps of a trigger sequence and you can reuse it by selecting its identification number.	Once configured, the counter persists within a trigger sequence step and can be reused within that step using the Reset occurrence counter action.

Global Counter's Terminal and Current Values

For a global counter, the following two values are relevant in a trigger setup:

- Terminal value - This value is defined for a global counter by using the counter as an event in an "If" clause. For example, the following "If" clause defines the terminal value of the global counter 1 as 7:



A global counter's terminal value remains zero until it is defined in an "If clause".

In a trigger setup, a global counter can have only one terminal value. This means that you cannot compare a global counter's current value to two different terminal values in the trigger setup.

- Current value - This is the current value of the global counter which is compared against the terminal value defined for the counter. At the beginning of an acquisition, a global counter's current value is always zero.

You can use Increment, Decrement, or Reset actions in trigger steps to change the current value of the global counter during an acquisition. A Reset action sets the global counter's current value to zero.

NOTE

During acquisition, once the global counter's current value reaches the counter's terminal value, then Increment or Decrement actions have no effect on the current value. In such a case, only resetting the counter can change its current value to zero. After the reset, the counter's current value can again be incremented.

Another situation in which Increment or Decrement have no effect on the current value is when you have not defined the global counter's terminal value by using it as an event in an If clause and trying to increment the current value. In such a situation, the global counter's terminal value as well as current value is zero and therefore equal, resulting in Increment and Decrement actions having no effect.

Examples

Global Counter

The screenshot shows two steps in an 'Advanced If/Then' configuration:

- Step 1:**
 - If: Anything occurs 1 time.
 - Then: Counter 1 Reset (highlighted with a red box).
 - Then: Goto 2.
- Step 2:**
 - If: Counter 1 >= 7 (highlighted with a red box).
 - Then: Trigger and fill memory with Default Storage.
 - Else if: Bus/Signal My Bus 1 All bits = 11 Hex.
 - Then: Counter 1 Increment (highlighted with a red box).
 - Then: Goto 2.
 - Else if: Bus/Signal My Bus 1 All bits = 22 Hex.
 - Then: Counter 1 Decrement (highlighted with a red box).
 - Then: Goto 2.

Annotations on the right side:

- Global counter's current value set to zero
- The event defines the global counter's terminal value to be 7
- Global counter's current value incremented
- Global counter's current value decremented

Event Counter

The screenshot shows Step 1 in an 'Advanced If/Then' configuration:

- If: Bus/Signal My Bus 1 All bits = C0 Hex.
- Then: Event Counter 1 Increment (highlighted with a red box).
- Then: Goto 2.

Annotation on the right side:

- Value of Event counter 1 to be incremented on the occurrence of the event

Occurrence Counter

The screenshot shows Step 1 in an 'Advanced If/Then' configuration:

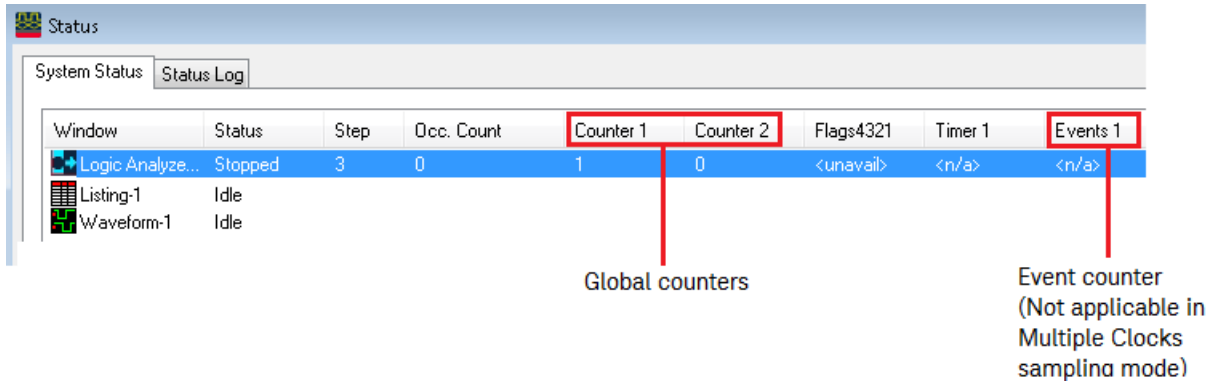
- If: Bus/Signal My Bus 1 All bits = FF Hex.
- Then: Trigger and fill memory with Default Storage.
- Else if: Bus/Signal My Bus 1 All bits = 00 Hex.
- Then: Reset occurrence counter (highlighted with a red box).
- Then: Goto 1.

Annotations on the right side:

- Occurrence counter for the step
- Occurrence counter reset to restart the search in the step

Viewing the Status of Counters

You can use the Status dialog box to view the current value of a counter. To access this dialog box, click the Status button in the Status bar at the bottom of the GUI.



- See Also
- To insert a counter action (see [page 180](#))
 - To insert a counter event (see [page 169](#))
 - Understanding Logic Analyzer Triggering, Counters (see [page 373](#))

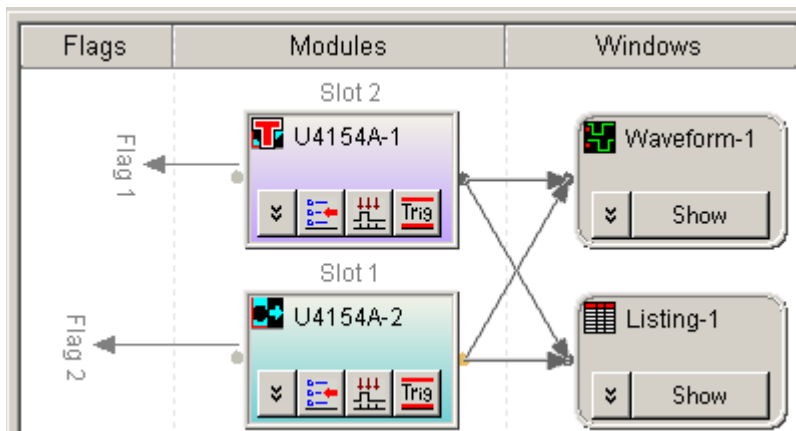
Using Flags

Flags can be used to signal between modules in the logic analysis system including a multiframe logic analysis system as well as between modules installed in an Keysight AMP/AXIe chassis. Using flags, logic analyzer modules can communicate back and forth with each other multiple times during a data acquisition, both before and after their trigger events occur. (By comparison, a module can arm another module one time when its trigger occurs.)

There are 4 flags that are shared across all connected logic analysis/AMP/AXIe system frames. By default, flags are cleared. In the Advanced Trigger dialog of the module, you can insert actions to set, clear, pulse set, or pulse clear a flag. You can also insert flag events in the Advanced Trigger dialog of different modules to test whether a particular flag is set or clear.

Setting a Flag

In legacy logic analysis systems, a particular flag may be driven (set) or received by multiple modules. However, in newer AMP/AXIe based instrument modules, only one module can set a particular flag. Multiple modules cannot set the same flag. But multiple modules can insert flag events for the same flag to test whether the flag is set or clear.



Clearing a Flag

A flag that is set by a module remains set until that module clears it. In legacy logic analysis systems, if multiple modules set the same flag, then all those modules must clear the flag before it becomes clear. In newer AMP/AXIe based instrument modules, multiple modules cannot set the same flag and therefore, the module that sets the flag can only clear it.

Flags can also be used to drive the logic analysis system's Port Out signal.

Viewing the Status of Flags

You can use the Status dialog box to view the current status of a flag. To access this dialog box, click the Status button in the Status bar at the bottom of the GUI.

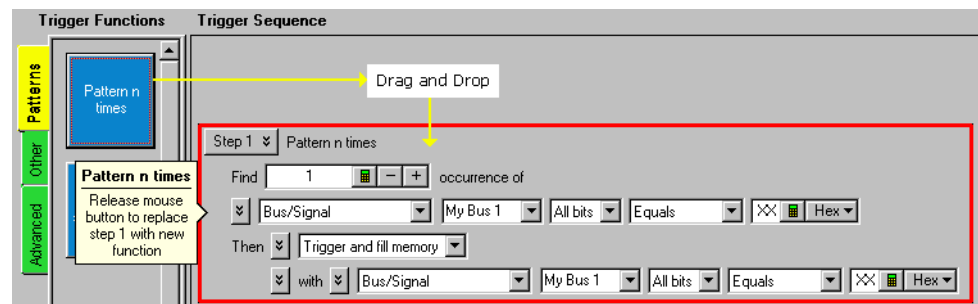
The flag status is not available if you selected the Multiple Clocks state sampling option for your logic analyzer.

- See Also
- To insert a flag action (see [page 180](#))
 - To insert a flag event (see [page 170](#))
 - Understanding Logic Analyzer Triggering, Flags (see [page 373](#))

To replace or insert trigger functions into trigger sequence steps

Multiple steps in the trigger sequence are necessary when you want to trigger on a sequence of events in the device under test. When you want to trigger on one event in (that is, a single sample from) the device under test, a single step in the trigger sequence is all you need.

- 1 In the Advanced Trigger dialog, **drag-and-drop** the desired Trigger Function (see [page 478](#)) into the Trigger Sequence display area.



To replace an existing step:

- Drag-and-drop the trigger function on top of an existing step in the trigger sequence. A red box around the old function indicates the replace operation.

To insert a trigger function as a new step:


- Drag-and-drop the trigger function above or below an existing step in the trigger sequence. When the mouse is positioned above or below an existing step, a red insert bar appears to indicate the relative insert location of the trigger function.

- See Also
- Understanding Logic Analyzer Triggering, Sequence Steps (see [page 371](#))
 - To delete trigger sequence steps (see [page 182](#))
 - State Mode Trigger Functions (see [page 490](#))
 - Timing Mode Trigger Functions (see [page 478](#))

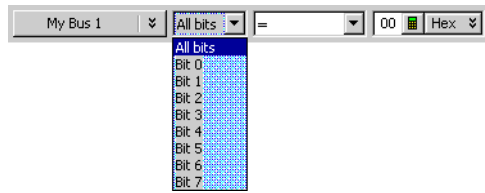
To specify bus/signal patterns

- 1 In the Advanced Trigger dialog, select the bus or signal.



Clicking  lets you select from recently used bus/signal names. Clicking elsewhere on a bus/signal name button opens a Select dialog for selecting a different name.

- 2 If a bus has been selected, either select **All bits** on the bus or select an individual bit.



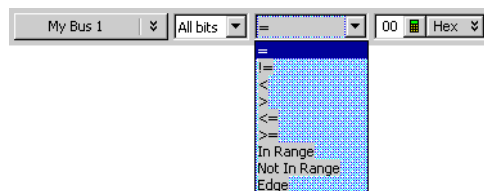
- 3 Specify the bus/signal value:

If a signal or one bit of a bus has been selected, select the signal pattern value (**High**, **Low**, or **Don't Care**).



If all bits of a bus have been selected:

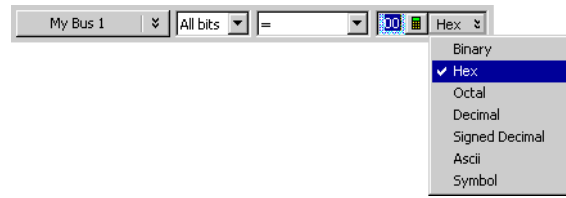
- a Select one of the operators: = (equal to), != (not equal to), < (less than), > (greater than), <= (less than or equal to), >= (greater than or equal to), **In Range**, **Not In Range**, or **Edge**.



NOTE

The <, >, <=, >=, **In Range**, and **Not In Range** operators are not available when a bus with reordered bits has been selected. Also, these operators cannot be used when the selected bus contains clock bits that span pod pairs. The **In Range** and **Not In Range** operators are limited to buses that span 2 or fewer pod pairs (up to 64 bits wide).

- b Select the number base (**Binary**, **Hex**, **Octal**, **Decimal**, **Signed Decimal**, also known as two's complement, **Ascii**, or **Symbol**).



- c Enter the pattern value(s).

When the **Symbol** number base is selected, you use the Select Symbol dialog (see [page 458](#)) to specify the pattern values.

- See Also
- To specify default storage (see [page 167](#))
 - To insert or delete events (see [page 168](#))
 - Understanding Logic Analyzer Triggering, Ranges (see [page 373](#))

To set a bus/signal edge in an advanced trigger

You can set a trigger on Rising, Falling, or Either Edge when the logic analyzer is configured in the Timing - Asynchronous sampling mode.

NOTE

For 16850-series and U4154A/B logic analyzer modules, you can set a trigger on edge in the Timing - Asynchronous sampling mode as well as State -Synchronous sampling mode.

Trigger on edge options are available in a simple trigger, an advanced trigger, and an eyescan trigger.

NOTE

The trigger event occurs between the previous state clock and the state clock at which the trigger point is marked.

If there is a rising edge and a falling edge on the same signal between the previous state clock and the state clock at which the trigger point is marked, such that the actual state of the signal is the same at both the previous state clock and the state clock at which the trigger point is marked, then the trigger will not occur.

In an advanced trigger step, you can set a trigger on edge on an individual bit/signal or multiple signals of a bus (edges are ORed in this case). The following screen displays an example of the Edge options available in an advanced trigger in timing mode.

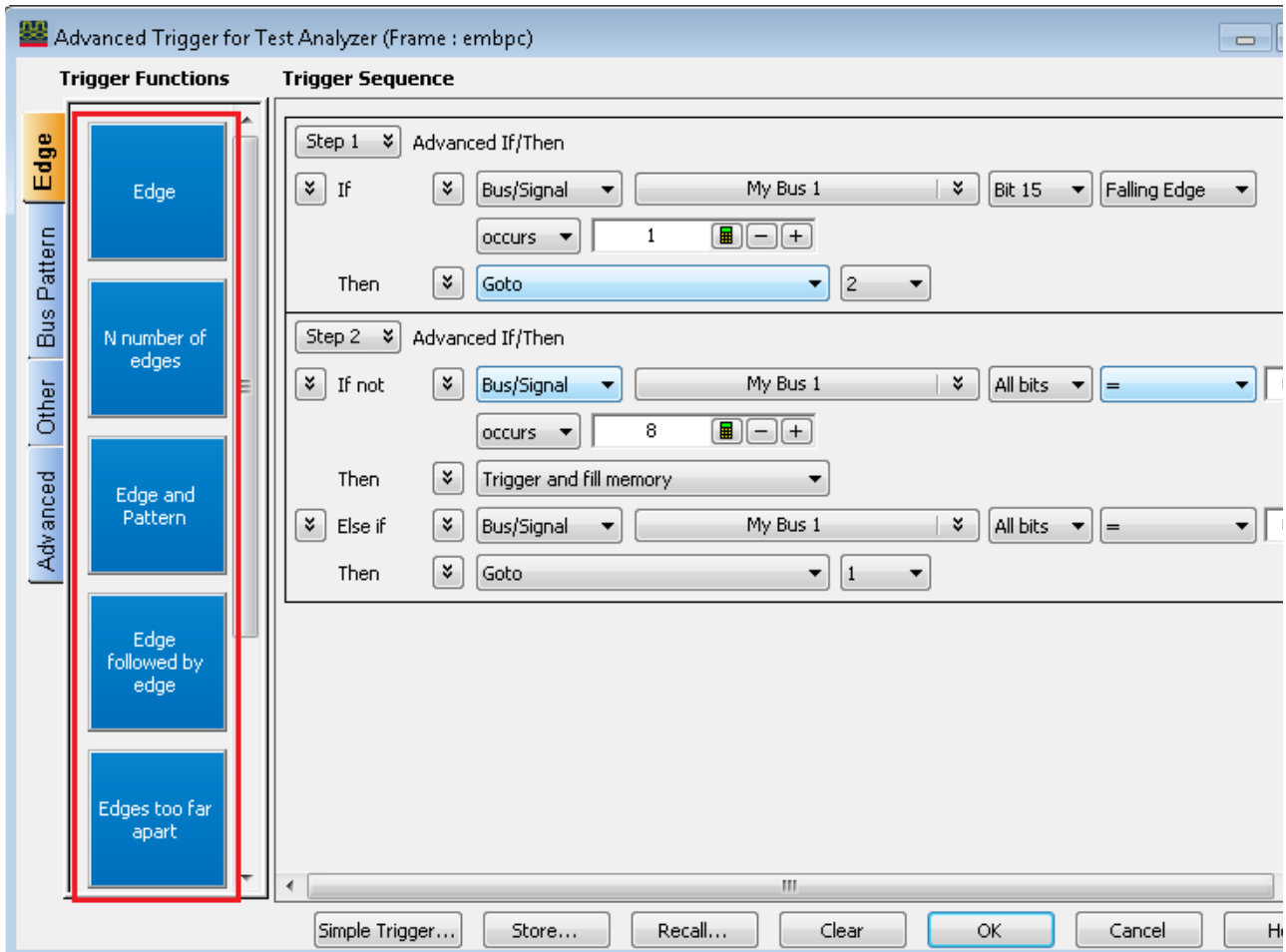
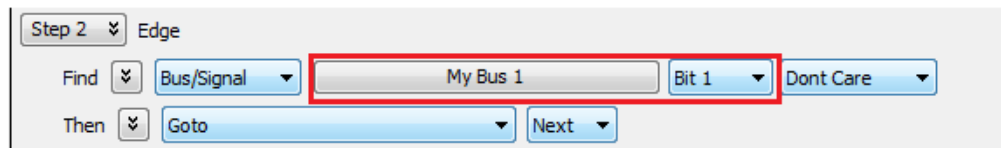


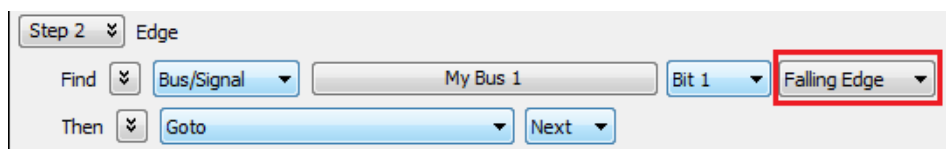
Figure 4 Trigger on edge options for the timing mode

To specify trigger on edge on a signal in an advanced trigger

- 1 In the Advanced Trigger dialog, select the bus and then its signal on which you want to set trigger on edge.

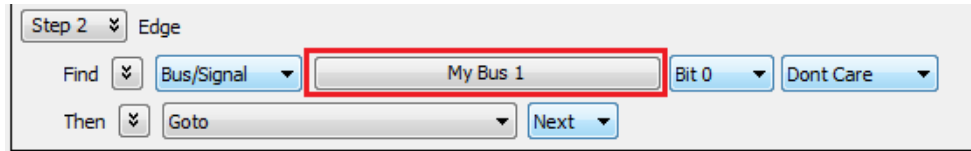


- 2 Select the edge value (**Rising Edge**, **Falling Edge**, or **Either Edge**) for the signal.

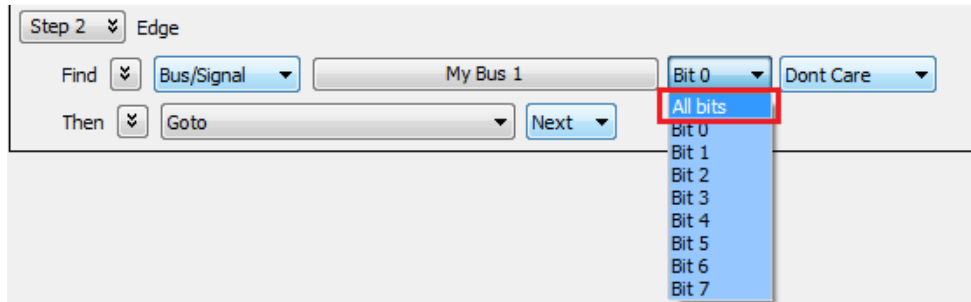


To specify bus/signal edges in advanced trigger

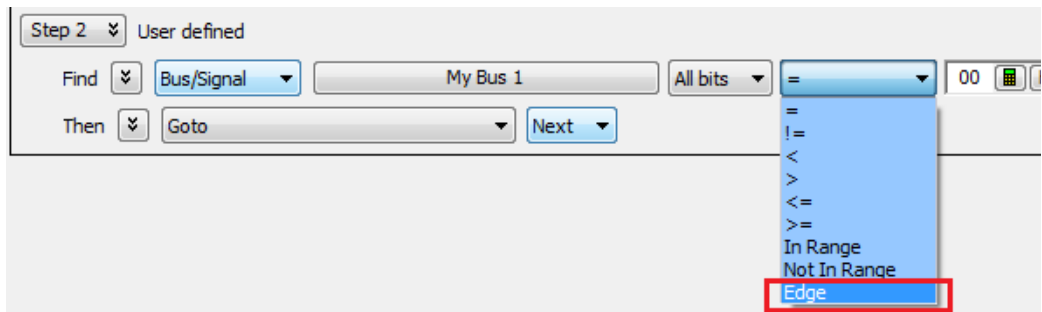
- 1 In the Advanced Trigger dialog, select the bus on which you want to set trigger on edge.



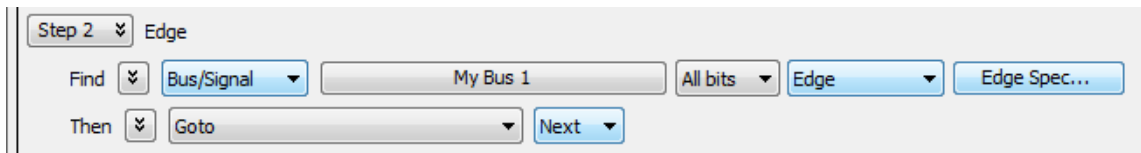
- 2 Select the **All bits** option for the bus.



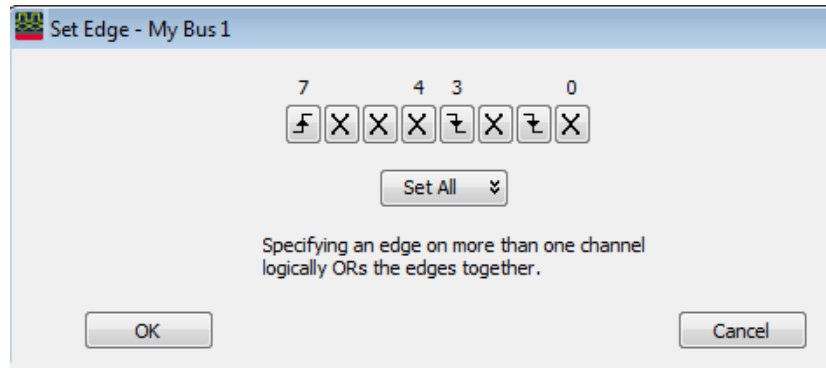
- 3 Select the **Edge** option.



When the **Edge** operator is selected, the **Edge Spec...** button is displayed.



- 4 Click the **Edge Spec...** button to open the **Set Edge** dialog for specifying multiple edges on a bus. In the Set Edge dialog, specify edges you are looking for; use the **Set All** button to make a selection for all signals in the bus.



5 Click **OK** to close the Set Edge dialog.

Trigger on edge in an eyescan trigger

You can also set a trigger on edge when configuring an eyescan trigger. This trigger allows you to control when logic analyzer should take samples to be used in the eyescan for determining the optimal threshold and sample positions.

To know more about eyescan triggers and how to configure these, refer to the topic [Modifying General or Target-specific Scan Qualification](#).

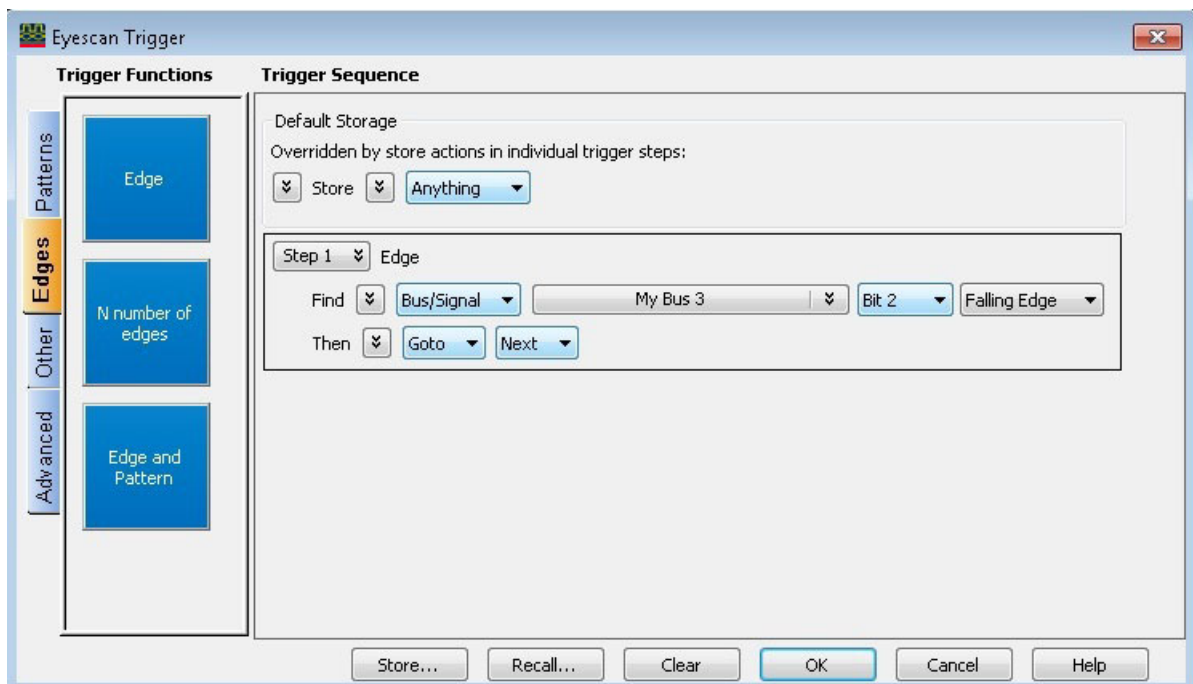


Figure 5 Trigger on edge in an eyescan trigger

- See Also
- To specify default storage (see [page 167](#))
 - To insert or delete events (see [page 168](#))
 - Understanding Logic Analyzer Triggering, Edges (see [page 373](#))

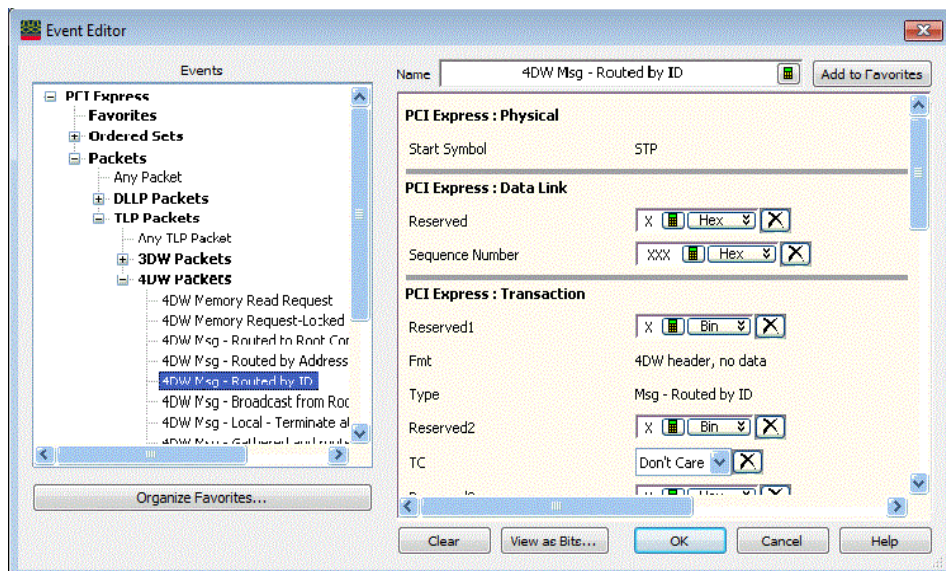
To specify packet events (in "Find a packet" trigger function)

- 1 In the Advanced Trigger dialog, replace or insert the Find a packet (see [page 496](#)) trigger function into the trigger sequence (see [To replace or insert trigger functions into trigger sequence steps](#) (see [page 158](#))).
- 2 In the "Find a packet" trigger sequence step, click **Select a bus**.
- 3 In the Choose a Protocol Family and Bus dialog (see [page 430](#)), select the protocol family and the type of bus being probed; then, click **OK**.
- 4 In the "Find a packet" trigger sequence step, click **Select a packet**.
- 5 In the Event Editor Dialog (see [page 433](#)), select the type of packet event and enter packet field values to trigger on; then, click **OK**.

See Also • Using the Packet Event Editor (see [page 164](#))

Using the Packet Event Editor

The packet event editor lets you specify packet events in the "Find a packet" trigger function.



To use the packet event editor:

- 1 Select the event type from the left side of the dialog.
- 2 Enter or select field values on the right side of the dialog.
To clear a field value, click .
- 3 If desired, you can enter or modify the **Name** of the event.
- 4 When you are done editing the packet event, click **OK**.

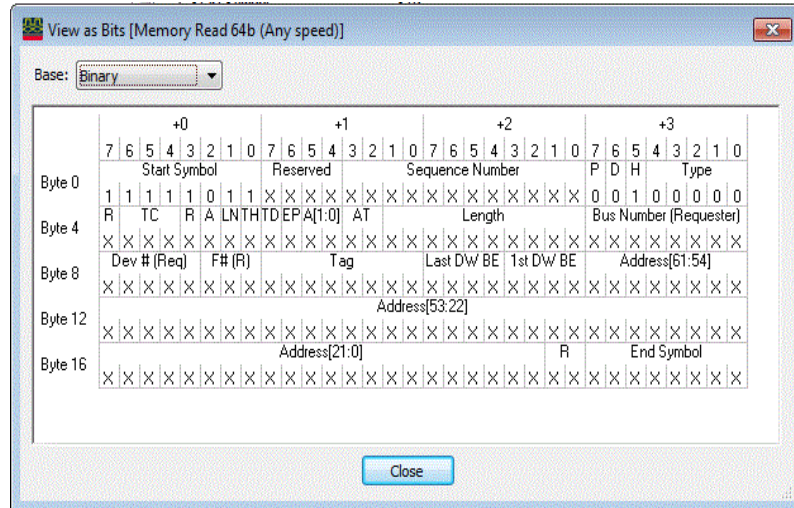
The packet event editor also allows you:

- To view a packet event as bits (see [page 165](#))
- To save favorite packet events (see [page 165](#))
- To organize favorite packet events (see [page 165](#))

See Also • Find a packet (see [page 496](#)) trigger function

To view a packet event as bits

- 1 While specifying packet events using the Event Editor Dialog (see [page 433](#)), click **View as Bits...**
The packet event is displayed in format similar to packet descriptions in specification documents.



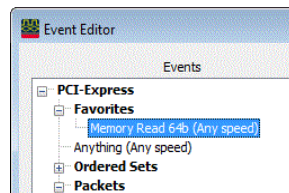
- 2 If desired, you can select a different number **Base**.
- 3 When you are finished viewing the packet event as bits, click **Close**.

See Also • To specify packet events (in "Find a packet" trigger function) (see [page 164](#))

To save favorite packet events

While specifying packet events using the Event Editor Dialog (see [page 433](#)), you can save the event as a favorite.

- 1 If desired, enter or modify the **Name** of the event.
- 2 Click **Add to Favorites**.
The packet event appears in the event list tree on the left side of the dialog.

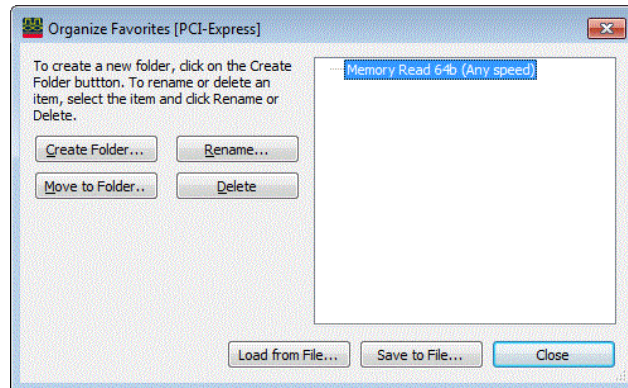


See Also • To organize favorite packet events (see [page 165](#))
• To specify packet events (in "Find a packet" trigger function) (see [page 164](#))

To organize favorite packet events

While specifying packet events using the Event Editor Dialog (see [page 433](#)), you can organize saved packet event favorites.

- 1 Click **Organize Favorites...**
- 2 In the Organize Favorites dialog, you can:



- Create folders, move selected events to folders, and rename or delete selected events.
 - Save favorites to a file, or load saved favorites from a file.
- 3 When you are done organizing packet event favorites, click **Close**.

- See Also
- To save favorite packet events (see [page 165](#))
 - To specify packet events (in "Find a packet" trigger function) (see [page 164](#))

To specify a trigger sequence step's goto or trigger action

- 1 In the Advanced Trigger dialog, within a sequence step, select the desired trigger action:

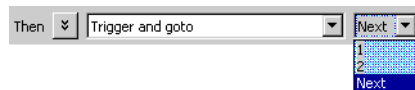


You can select:

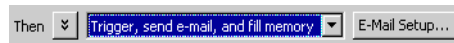
- **Goto**—To go to another trigger sequence step.



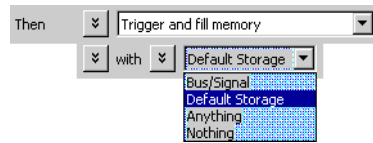
- **Trigger and fill memory**—To trigger the logic analyzer and fill memory, without going to any other steps in the trigger sequence.
- **Trigger and goto**—To trigger the logic analyzer and go to another trigger sequence step. (This can be useful when you use trigger sequence steps to specify what samples get stored.)



- **Trigger, send e-mail, and fill memory**—To trigger the logic analyzer, send e-mail, and fill memory, without going to any other trigger sequence steps. Clicking **E-Mail Setup...** opens the E-mail dialog (see [page 432](#)) for entering the e-mail address, subject, and message.



- 2 If you selected one of the actions that specify **fill memory** and you are in the state mode or in the store qualified timing mode with custom storage selected, enter the storage qualifier used to fill memory.



- See Also
- To insert or delete events (see [page 168](#))
 - To specify default storage (see [page 167](#))
 - To insert or delete actions (in a trigger sequence step) (see [page 179](#))

To specify default storage

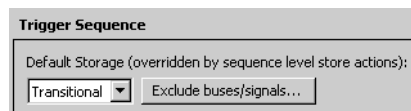
Storage qualifiers are used to specify which samples (captured from the device under test) are stored in logic analyzer memory. By storing only the samples you are interested in, you can make better use of the available memory and capture activity for a greater amount of time.

Default Storage means "unless *storage control* actions or *fill memory* storage qualifiers in trigger sequence steps specify otherwise, this is what should be stored". Storage qualifiers in trigger sequence steps always override default storage.

The default storage qualifier is available in state sampling mode and in the store qualified (transitional) timing mode.

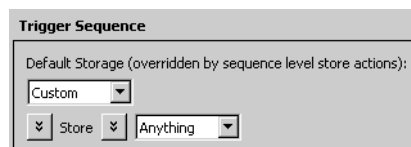
In Transitional / Store Qualified Timing Mode

- 1 Select the type of storage qualification: either **Transitional** or **Custom**.
 - When **Transitional** is selected, samples that have transitions from the previous sample are stored.

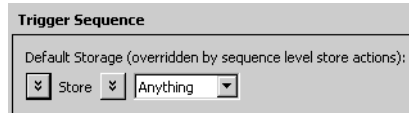


If you want to exclude certain bus/signal transitions from being stored, click **Exclude buses/signals...**, and specify which buses/signals should be excluded.

- When **Custom** is selected, edit or insert events that should be stored (or not stored) in logic analyzer memory.




In State Mode 1 Edit or insert events that should be stored (or not stored) in logic analyzer memory.



- See Also
- To insert or delete events (see [page 168](#))
 - To negate events (see [page 177](#))
 - To change the evaluation order of AND/OR'ed events (see [page 177](#))
 - To insert a storage control action (see [page 181](#)) (in a trigger sequence step)
 - To specify a trigger sequence step's goto or trigger action (see [page 166](#))
 - Understanding Logic Analyzer Triggering, Storage Qualification (see [page 375](#))

To insert or delete events

- 1 In the Advanced Trigger dialog, in a trigger sequence step, click the  button associated with an event (after **Find** or **If** in trigger sequence step conditions, or after **Store** or **with** in storage qualifiers), and choose **Insert Event After (AND/OR)**, **Insert Event Before (AND/OR)**, or **Delete Event**.



- 2 If inserting an event, select the type of event.



Depending on where you are inserting the event, the following event types may be available:

- **Bus/Signal**—Bus/signal value, To specify bus/signal patterns or edges (see [page 159](#)).
 - **Anything**—Any sample.
 - **Nothing**—No sample.
 - **Timer**—A timer value, see To insert a timer event (see [page 169](#)).
 - **Counter**—A counter value, see To insert a counter event (see [page 169](#)).
 - **Flag**—A flag value, see To insert a flag event (see [page 170](#)).
 - **Arm in from**—An arming signal from another logic analyzer module or an external instrument, see To insert an "Arm in from" event (see [page 170](#)).
- 3 If inserting an event, specify whether the event should be **And**'ed or **Or**'ed with the other events.



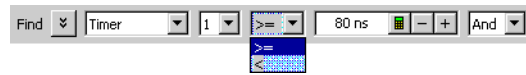
- See Also
- To negate events (see [page 177](#))
 - To change the evaluation order of AND/OR'ed events (see [page 177](#))
 - Understanding Logic Analyzer Triggering, Boolean Expressions (see [page 372](#))

To insert a timer event

- 1 Select the timer that you want to check.



- 2 Select the timer compare operator.



- 3 Enter the timer value to compare against.

NOTE

The **Start from reset** timer action can be placed in either the same sequence step as the timer check event, or it can be placed in a preceding trigger step. Checking a timer without starting it will generate an error.

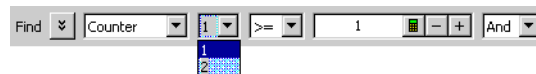
For more information on timers, see Using Timers (see [page 153](#)).

- See Also
- To insert or delete events (see [page 168](#))

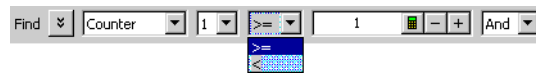
To insert a counter event

A counter must be started with a counter action before it can be evaluated with a counter event.

- 1 Select the counter that you want to check.



- 2 Select the counter compare operator.



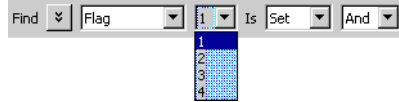
- 3 Enter the counter value to compare against.

For more information on counters, see Using Counters (see [page 154](#)).

- See Also
- To insert or delete events (see [page 168](#))

To insert a flag event

- 1 Select the flag that you want to check.



- 2 Enter the flag value to compare against.



There is approximately 100 ns of delay before a flag action can be seen by a flag event.

Multiple modules can insert flag events for the same flag to test whether the flag is set or clear.

For more information on flags, see Using Flags (see [page 157](#)).

See Also • To insert or delete events (see [page 168](#))

To insert an "Arm in from" event

- 1 Specify the source of the arming signal by selecting another module or **External trigger**.



For more information on triggering on signals from other logic analyzer modules or external instruments, see Triggering From, and Sending Triggers To, Other Modules/Instruments (see [page 186](#)).

See Also • To insert or delete events (see [page 168](#))

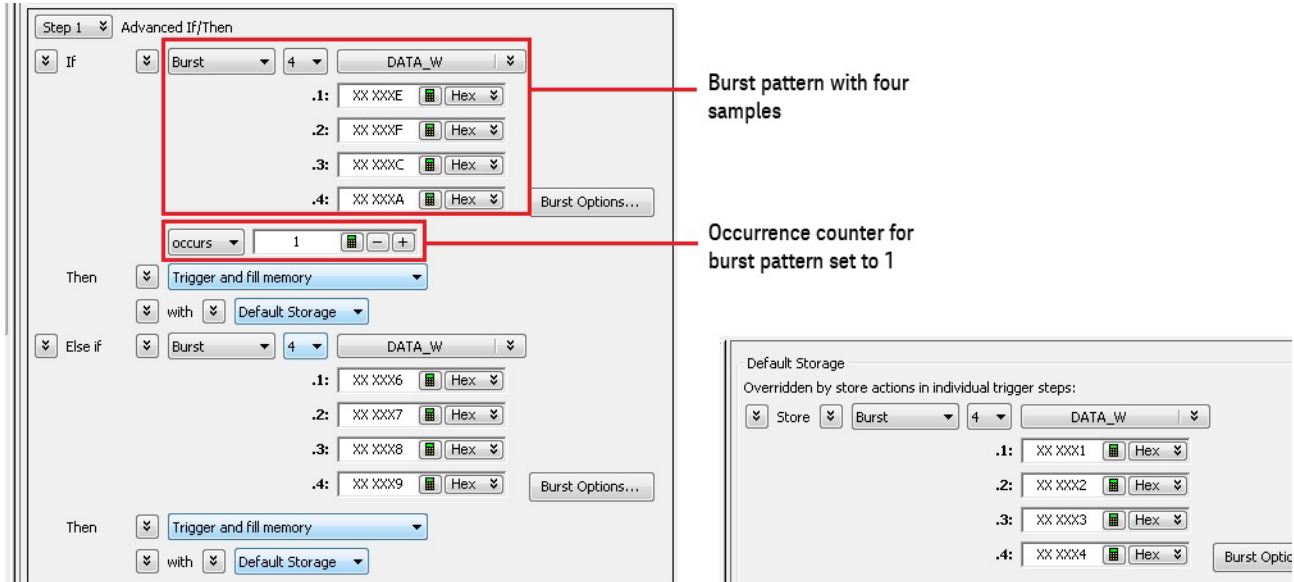
Using Burst Patterns

While configuring the trigger sequence step(s) of an Advanced trigger, you can use burst patterns to:

- define the "If" and "Else If" clauses in the trigger steps based on burst patterns.
- define the trigger's "default storage" and the "fill memory storage qualifiers in the trigger steps" based on burst patterns.

When using a burst pattern in an If/Else If clause, the trigger occurs when the complete burst pattern condition is met. If you have specified an occurrence counter greater than 1 for a burst pattern, then the trigger occurs when the complete burst pattern condition has consecutively occurred for the number of times specified as the occurrence counter.

When using a burst pattern in the trigger's storage qualification, the complete burst pattern found is stored.



To know how to specify burst patterns in "If", "Else If", or "storage qualification", refer to the topic ["To specify burst patterns"](#) on page 173.

Setting the Burst Mode for a Trigger Sequence

Depending on whether you want to use burst patterns in "storage qualification" or in "If/Else If" clauses in a trigger sequence, you need to set the burst mode matching the usage.

By setting the correct burst mode, you ensure that the burst patterns work as per their intended usage (described below).

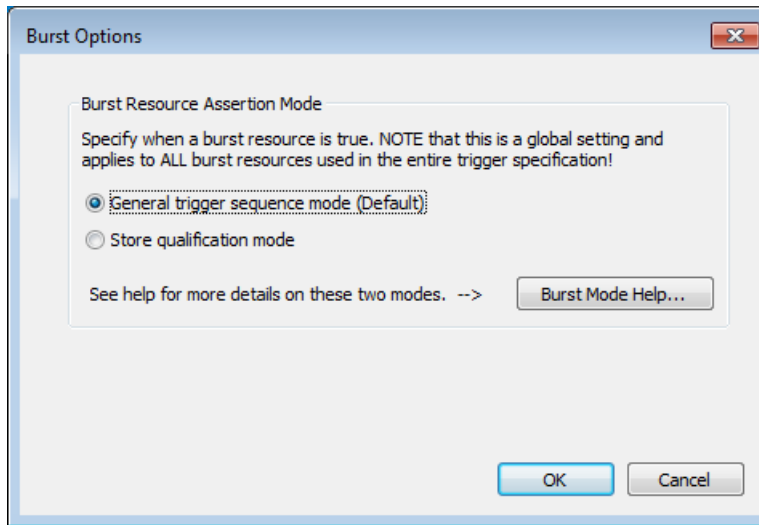
To set the burst mode, click the **Burst Options...** button displayed with a burst pattern in the trigger sequence.

NOTE

The burst mode that you set is applicable to all burst patterns that you use in a trigger sequence.

The **Burst Options...** button is available with each burst pattern that you use in a trigger sequence. Changing the mode by clicking any of these buttons reflects that change across all the burst patterns in the trigger sequence.

On clicking the **Burst Options...** button, the **Burst Options** dialog box is displayed. Select a burst mode from the two burst modes for the entire trigger sequence.



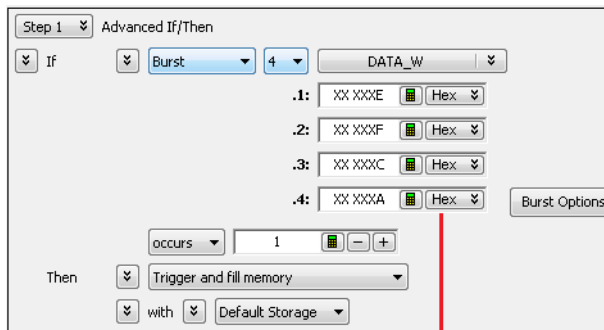
General trigger sequence mode	Store qualification mode
-------------------------------	--------------------------

Select this mode when you want to use burst patterns in "If/Else If" clauses in the trigger sequence steps.

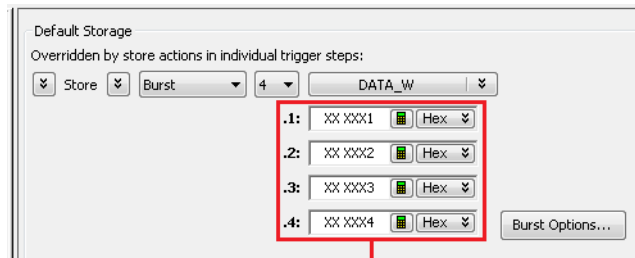
Select this mode when you want to use burst patterns in "default storage" or the "fill memory storage qualifiers in the steps".

On selecting this mode, the trigger occurs on the last sample that you specified in a burst pattern, that is, when the entire burst pattern condition is met. The trigger point is therefore, when the last sample specified in the burst pattern is found.

On selecting this mode, each sample of the burst pattern that is found in the trace is stored. The entire burst pattern that you specified in the default storage is therefore stored, when found in the trace. This default storage, however, is overridden by any storage actions that you set in individual trigger steps.



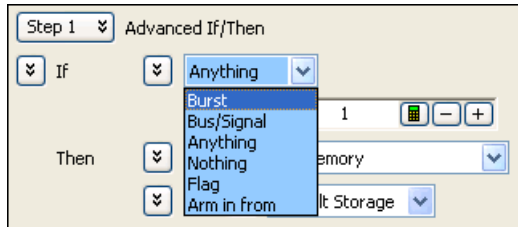
Trigger point is the last sample in the burst pattern



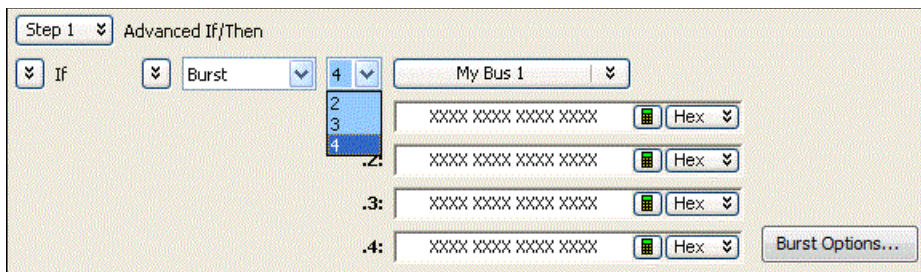
All the samples specified in the burst pattern are stored, when found.

To specify burst patterns

- 1 In the **Advanced Trigger** dialog, select the Burst event type.

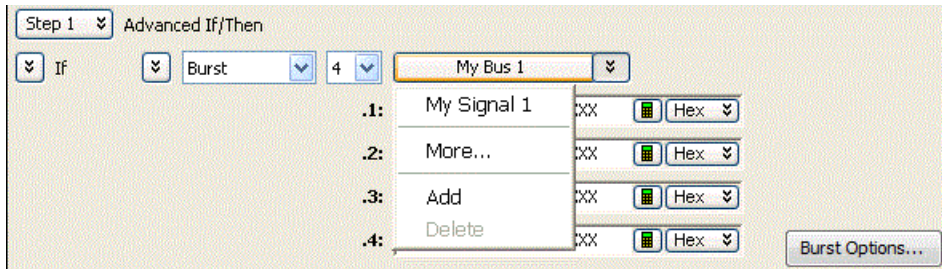


- 2 Select the number of samples in the burst pattern.



You can have up to eight 2-sample burst patterns or four 3- or 4-sample burst patterns.

- 3 Select the bus or signal.



Clicking  lets you:

- Select from recently used bus/signal names.
- Select from other bus/signal names (More...).
- Add another bus/signal to the burst pattern.

Additional buses/signals are AND'ed in the burst pattern. You can add as many buses/signals as you like, without consuming additional trigger resources.

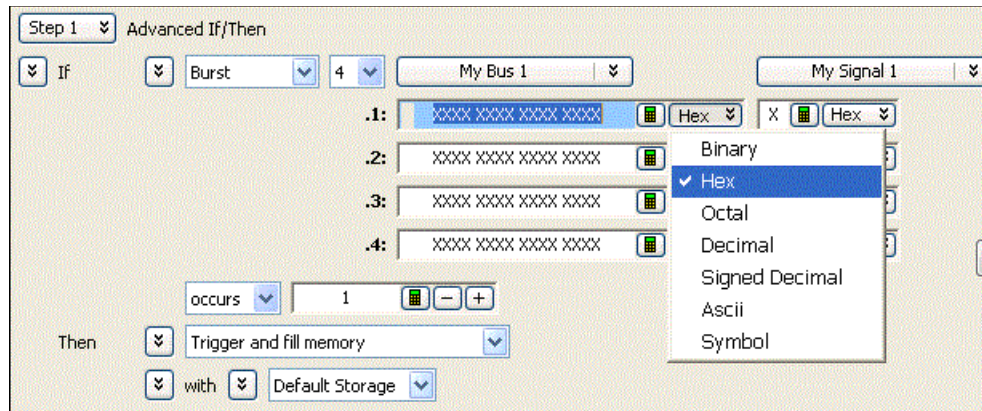
NOTE

Add buses/signals to the burst pattern instead of inserting AND'ed events. AND'ed events consume trigger resources.

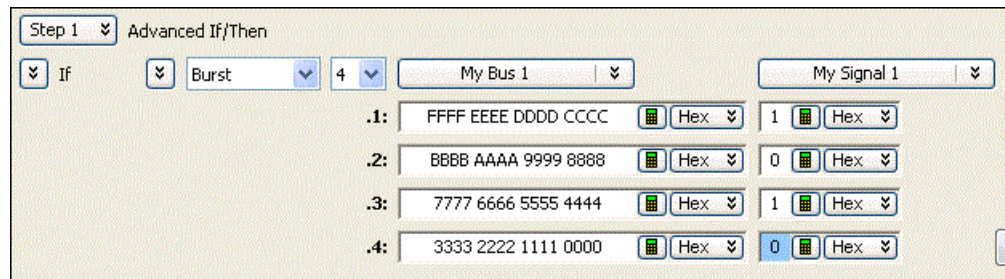
- Delete the bus/signal from the burst pattern.

Clicking elsewhere on a bus/signal name button opens a Select dialog for selecting a different name.

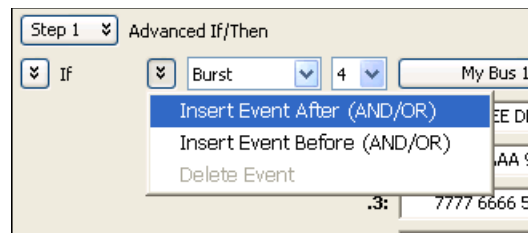
- 4 Specify the burst pattern values:
 - a Select the number base (Binary, Hex, Octal, Decimal, Signed Decimal, also known as two's complement, Ascii, or Symbol).



- b Enter the burst pattern value(s).
When the Symbol number base is selected, you use the Select Symbol dialog to specify the pattern values.



- 5 You can insert events AND'ed or OR'ed with the burst pattern.



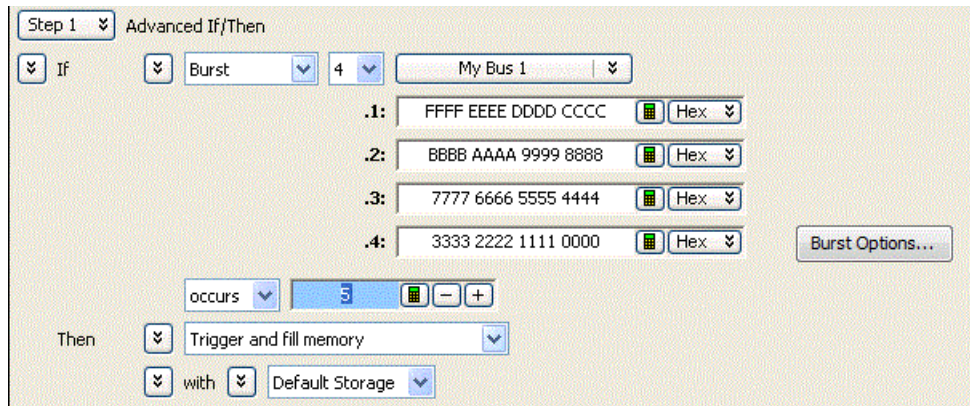
As previously mentioned, instead of inserting AND'ed events, add buses/signals to the burst pattern whenever possible to conserve trigger resources.

You can OR burst patterns to effectively increase the sample depth of a pattern (for example, find 3 occurrences of burst A OR burst B OR burst C) provided that the device under test does not send the bursts out of order (or you don't care if it does).

Burst Patterns in "If" and "Else If" Clauses

"If" Clauses

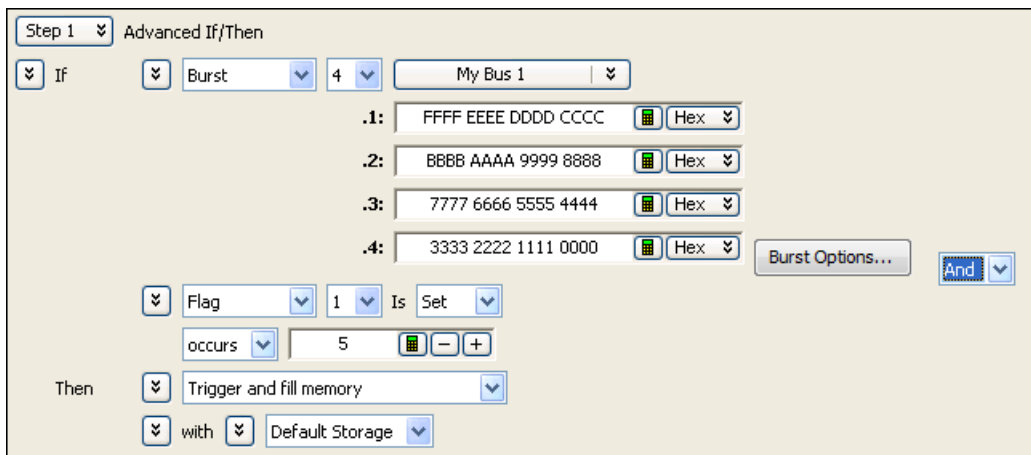
The occurrence counter associated with the "If" clause in a trigger sequence step takes action on complete burst patterns. For example, you can look for the fifth eventual occurrence of a 4-sample burst:



The "If" clause action occurs after the last sample of the last burst.

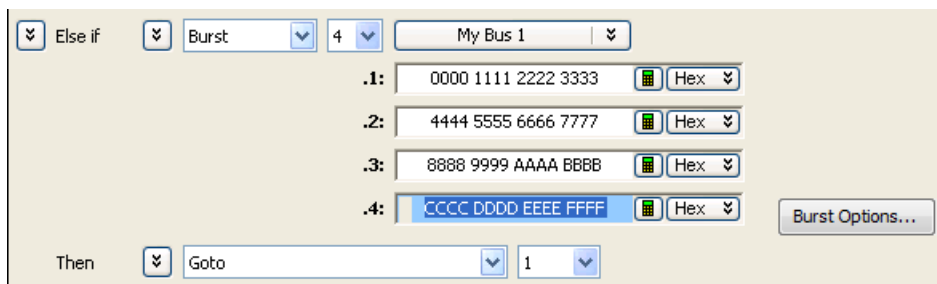
AND'ed events consider complete burst patterns. In the following example, the action occurs after:

- 5 complete bursts during which the flag is set.



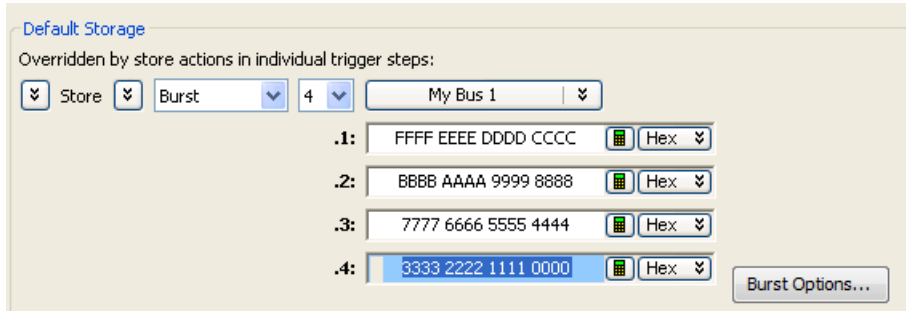
"Else If" Clauses

Burst patterns used in "Else If" clauses work the same way as the burst patterns in "If clauses". Unlike "If clauses", you cannot set an occurrence counter for a burst pattern in "Else if" clauses.

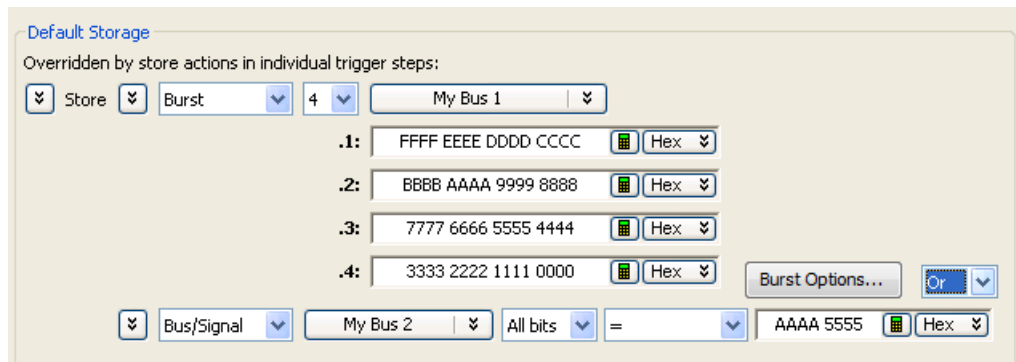


Burst Patterns in Default Storage

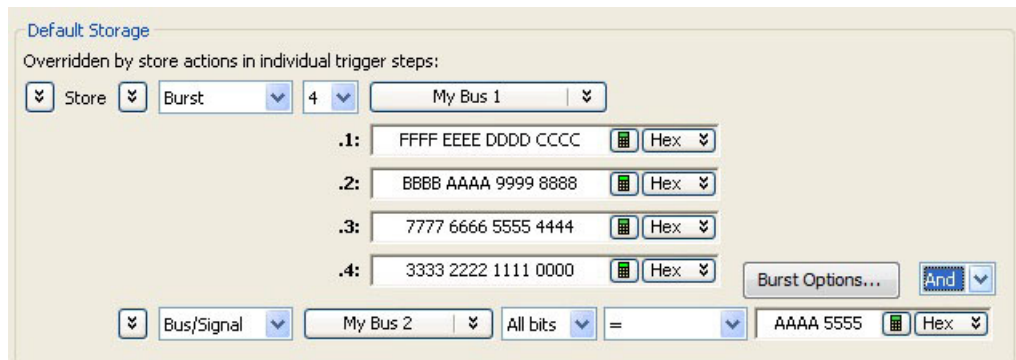
When a single burst pattern event is specified as default storage, all burst samples that you specify in that pattern are stored, when found.



When a burst pattern is OR'ed with other events, the union of burst samples and other event samples are stored.




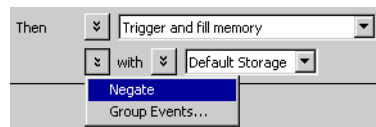
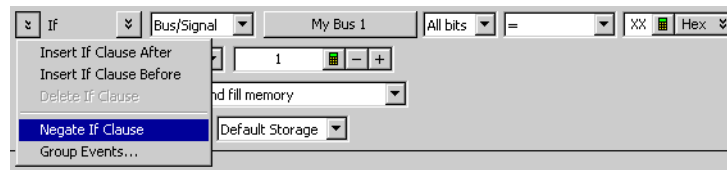
When a burst pattern is AND'ed with other events, the intersection of burst samples and other event samples are stored.



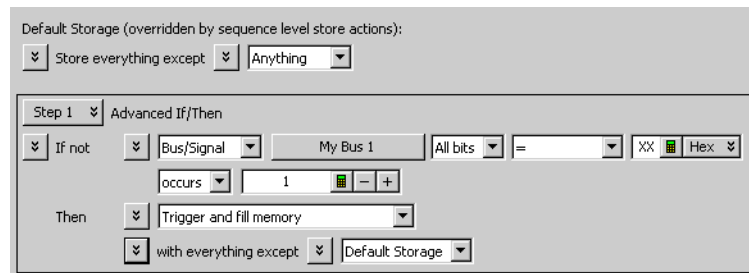
To negate events

Everywhere in the Advanced Trigger dialog where you can edit or insert events, you can also negate the events.

- 1 Click the  button associated with the events, and choose **Negate**.



Text in the dialog changes to indicate that events are negated.

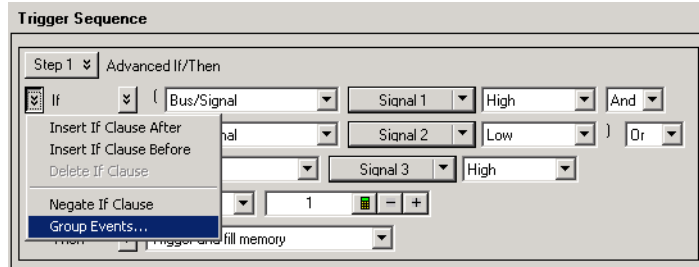


See Also [To insert or delete events \(see page 168\)](#)

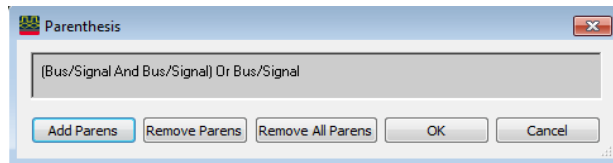
To change the evaluation order of AND/OR'ed events

When specifying advanced triggers (or after converting trigger functions to advanced *if/then* steps) and there are multiple events in an event list, you can specify their evaluation order by grouping the events.

- 1 In the Trigger tab's Trigger Sequence area, select the *If*, *If not*, *Else if*, or *Else if not* button; then, choose **Group Events...**



- 2 In the Parenthesis dialog, either select **Add Parens** button to group events or select **Remove Parens** to ungroup events.



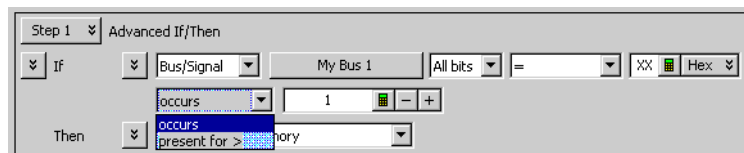
- 3 When you have finished grouping events, click **OK**.

See Also • To insert or delete events (see [page 168](#))

To choose between a duration or occurrence count for events (timing mode)


When specifying advanced triggers in the timing mode, you can choose between specifying an occurrence count for events or a time that the events must be present for.

- 1 In the trigger sequence step, select either **occurs** or **present for >** to change the setting.



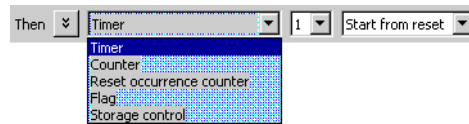
See Also • To convert a trigger sequence step to Advanced If/Then trigger functions (see [page 183](#))

To insert or delete actions (in a trigger sequence step)

- 1 In the Advanced Trigger dialog, in a trigger sequence step, click the  button associated with an action (after **Then**), and choose **Insert Action After**, **Insert Action Before**, or **Delete Action**.



- 2 If inserting an action, select the type of action.



The following action types are available:

- **Timer**—For starting, stopping, pausing, or resuming a timer, see To insert a timer action (see [page 179](#)).
- **Counter**—For incrementing or resetting a counter, see To insert a counter action (see [page 180](#)).
- **Reset occurrence counter**—For resetting the occurrence counter, see To insert a reset occurrence counter action (see [page 180](#)).
- **Flag**—For setting or clearing a flag, see To insert a flag action (see [page 180](#)).
- **Storage control**—For storing samples or not or for turning on/off default storage, see To insert a storage control action (see [page 181](#)).

See Also • To insert or delete events (see [page 168](#))

To insert a timer action

- 1 Select the timer that you want to specify an action for.



- 2 Specify the timer action by selecting either **Start from reset**, **Stop and reset**, **Pause**, or **Resume**.



NOTE

The **Start from reset** timer action can be placed in either the same sequence step as the timer check event, or it can be placed in a preceding trigger step. Checking a timer without starting it will generate an error.

For more information on timers, see Using Timers (see [page 153](#)).

See Also · To insert or delete actions (in a trigger sequence step) (see [page 179](#))

To insert a counter action

- 1 Select the counter that you want to specify an action for.



- 2 Specify the counter action by selecting either **Increment** or **Reset**.

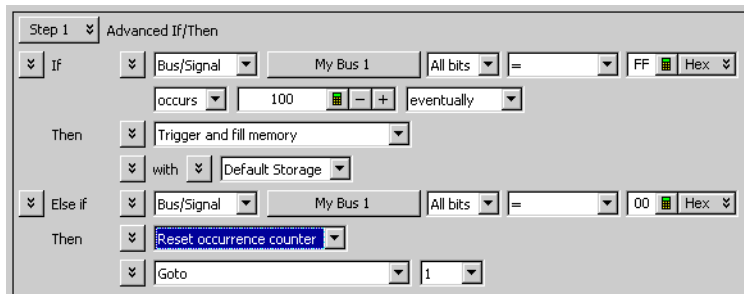


For more information on counters, see Using Counters (see [page 154](#)).

See Also · To insert or delete actions (in a trigger sequence step) (see [page 179](#))

To insert a reset occurrence counter action

The trigger sequence step below shows how the **Reset occurrence counter** action is used.



While searching for a number of occurrences of one event, if some other event is found, you can reset the occurrence counter and restart the search.

See Also · To insert or delete actions (in a trigger sequence step) (see [page 179](#))
 · Understanding Logic Analyzer Triggering, Occurrence Counters (see [page 373](#))

To insert a flag action

- 1 Select the flag that you want to specify an action for.



- 2 Specify the flag action by selecting either **Set**, **Clear**, **Pulse set**, or **Pulse clear**.



Flags in pulse mode sit in the opposite state when not being pulsed. If you insert a **Pulse set** action for a flag in one module, you cannot insert a **Pulse clear** action for the same flag in a different module.

NOTE

Within a module, the same flag cannot be used in both pulse and level (Set/Clear) modes. If a flag action is inserted or modified with a different mode than other actions for the same flag, all actions for that flag will change to match the new mode.

In case of multiple modules, legacy logic analysis systems allow a particular flag to be set by multiple modules. However, in newer AMP/AXIe based instrument modules, only one module can set a particular flag. Multiple modules cannot set the same flag.

A flag that is set by a module remains set until that module clears it. In legacy logic analysis systems, if multiple modules set the same flag, then all those modules must clear the flag before it becomes clear. In newer AMP/AXIe based instrument modules, multiple modules cannot set the same flag and therefore, the module that sets the flag can only clear it.

3 If you selected **Pulse set** or **Pulse clear**, enter the pulse width.

NOTE

Within a module, a flag's pulse width must be the same in every action for that flag. Whenever the pulse width is changed in a flag action, it changes in all other actions for that flag.

For more information on flags, see Using Flags (see [page 157](#)).

See Also • To insert or delete actions (in a trigger sequence step) (see [page 179](#))

To insert a storage control action

1 Specify the storage control action by selecting either **Store sample**, **Don't store sample**, **Turn on default storage**, or **Turn off default storage**.

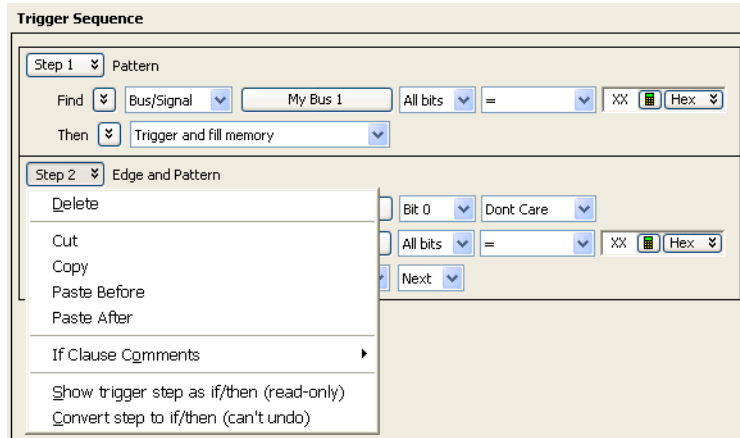


For more information on sequence step storage and storage control actions, see Understanding Logic Analyzer Triggering, Storage Qualification (see [page 375](#)).

See Also • To insert or delete actions (in a trigger sequence step) (see [page 179](#))

To cut, copy, and paste trigger sequence steps

- 1 In a trigger sequence step, click **Step N** and choose **Cut**, **Copy**, **Paste Before**, or **Paste After**.

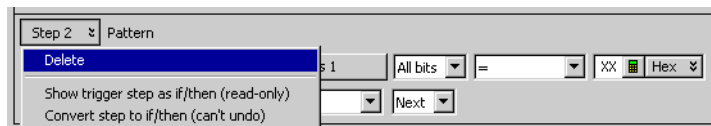


There must be at least one step in the trigger sequence.

See Also • To delete trigger sequence steps (see [page 182](#))

To delete trigger sequence steps

- 1 In a trigger sequence step, click **Step N** and choose **Delete**.



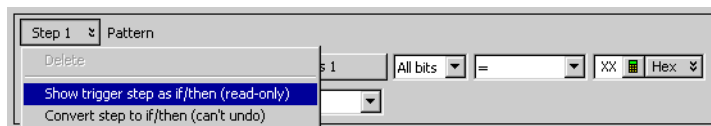
There must be at least one trigger sequence step.

To show a trigger sequence step as Advanced If/Then trigger functions

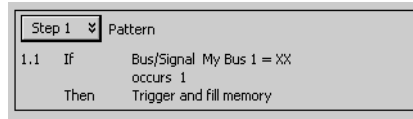
NOTE

The Advanced (If/Then or N-Way Branch) trigger functions do not allow alternative display types. By default, they are in the expanded graphical form that cannot be changed.

- 1 In a trigger sequence step, click **Step N** and choose **Show trigger step as if/then (read-only)**.

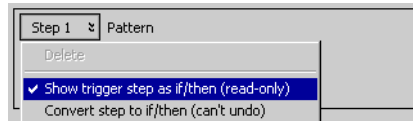


The trigger sequence step will be shown as the equivalent Advanced If/Then trigger functions in read-only form.



To undo a trigger sequence step shown as Advanced If/Then trigger functions

- 1 In a trigger sequence step, click **Step N** and choose the checked **Show trigger step as if/then (read-only)** item to return to the normal view of the trigger function.



See Also • To convert a trigger sequence step to Advanced If/Then trigger functions (see [page 183](#))

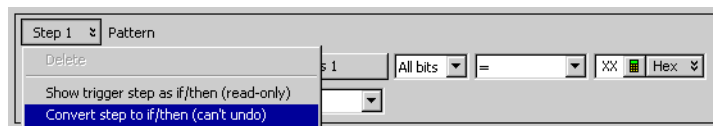
To convert a trigger sequence step to Advanced If/Then trigger functions

If the trigger function you need doesn't exist, start with a trigger function that is close, convert the trigger sequence step to advanced If/Then trigger functions, and edit the If/Then trigger functions.

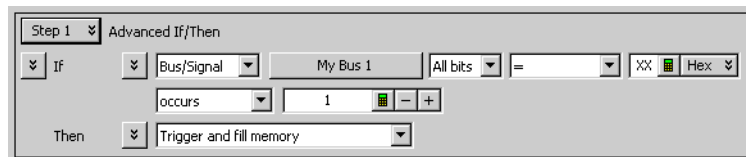
NOTE

The Advanced (If/Then or N-Way Branch) trigger functions do not allow alternative display types. By default, they are in the expanded graphical form that cannot be changed.

- 1 In a trigger sequence step, click **Step N** and choose **Convert trigger step to if/then (can't undo)**.



The trigger sequence step will be converted to the equivalent Advanced If/Then trigger functions.



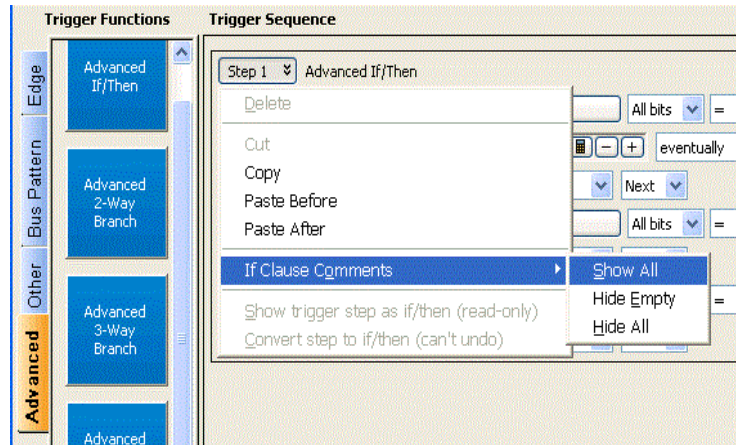
See Also • To show a trigger sequence step as Advanced If/Then trigger functions (see [page 182](#))
 • Understanding Logic Analyzer Triggering, Branches (see [page 373](#))

To display or hide "If" clause comments

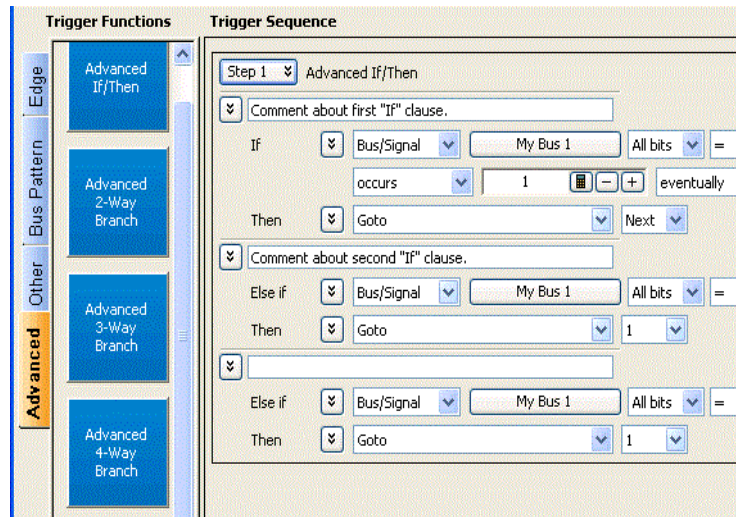
When using Advanced If/Then trigger functions, you can include comments with the "If" clauses in a trigger sequence step. It is useful to have descriptions in complex trigger functions.

To show all "If" clause comments

- 1 In a trigger sequence step, click **Step N** and choose **If Clause Comments>Show All**.

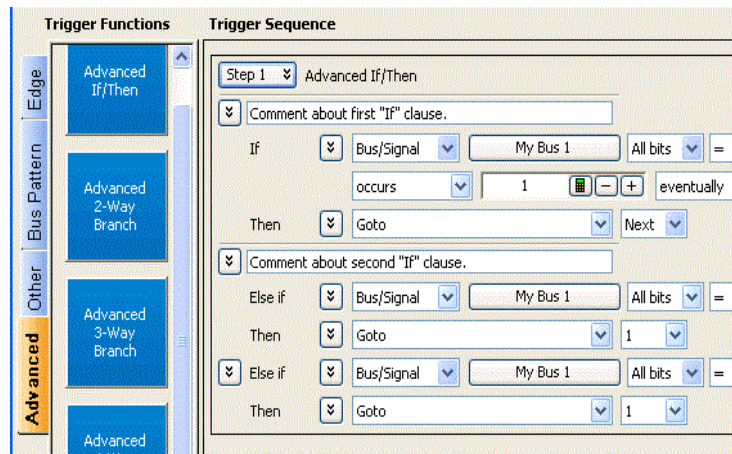


All "If" clause comments are show, including empty comments.

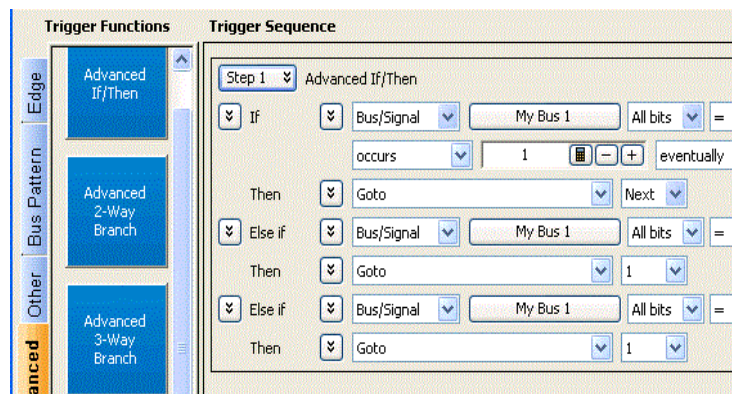


To hide empty "If" clause comments

- 1 In a trigger sequence step, click **Step N** and choose **If Clause Comments>Hide Empty**. Empty "If" clause comments are hidden.



- To hide all "If" clause comments
- 1 In a trigger sequence step, click **Step N** and choose **If Clause Comments>Hide All**. All "If" clause comments are hidden.



See Also · To convert a trigger sequence step to Advanced If/Then trigger functions (see [page 183](#))

To clear the trigger sequence

- 1 In the Advanced Trigger dialog, click **Clear** at the bottom of the dialog.

Triggering From, and Sending Triggers To, Other Modules/Instruments

ARM In from modules

You can cause the trigger from one module to arm another module installed in the same chassis or in a chassis/logic analysis system (interconnected via multi-frame cables in a multi-frame setup). Refer to the topic ["To arm one module with another module's trigger"](#) on page 186 to know about ARM In feature in detail.

External triggering


You can also send a module's trigger signal to an external instrument, or you can allow a signal from an external instrument to arm a module.

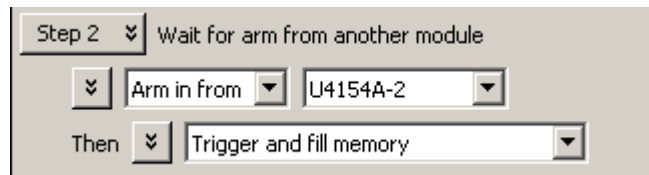
There are Trigger In and Trigger Out BNC connectors located on the logic analyzer (front panel of AXIe chassis for the U4154A/B module). Use these connectors to connect the analyzer to an external instrument and either send or receive a trigger signal. To know more about external triggering, refer to the topics ["To trigger other instruments - Trigger Out"](#) on page 191 and ["To trigger analyzer from another instrument - Trigger In"](#) on page 192.

To arm one module with another module's trigger

A module can receive ARM In from a single or multiple modules installed in the same chassis or in an interconnected chassis of a multiframe setup. To use this ARM In feature, you use the *Wait for arm from another module* trigger function in the Advanced Trigger dialog. This topic describes various ARM In scenarios.

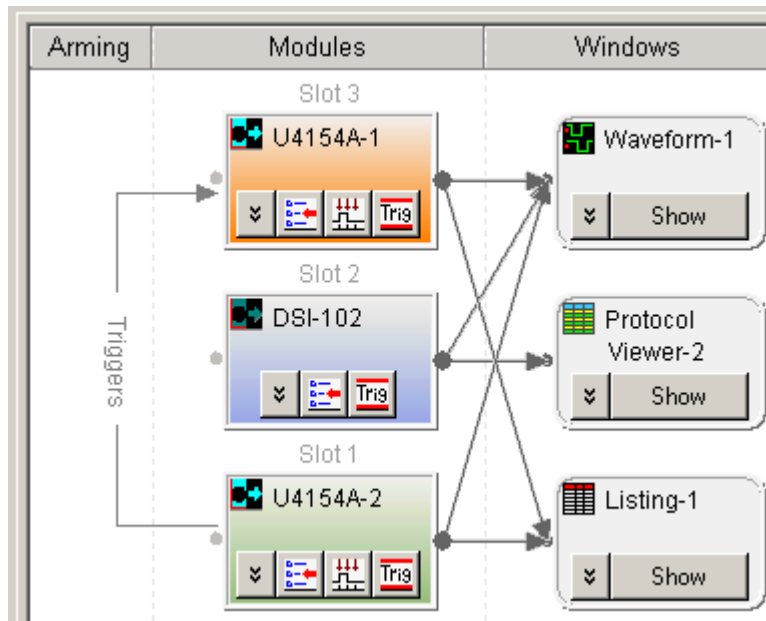
ARM In from a module to another module in the same/different chassis

- 1 Click  in the analyzer setup toolbar, or choose **Setup>(Logic Analyzer Module)>Advanced Trigger...** from the menu bar.
- 2 In the Advanced Trigger dialog (see [page 424](#)), select the **Other** trigger functions tab; then, drag-and-drop the **Wait for arm from another module** trigger function into the trigger sequence area.



- 3 From the module name drop-down, select the module whose trigger will arm this module (and satisfy the event condition in the trigger sequence step).
- 4 Click **OK** in the Advanced Trigger dialog.

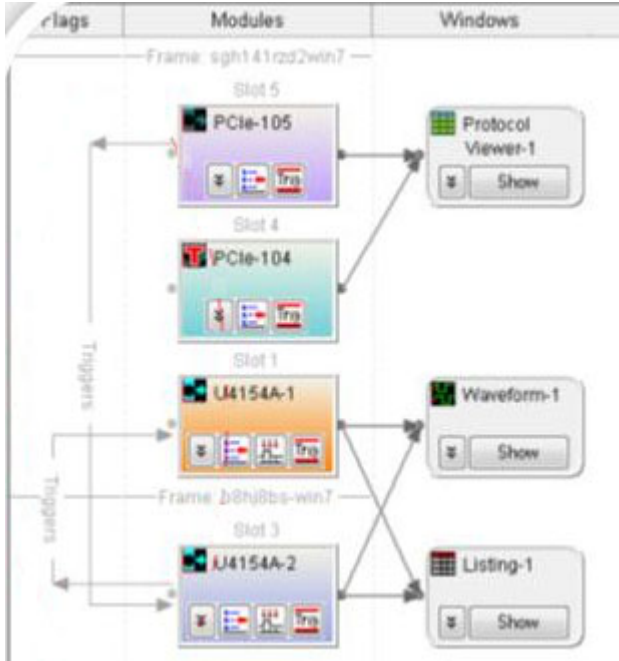
The arming setup in the Overview window looks like:



- Run the measurement (see Running/Stopping Measurements (see [page 196](#))).
When the arm from another module (U4154A/B-2 in this case) is received, the module (U4154A/B-1 in this case) takes the action(s) described in its trigger sequence step.

NOTE

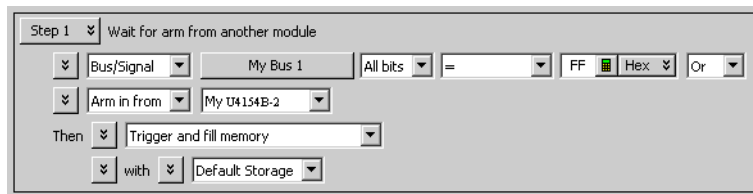
You can create an ARM In trigger in the same manner (as described above) between modules installed in different chassis of a multiframe setup. In a multiframe setup, flags are consumed for each ARM IN trigger configuration that you create. There are 4 flags available. If all these flags are unused, then you can create a maximum of four ARM In configurations across the connected chassis of the multiframe chain. An example of two ARM In configurations in a multiframe setup is displayed below.



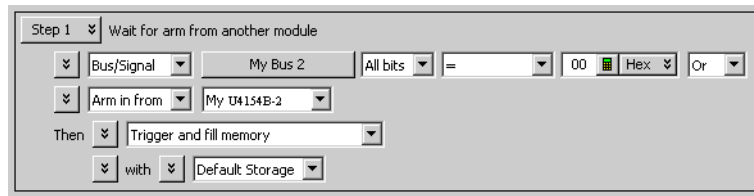
And vice-versa

You may have a situation where you have two modules looking for trigger events, and when either module finds its trigger event, the other should be armed. To do this:

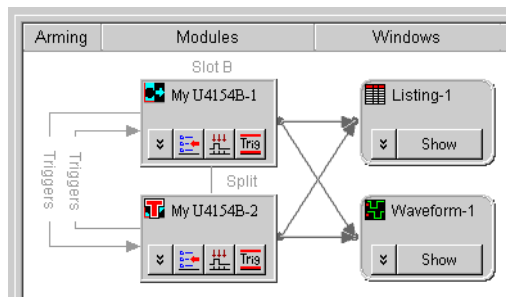
- 1 Set up the first module's trigger:



- 2 Set up the second module's trigger:



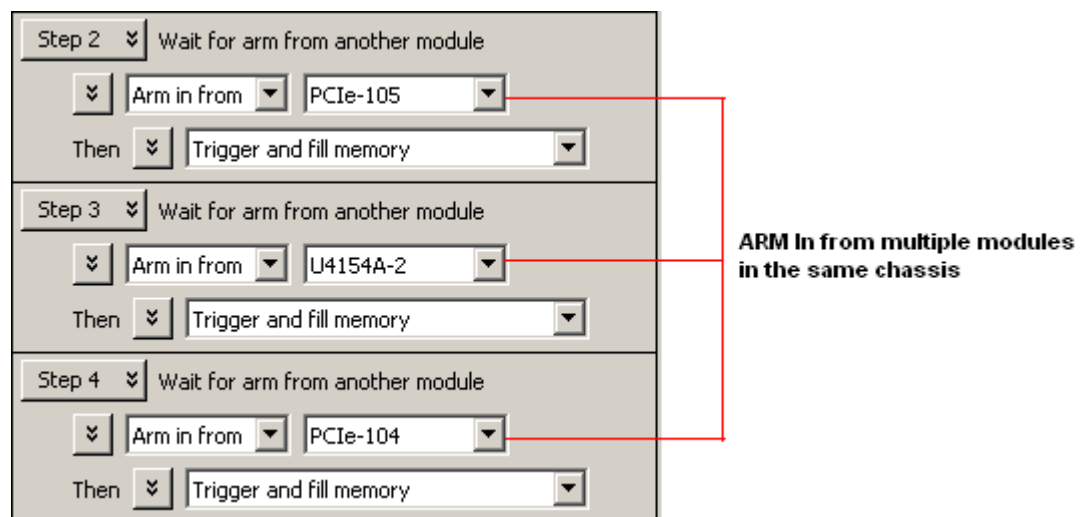
The arming setup in the Overview window looks like:



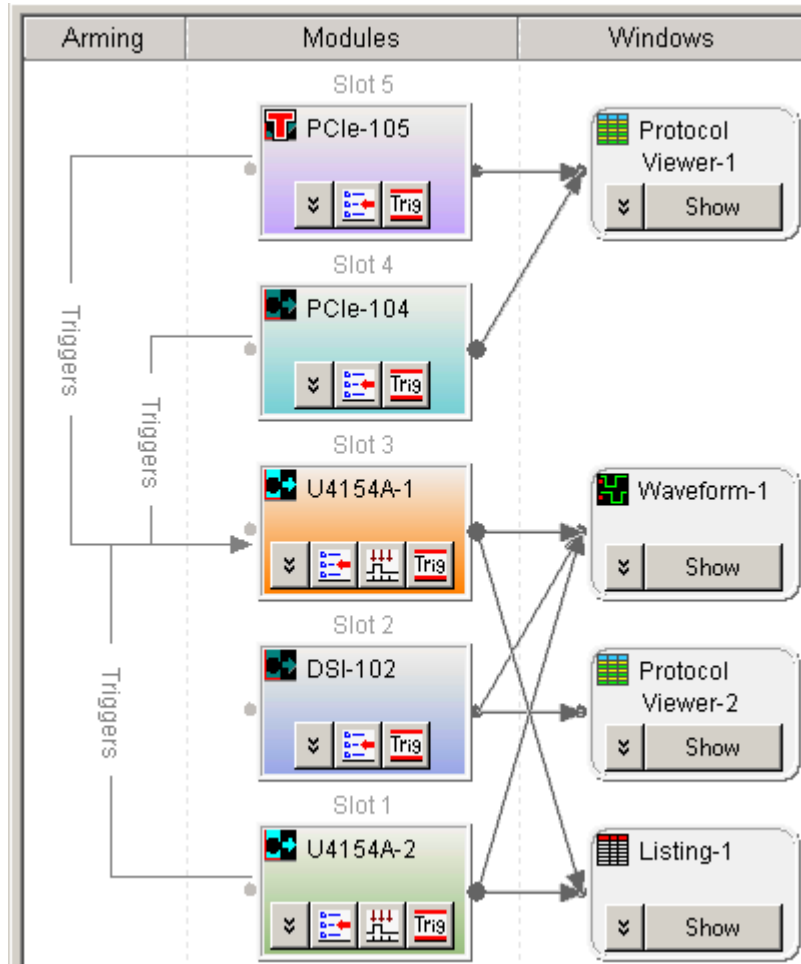
3 Run the measurement.

ORed ARM In from multiple modules to a module in the same chassis

You can set the trigger configurations of a module to receive ARM In from multiple modules in the same chassis. On doing so, the trigger configuration acts as an ORed combination of ARM In from multiple modules. Any of the selected modules can send ARM In to trigger the module. In such a scenario, the trigger configuration of the module receiving the ARM In consists of multiple *Wait for arm from another module* trigger sequence steps as displayed below.



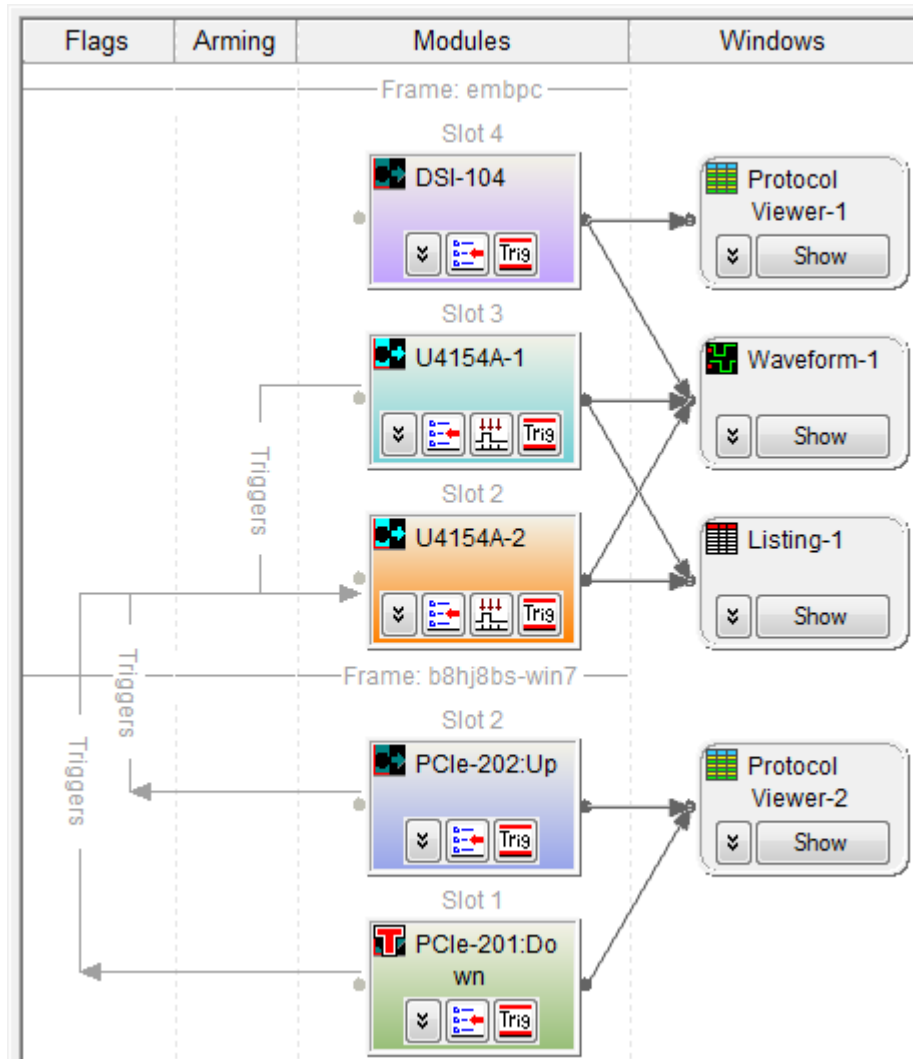
The arming setup in the Overview window looks like:



As displayed in the above figure, an ORed ARM In trigger is represented by multiple unidirectional lines merging together before pointing to the destination module receiving the ARM In.

ORed ARM In from multiple modules to a module in a multiframe setup

You can also set the ORed ARM In for a module from multiple modules in other connected chassis of a multiframe setup. To create an ORed ARM In in a multiframe scenario, you need to create multiple **Wait for arm from another module** trigger sequence steps for the receiving module. The following screen displays the ORed ARM In trigger for the U4154A/B-2 modules from three modules in a multiframe setup.



- See Also
- Wait for arm from another module (state) (see [page 499](#))
 - Wait for arm from another module (timing) (see [page 487](#))

To trigger other instruments - Trigger Out

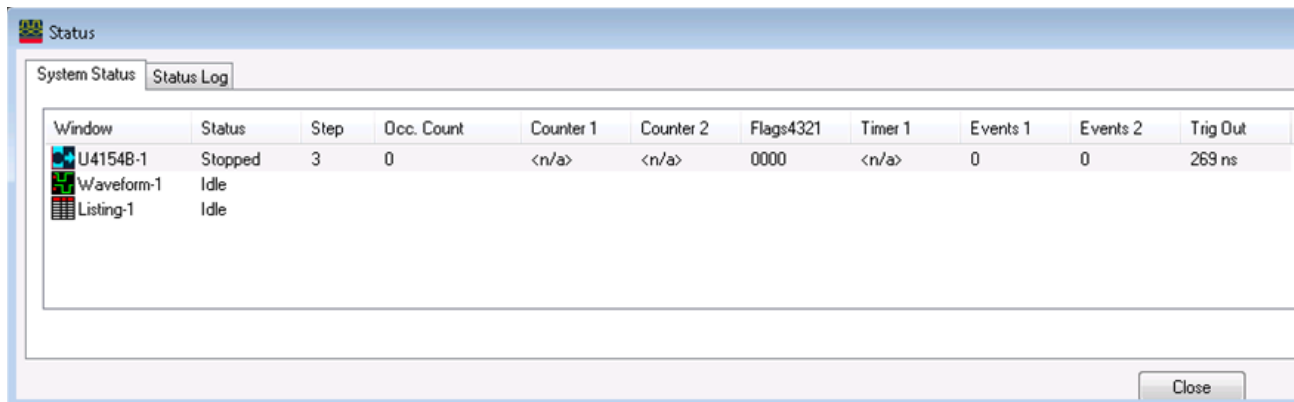
- 1 Connect a BNC cable from the **Trigger Out** BNC to the external instrument you want to trigger.
- 2 Choose **Setup>External Trigger...**
- 3 In the External Trigger dialog (see [page 437](#)):
 - Enable the output.
 - Select the polarity (active high or active low).
 - Select the output mode (use feedthrough to see flag settings on the output).
 - Select the trigger and flag events that cause **Trigger Out**.
 - Click **OK**.
- 4 Configure the logic analyzer as you would normally for any other measurement.

- When the analyzer's trigger sequence becomes true and the analyzer triggers, a trigger signal is sent out through the **Trigger Out** BNC to the external instrument.

Correlating the external instrument to your logic analyzer

If you are using a 16850-series or a U4154-series logic analyzer then the **Trig Out** value in the **Status** dialog will help you correlate the external instrument to your 16850-series or U4154-series analyzer.

The Trig Out column in the Status dialog box provides a time measurement that you can use to correlate the trace captured on the logic analyzer with a second analyzer or a scope.



This time represents the time interval from a trigger condition appearing on a probe tip of the analyzer to the occurrence of a trigger pulse on the Trigger Out BNC.


This value varies depending upon the number of channels in the analyzer and the State speed of the target or the Timing Speed selected for the measurement. Knowing this Trig Out time will allow you to properly correlate an external scope or other analyzer.

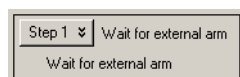
This time is calculated only for the 16850-series and the U4154- series of analyzers.

If you are using a Keysight multiframe cable between your analyzers or you have added an external Keysight Oscilloscope with the "Setup->Add External Oscilloscope" to your measurement, then the correlation of the instruments will be determined automatically.

See Also · External Trigger Dialog (see [page 437](#))

To trigger analyzer from another instrument - Trigger In

- Connect a BNC cable from the **TRIGGER IN** BNC to the external instrument that will send the trigger signal.
- Choose **Setup>External Trigger...**
- In the External Trigger dialog (see [page 437](#)), specify whether a rising or falling edge on the **Trigger In** BNC will indicate a trigger; then, click OK.
- Click  in the analyzer setup toolbar, or choose **Setup>(Logic Analyzer Module)>Advanced Trigger...** from the menu bar.
- In the Advanced Trigger dialog (see [page 424](#)), select the **Other** trigger functions tab; then, drag-and-drop the **Wait for external arm** trigger function into the trigger sequence area.



- 6 Click **OK** in the Advanced Trigger dialog, and run the measurement (see Running/Stopping Measurements (see [page 196](#))).
- 7 Run the measurement on the external instrument.
When the arm from the external instrument is received, the logic analyzer takes the actions described in the trigger sequence step.

- See Also
- Wait for external arm (state) (see [page 498](#))
 - Wait for external arm (timing) (see [page 486](#))
 - External Trigger Dialog (see [page 437](#))

Storing and Recalling Triggers

Triggers are stored in three ways:

- Automatically, after measurements are run, to the recently-used list.
- By storing them (see [page 194](#)) to the favorites list.
- By storing them (see [page 194](#)) to XML format trigger specification files.

You can recall triggers (see [page 194](#)) from the recently-used list, the favorites list, or from XML-format trigger specification files.

You can move recently-used triggers to the favorites list (see [page 195](#)).

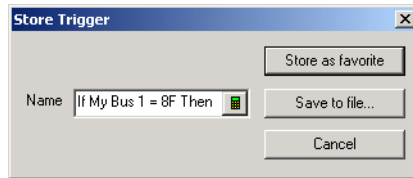
You can control the length of the recently-used and favorites list by setting the trigger history depth (see [page 195](#)).

NOTE

The current trigger setup (and the favorites list) are stored as part of the logic analyzer configuration. If you load a new configuration file, the trigger setup (and the favorites list) will be overwritten.

To store a trigger

- 1 Choose the **Setup>(Logic Analyzer Module)>Store Trigger...** command, or in the Advanced Trigger dialog, click **Store...**
- 2 In the Store Trigger dialog:



To store the trigger in the favorites list:

- a Enter the name of the trigger.
- b Click **Store as favorite**.

To store the trigger in an XML format file:

- a Click **Save to file...**
- b In the Save As dialog, enter the name of the file, and click **Save**.

- See Also
- To recall a trigger (see [page 194](#))
 - To set the trigger history depth (see [page 195](#))

To recall a trigger

- 1 Choose **Setup>(Logic Analyzer Module)>Recall Trigger...** from the menu, or in the Advanced Trigger dialog, click **Recall...**
- 2 In the Recall Trigger dialog (see [page 453](#)):
 - Select the desired trigger from the favorites or recently-used list; then, click **OK**.
 - Or, to recall a trigger from a previously saved XML format trigger specification file, click **Open...** and select the file.

- To move a recently-used trigger to the favorites list
- 1 Choose **Setup>(Logic Analyzer Module)>Recall Trigger...** from the menu.
 - 2 In the Recall Trigger dialog (see [page 453](#)), select the trigger from the recently-used list.
 - 3 Click **Store Selected Recent Trigger To Favorites List**.

See Also · To store a trigger (see [page 194](#))

To set the trigger history depth


- 1 Choose the **Edit>Options...** command.
- 2 In the Options dialog, enter the **Trigger History Depth**.
The number you enter is used for both the recently-used trigger list and the favorites list.

See Also · To store a trigger (see [page 194](#))
· To recall a trigger (see [page 194](#))

Running/Stopping Measurements

To run the analyzer in single run mode


The single run measurement captures data and fills trace memory one time. The amount of data stored during a single run is equal to the amount of trace memory allotted. For example, if trace memory is equal to 2M, the amount of data stored after one run is equal to 2M.

- From the menu bar, choose **Run/Stop>Run**, or click the  icon from the run/stop toolbar (see [page 401](#)).

See also To save captured data after each run (see [page 196](#)) below.

To run the analyzer in repetitive run mode

The run repetitive measurement captures data and fills trace memory repetitively. The amount of data stored in a repetitive run is the same as a single run. During a repetitive run, once the trace memory is full, the system clears the trace memory and begins to refill with new data. This cycle continues until the run is stopped.

- From the menu bar, choose **Run/Stop>Run Repetitive**, or click the  icon from the run/stop toolbar (see [page 401](#)).

NOTE

If you are repeatedly making measurements and looking at data some fixed time after the trigger (for example), you can change the "go to trigger on run" behavior (see [page 196](#)) so that the location being displayed doesn't change after each measurement.

See also To save captured data after each run (see [page 196](#)) and To stop repetitive runs after a certain number of acquisitions (see [page 196](#)) below.


To view analyzer run status

- From the menu bar, choose **Run/Stop>Status...**, or click **Status...** in the status bar.

The run status is displayed in the System Status tab of the Status dialog (see [page 469](#)).

To stop the analyzer

When a measurement is stopped, the amount of data gathered is equal to the amount of trace memory used up until the stop occurred. For example, if trace memory is equal to 2M and the measurement is stopped exactly half way through the run, the amount of data in trace memory would equal 1M.

- From the menu bar, choose **Run/Stop>Stop**, or click the  icon from the run/stop toolbar (see [page 401](#)).

To save captured data after each run

- From the menu bar, choose **Run/Stop>Run Properties...**
- In the Run Properties dialog (see [page 454](#)), check **Save after every acquisition** and set the additional options for saving data after each run; then, click **OK**.
- Run the analyzer in either single run mode or repetitive run mode.

To stop repetitive runs after a certain number of acquisitions

- From the menu bar, choose **Run/Stop>Run Properties...**
- In the Run Properties dialog (see [page 454](#)), check **Stop running after** and enter the number of acquisitions; then, click **OK**.
- Run the analyzer in repetitive run mode.

See Also

- To change the "Go to Trigger on Run" option (see [page 196](#))
- Run Properties Dialog (see [page 454](#))

To change the "Go to Trigger on Run" option

After a measurement is run and it completes, the default behavior of the *Keysight Logic Analyzer* application is to display the data captured around the system trigger.

If you are repeatedly making measurements and looking at data some fixed time after the trigger (for example), you can change the "go to trigger on run" behavior so that the location being displayed doesn't change after each measurement.

- 1 Choose **Edit>Options...**
- 2 In the Options dialog (see [page 447](#)), check or uncheck the **Go to Trigger on Run** box.
Option settings are saved in the registry; this means your changes will be present the next time you start the *Keysight Logic Analyzer* application.

Saving Captured Data (and Logic Analyzer Setups)


You can save logic analyzer setups and captured data to configuration files. Later, the configuration files can be opened to set up the logic analyzer and re-load the data. When saving configuration files, you can choose to save only the logic analyzer setup (that is, without the data).

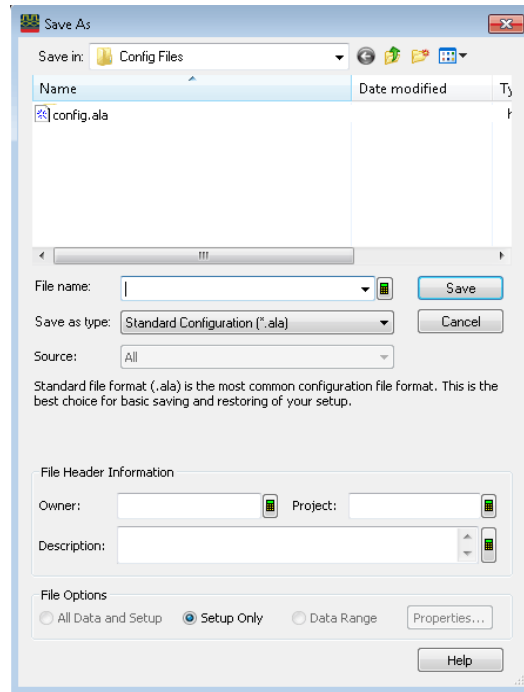
You can also save captured data to comma-separated value (CSV) files.

- To save a configuration file (see [page 198](#))
- To export data to standard CSV format files (see [page 199](#))
- To export data to module CSV format files (see [page 201](#))
- To export data to module binary (ALB) format files (see [page 203](#))

To save a configuration file

The save feature allows you to save a configuration file for later use. The first time a file is saved the logic analyzer configuration file dialog box will appear. The **Save As...** feature allows an existing configuration file to be saved under a different name.

- 1 From the menu bar, select **File>Save** or select the  icon in the standard toolbar (see [page 398](#)).



- 2 Enter the **File name**.
- 3 Select the **Save as type**.
For information on when to use the ALA (*.ala) format and when to use the XML (*.xml) format, see [ALA vs. XML, When to Use Each Format](#) (see [page 380](#)).
- 4 If you are saving as an XML format file, select the **Source**.
You can save configuration information and data from all modules or individual modules.
- 5 If desired, fill-in the **Owner**, **Project**, and **Description** fields under the file header information. These fields help identify the configuration file when it is reopened.

- 6 Select the file options:
 - **All Data and Setup** – if you wish to save captured data and instrument settings.
 - **Setup Only** – if you wish to save only the instrument settings and not the captured data.
 - **Range** – if you wish to save instrument settings and a range of captured data. Click **Properties...**; in the Range Properties dialog, specify the range.

NOTE

Configuration files that include data are much larger than files that do not contain data.

- 7 Click **Save**.

NOTE

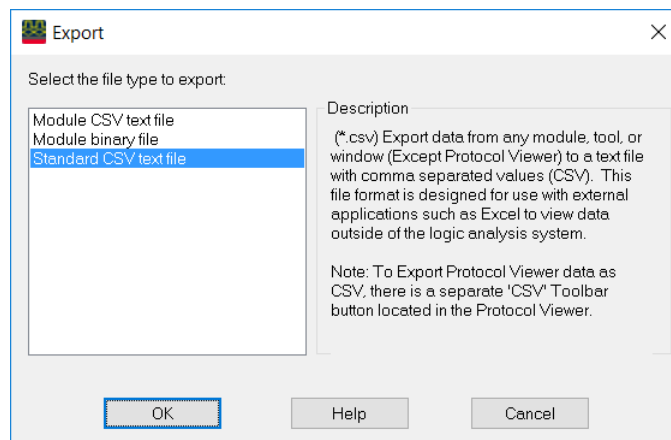
If you are using the logic analyzer without a keyboard, you can access an on-screen keyboard by selecting **Start>Programs>Accessories>Accessibility>On-Screen Keyboard**.

- See Also
- To open a configuration file (see [page 208](#))
 - Offline Analysis (see [page 217](#))
 - ALA vs. XML, When to Use Each Format (see [page 380](#))
 - ALA Format (see [page 504](#))
 - "XML Format" (in the online help)

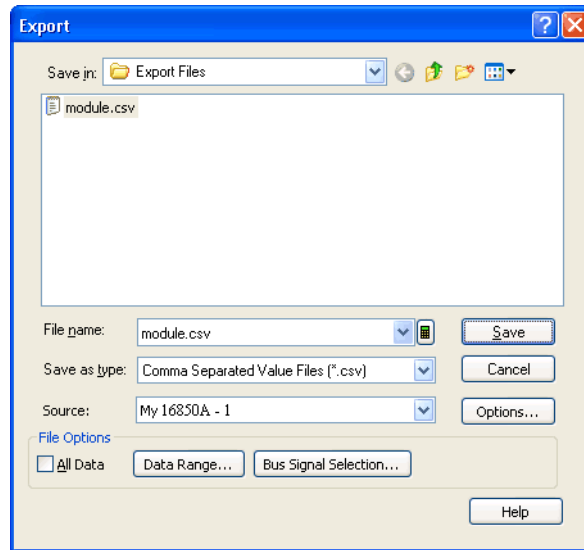
To export data to standard CSV format files

You can export captured data to standard comma-separated value (CSV) files.

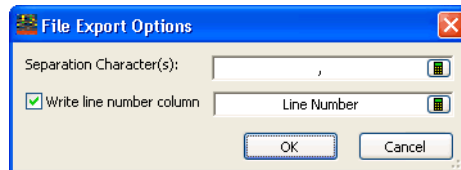
- 1 From the menu bar, select **File>Export...**
- 2 In the Export dialog, select **Standard CSV text file**; then, click **OK**.



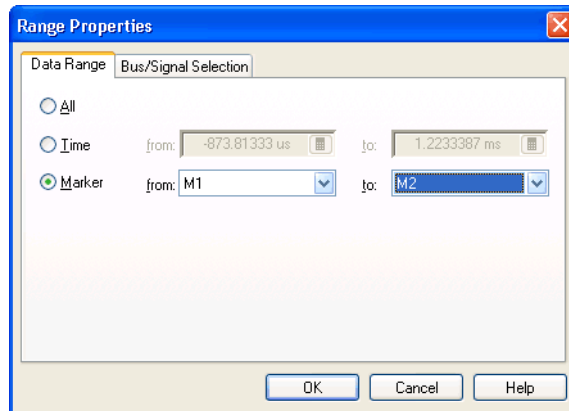
- 3 In the following Export dialog:



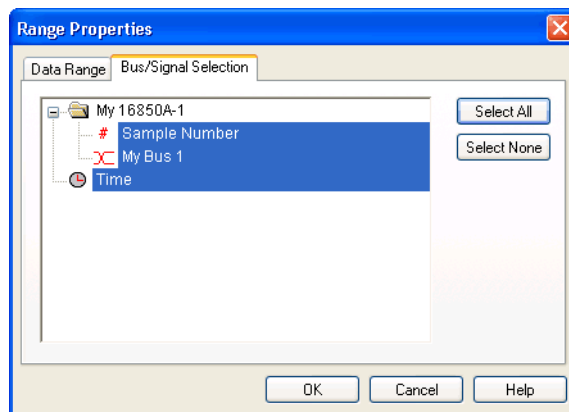
- a Enter the CSV file name.
- b Select the **Source** module, tool, or display window from which to export data.
- c If you want to use a delimiter other than a comma, or if you want to specify that line numbers be written, click **Options....** In the File Export Options dialog, make your selections; then, click **OK**.



- d If you want to export a range of data and/or selected bus/signal data, uncheck **All Data**.
- e To specify a range of data to export, click **Data Range....** In the Data Range tab of the Range Properties dialog, select the range by time or by markers; then, click **OK**.



- f To select certain bus/signal data to export, click **Bus Signal Selection...**. In the Bus/Signal Selection tab of the Range Properties dialog, select the buses/signals whose data you want to export; then, click **OK**.



NOTE

You can only choose particular buses/signals when a module or tool is selected as the **Source** of the data export. When a display window is selected as the **Source**, all buses/signals are exported. (You can, however, delete unwanted buses/signals from a display window before exporting its data.)

- 4 Click **Save**.

See Also • Standard CSV Format (see [page 504](#))

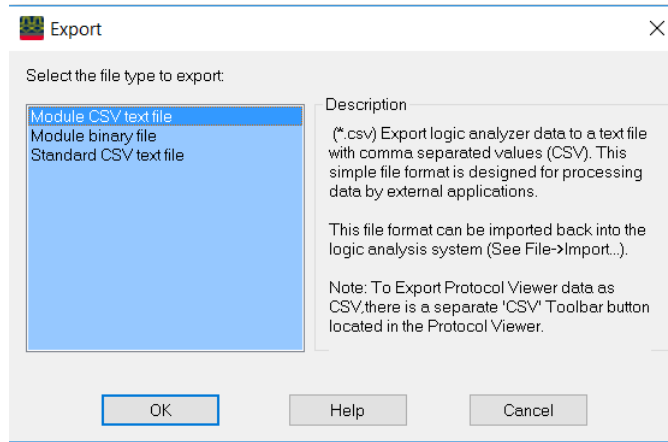
To export data to module CSV format files

You can export captured data to module comma-separated value (CSV) files. Module CSV files can be post-processed.

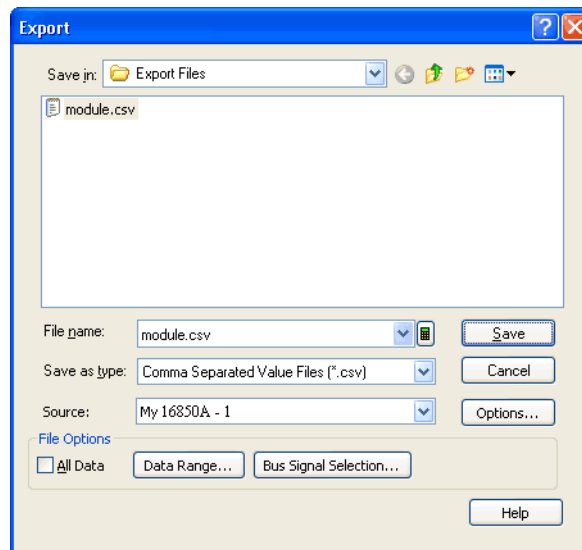
CAUTION

Do not modify module CSV files with Microsoft Excel.

- 1 From the menu bar, select **File>Export...**
- 2 In the Export dialog, select **Module CSV text file**; then, click **OK**.



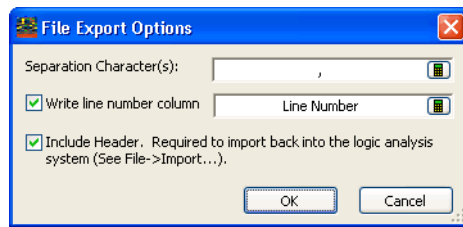
- 3 In the following Export dialog:



- a Enter the CSV file name.
- b Select the **Source** module from which to export data.

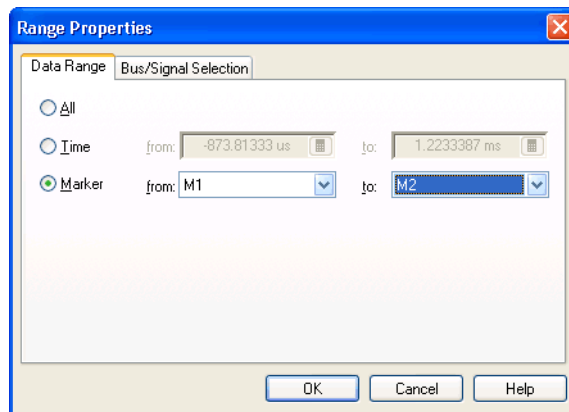
You can export data to module CSV format files from logic analyzer. You can export timing zoom data from a logic analyzer module, but it must be exported separately from the module's main data.

- c If you want to use a delimiter other than a comma, if you want to specify that line numbers be written, or if you want to exclude the header information, click **Options....** In the File Export Options dialog, make your selections; then, click **OK**.

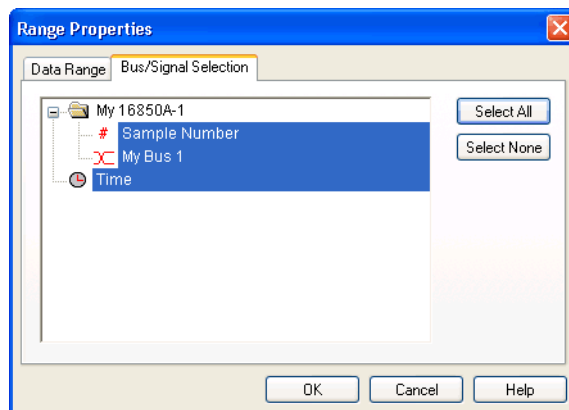


Header-less files may be easier for external tools to use.

- d If you want to export a range of data and/or selected bus/signal data, uncheck **All Data**.
- e To specify a range of data to export, click **Data Range...** In the Data Range tab of the Range Properties dialog, select the range by time or by markers; then, click **OK**.



- f To select certain bus/signal data to export, click **Bus Signal Selection...** In the Bus/Signal Selection tab of the Range Properties dialog, select the buses/signals whose data you want to export; then, click **OK**.



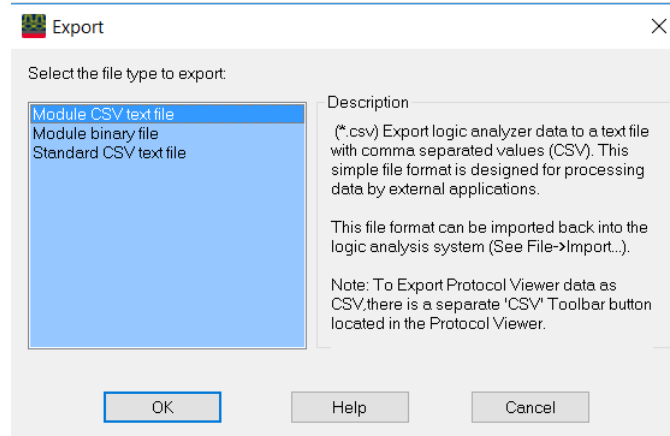
- 4 Click **Save**.

See Also · Module CSV Format (see [page 504](#))

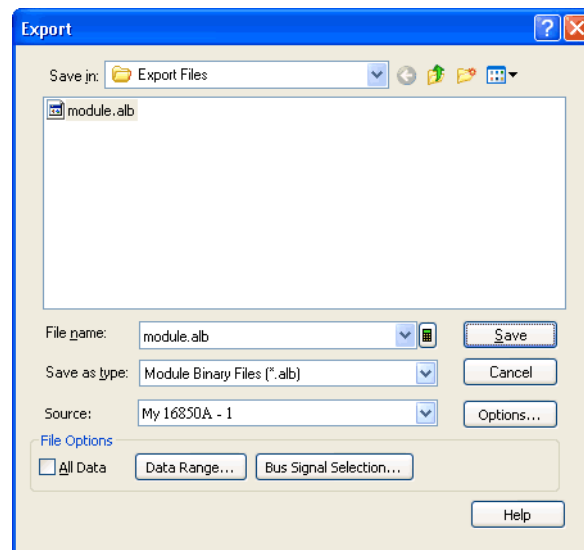
To export data to module binary (ALB) format files

You can export captured data to module binary files.

- 1 From the menu bar, select **File>Export...**
- 2 In the Export dialog, select **Module binary file**; then, click **OK**.



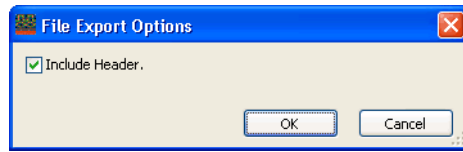
- 3 In the following Export dialog:



- a Enter the ALB file name.
- b Select the **Source** module from which to export data.

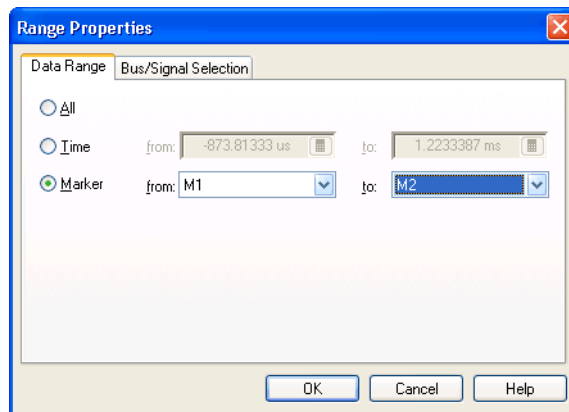
You can export data to module binary format files from logic analyzer modules only. You can export timing zoom data from a logic analyzer module, but it must be exported separately from the module's main data.

- c If you want to exclude the header information, click **Options....** In the File Export Options dialog, make your selections; then, click **OK**.

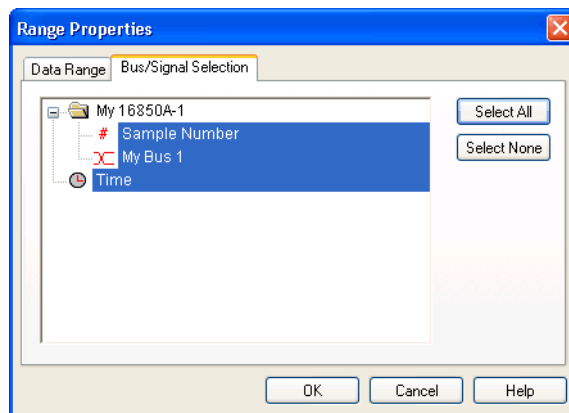


Header-less files may be easier for external tools to use.

- d If you want to export a range of data and/or selected bus/signal data, uncheck **All Data**.
- e To specify a range of data to export, click **Data Range...** In the Data Range tab of the Range Properties dialog, select the range by time or by markers; then, click **OK**.



- f To select certain bus/signal data to export, click **Bus Signal Selection...** In the Bus/Signal Selection tab of the Range Properties dialog, select the buses/signals whose data you want to export; then, click **OK**.



- 4 Click **Save**.

See Also · [Module Binary \(ALB\) Format \(see page 511\)](#)

Extending Capture Capability with COM/DCOM

With the integrated COM/DCOM, you can extend the data capture capabilities of the logic analyzer. For example:

- In the case that the logic analyzer isn't able to trigger on the event of interest, COM/DCOM can be used to do a run, analyze the captured data looking for the event and if not found, run again. For events that are relatively frequent, this allows the logic analyzer to find events that are too complex to be able to define a trigger.
- In situations where you need a lot of data to find an elusive fault, you can set up the logic analyzer to repetitively run and save data.
- You can create dynamic triggers between repetitive runs by performing a run, modifying the trigger based on analysis of the captured data, and then running again.

These are all things you could previously do with the logic analyzer's COM automation capabilities; however, it's easier and more convenient now that COM/DCOM is integrated with the *Keysight Logic Analyzer* application.

- See Also
- "Using the Advanced Customization Environment (ACE)" (in the online help)

7 Analyzing the Captured Data

Loading Saved Data and Setups /	208
Offline Analysis /	217
Analyzing Waveform Data /	220
Analyzing Listing Data /	235
Displaying Names (Symbols) for Bus/Signal Values /	243
Marking, and Measuring Between, Data Points /	245
Searching the Captured Data /	264
Comparing Captured Data to Reference Data /	274
Viewing Source Code Associated with Captured Data /	277
Analyzing Packet Data /	283
Analyzing the Same Data in Different Ways (Using the Overview Window) /	307
Setting the System Trigger and Skew Between Modules /	312
Using Display Windows /	313
Printing Captured Data /	314

Loading Saved Data and Setups

You can set up the logic analyzer and load data by opening previously saved configuration files. This lets you return to the stopping point of a previous logic analysis session, load previously saved data for *offline analysis*, or just load saved logic analyzer setups. When opening configuration files that contain data, you can choose to load only the logic analyzer setup (that is, without the data).

- To open a configuration file (see [page 208](#))
- To recall a recently used configuration file (see [page 210](#))
- To transfer module setups to/from multi-module systems (see [page 210](#))
- To transfer module setups to/from multi-module systems (see [page 210](#))
- Using Data Import Modules (see [page 211](#))
 - To create a data import module (see [page 211](#))
 - To edit data import module bus/signal definitions (see [page 213](#))
 - To view data import module file information (see [page 215](#))
 - To re-read data import module files (see [page 215](#))

See Also • [Offline Analysis \(see page 217\)](#)

To open a configuration file


You can open configuration files to return to a previous logic analysis session, to load previously saved data for *offline analysis*, or to load saved logic analyzer setups.

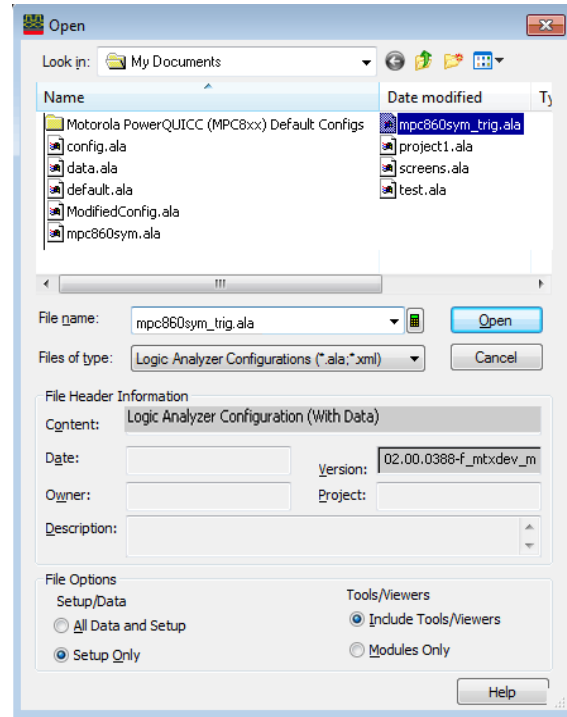
NOTE

To avoid *pod truncation* (see [page 616](#)) when opening configuration files for offline analysis, open the configuration file in a second instance of the *Keysight Logic Analyzer* application (which runs in *Offline mode*).

A quick way to start the *Keysight Logic Analyzer* application and open a configuration file is by double-clicking an ALA format configuration file in Windows Explorer. (An association for the .ala file extension was set up when the application was installed.) When you do this, however, there are no options for partial loading (setup only, modules only, etc.).

To open a configuration file from within the *Keysight Logic Analyzer* application:

- 1 From the menu bar, select **File>Open...** or select the  icon in the standard toolbar (see [page 386](#)).



The file browser portion of the Open dialog behaves the same as other standard Windows file browser dialogs (that is, you can rename files, use right-mouse operations, etc.).

- 2 Select the type of configuration file you wish to open (either *.ala or *.xml).
For information on when ALA (*.ala) and XML (*.xml) formats are used, see [ALA vs. XML, When to Use Each Format](#) (see [page 380](#)).
- 3 Select the name of the configuration file you wish to open.
The **Content**, **Date**, **Version**, **Owner**, **Project**, and **Description** fields show information about the selected configuration file. The file was created with the *Keysight Logic Analyzer* version shown in the **Version** field. The **Date** field displays the date the configuration file was created.
- 4 Select the appropriate Setup/Data option.
 - Select **All Data and Setup** to load the logic analyzer setup and data.
 - Select **Setup Only** to load only the logic analyzer setup.
- 5 Select the appropriate Tools/Viewers option.
 - Select **Include Tools/Viewers** to load tools and viewers, as well as modules, from the configuration file.
 - Select **Modules Only** to load only the module information from the configuration file.
- 6 Select **Open**.

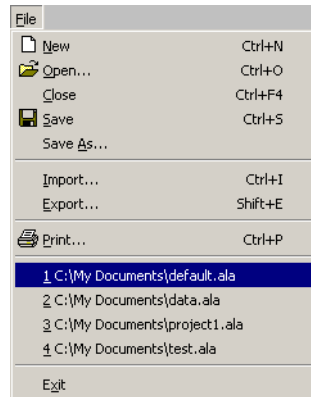
NOTE

If you are using the logic analyzer without a keyboard, you can access an on-screen keyboard by selecting **Start>Programs>Accessories>Accessibility>On-Screen Keyboard**.

- See Also
- To recall a recently used configuration file (see [page 210](#))
 - To save a configuration file (see [page 198](#))
 - To transfer module setups to/from multi-module systems (see [page 210](#))
 - Offline Analysis (see [page 217](#))
 - ALA vs. XML, When to Use Each Format (see [page 380](#))
 - ALA Format (see [page 504](#))
 - "XML Format" (in the online help)

To recall a recently used configuration file

- 1 From the menu bar, select **File**.
- 2 Select the configuration file you wish to open from the list provided.



To transfer module setups to/from multi-module systems

You can move logic analyzer setups from one logic analysis system to another by saving/opening setups to/from configuration files. When systems have multiple modules, you must map which module setup from the configuration file gets loaded into which module in the logic analysis system.

- 1 Make sure the setup (configuration file) you want to load is in the proper format:
 - If the modules are compatible (for example, in the same or similar logic analyzer families), you can use ALA format configuration files to move a setup from one module to another.
 - If the modules are not compatible, you must use XML format configuration files to move a setup from one module to another.
- 2 Open the configuration file (see To open a configuration file (see [page 208](#))).
- 3 Answer the question about clearing all modules before loading.
- 4 In the Module Mapping dialog (see [page 444](#)), select **Manually specify module mapping**; then, click **Specify Mapping....**
- 5 In the Specify Mapping dialog (see [page 468](#)), for the module you want load the setup into, select the module setup from the configuration file to load.
- 6 Click **OK** to close the Specify Mapping dialog.
- 7 Click **OK** to close the Module Mapping dialog and load the setup.

When loading module setups from XML format configuration files, an information dialog describes any parsing errors or warnings.

- See Also
- To save a configuration file (see [page 198](#))

- To open a configuration file (see [page 208](#))
- ALA vs. XML, When to Use Each Format (see [page 380](#))

Using Data Import Modules

Data import modules read data from module CSV or module binary (ALB) files and make it available to tools and display windows. Module CSV or module binary (ALB) files can be created by external tools or saved from any module using the main menu's File>Export... command.

NOTE

Data import modules are a licensed feature. You can use data import modules without a license, but the amount of data that can be imported is limited to 16 rows.

- To create a data import module (see [page 211](#))
- To edit data import module bus/signal definitions (see [page 213](#))
- To view data import module file information (see [page 215](#))
- To re-read data import module files (see [page 215](#))

Data import modules (and import file names) are saved with logic analyzer configurations (both ALA and XML format). If a configuration is saved "with data" and then opened again, the import module's data is present, and is not re-read from the import file. If a configuration is saved "without data" (setup only) and opened again, you must re-read the data import module file (see [To re-read data import module files \(see \[page 215\]\(#\)\)](#)).

See Also

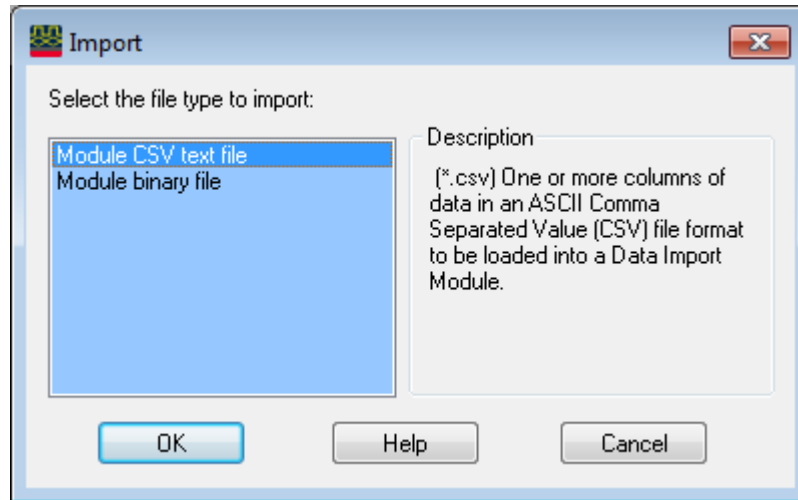
- To export data to module CSV format files (see [page 201](#))
- Module CSV Format (see [page 510](#))
- To export data to module binary (ALB) format files (see [page 203](#))
- Module Binary (ALB) Format (see [page 505](#))

To create a data import module

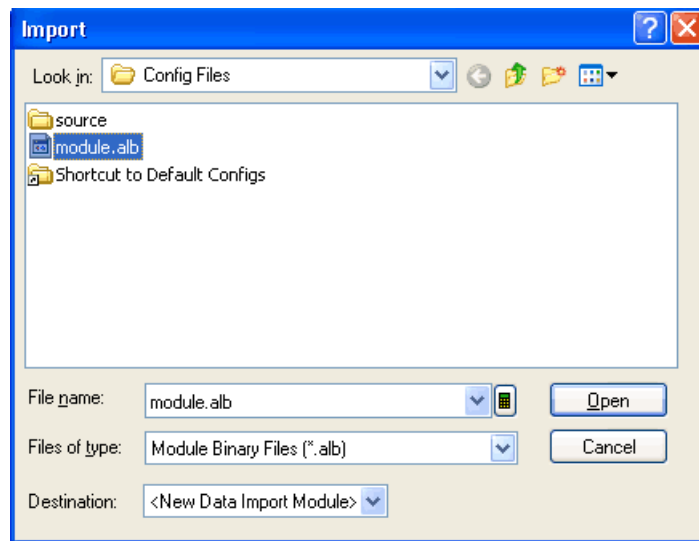
NOTE

Copy module CSV or module binary (ALB) files to the local hard disk before importing them. Performance of the Agilent Logic Analyzer application is much better when importing files from the local hard disk than when importing them from a network drive.

- 1 From the menu bar, select **File>Import...**
- 2 In the **Import** dialog, select Module CSV text file or Module binary file, and click **OK**.

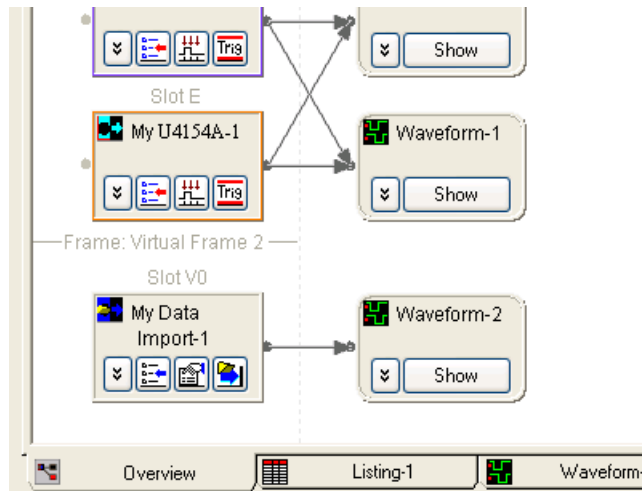


In the following **Import** dialog:



- a Select or enter the File name of the data file you wish to import.
- b Select the Destination of the data file you wish to import. You can choose an existing data import module or "<New Data Import Module>".
- c Click **Open**.


Data import modules appear in the Overview window like other logic analyzer modules. Because the data does not come from acquisition hardware in a logic analysis system frame, a virtual frame is created for data import modules. You can add tools and display windows to data import modules just like you add them to logic analyzer modules.




See Also

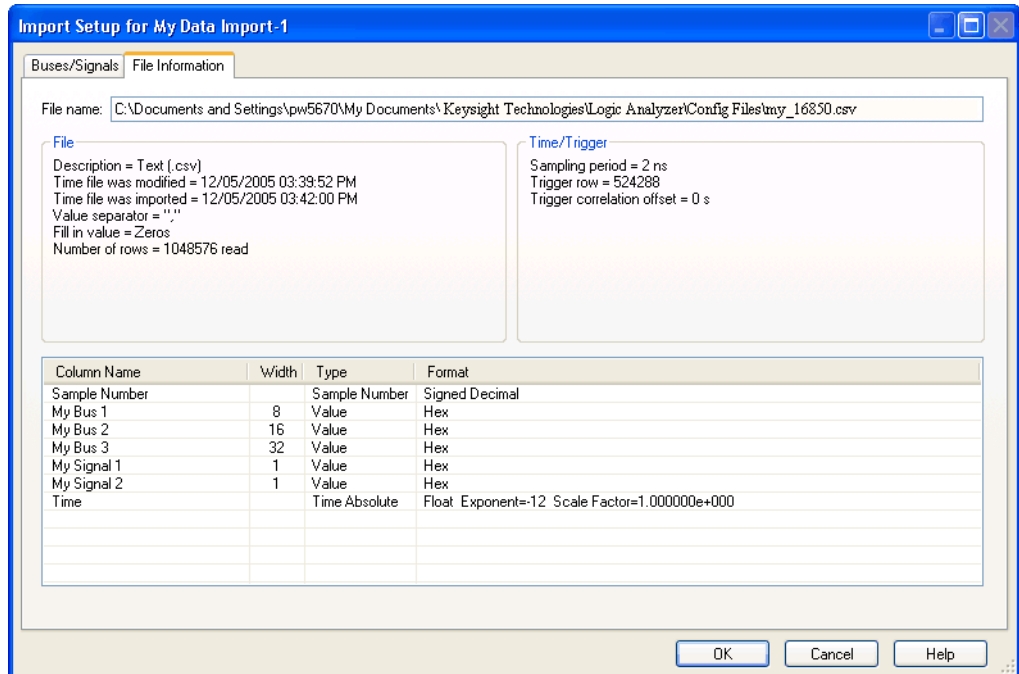
- To edit data import module bus/signal definitions (see [page 213](#))
- To view data import module file information (see [page 215](#))
- To re-read data import module files (see [page 215](#))
- Module CSV Format (see [page 505](#))
- To export data to module CSV format files (see [page 201](#))
- Module Binary (ALB) Format (see [page 505](#))
- To export data to module binary (ALB) format files (see [page 203](#))

To edit data import module bus/signal definitions

- 1 Click  in the data import toolbar, or choose **Setup>(Data Import Module)>Bus/Signals...** from the menu bar.
- 2 In the **Import Setup** dialog's **Buses/Signals** tab (see [page 442](#)):

To view data import module file information

- 1 Click  in the data import toolbar, or choose **Setup>(Data Import Module)>File Info...** from the menu bar.
- 2 In the **Import Setup** dialog's **File Information** tab (see [page 443](#)):



- The file name and other file information is displayed.
- The time column and trigger row are displayed. The trigger correlation offset is displayed.
- The module data import file's column name, width, type, and format are displayed.

See Also


- To edit data import module bus/signal definitions (see [page 213](#))
- To re-read data import module files (see [page 215](#))
- Module CSV Format (see [page 510](#))

To re-read data import module files

When you create a data import module, data is read from the imported file. You can also cause the data import file to be re-read without going through the file selection dialog again.

To re-read the data import file

Do one of the following:

- From the menu bar, choose **Run/Stop>(Data Import Module)>Read**.
- Click the  icon from the data import toolbar (see [page 400](#)) or from the data import module in the Overview window.

To view data import module read status

From the menu bar, choose **Run/Stop>Status...**, or click **Status...** in the status bar.

The read status is displayed in the System Status tab of the Status dialog (see [page 542](#)).

See Also

- To view data import module file information (see [page 215](#))

Offline Analysis

Offline analysis lets you analyze captured data while the logic analyzer's data acquisition hardware is used for making other measurements.

You can use the *Keysight Logic Analyzer* application on stand-alone personal computers, 16850-series logic analysis systems to perform offline analysis.

By placing configuration/data files on shared file systems, offline analysis can be performed from remote locations on the network.

Example: A typical scenario is to use the *Keysight Logic Analyzer* application to configure and exchange logic analyzer configuration files containing trigger setups with a team of colleagues located on-site or in remote locations.

General offline analysis considerations

- When analyzing data offline, there is no data acquisition hardware, so functions such as triggering, hardware assist, and run functions are not available.
- Multiple instances of the *Keysight Logic Analyzer* application can be displayed side-by-side on a logic analyzer or a personal computer, but their data cannot be time-correlated.
- You can install the *Keysight Logic Analyzer* on any computer meeting the minimum PC requirements (see [page 218](#)); however, licensed tools require a license for each computer they run on.

For more specific information about offline analysis, see:

- [Offline Analysis on Logic Analyzers \(see page 217\)](#)
- [Offline Analysis on Personal Computers \(see page 218\)](#)

To return to online analysis, see:

- [Connecting to a Logic Analysis System \(see page 59\)](#)

See Also • [Offline File Formats \(see page 218\)](#)

Offline Analysis on Logic Analyzers

You can perform offline analysis with 16850-series logic analyzers, in general, by running a second instance of the *Keysight Logic and Protocol Analyzer* application and loading previously saved data into that instance. With two instances of the application running, one in online (either *Local* or *Remote*) mode and one in *Offline* mode, you can continue making measurements in one instance while you perform offline analysis in the other.

Keep these things in mind when performing offline analysis with a logic analyzer or logic analysis system:

- You can start multiple instances of the *Keysight Logic Analyzer* application.
If logic analyzer acquisition hardware is present, the first instance opens in online (either *Local* or *Remote*) mode. If acquisition hardware is not present, the first instance opens in *Offline* mode.
- Logic analyzer run functions do not work in *Offline* mode.
- In the *Offline* mode, you can continue to create triggers and save them in configuration files that can be opened by other instances of the *Keysight Logic Analyzer* application.
- If you open a logic analyzer configuration (.ala) file for offline analysis in an online (either *Local* or *Remote*) instance of the *Keysight Logic Analyzer* application, run functions will overwrite the data that has been loaded.

NOTE

To open, copy, or save files directly from shared disk drives, make sure to configure all LAN connections (see [page 315](#)) to enable file sharing.

- See Also
- To open a configuration file (see [page 208](#))
 - To save a configuration file (see [page 198](#))

Offline Analysis on Personal Computers

A personal computer (PC) with the *Keysight Logic Analyzer* application installed can perform offline analysis on configuration files (.ala format) from any 16850-series logic analysis system.

When using the *Keysight Logic Analyzer* application by itself on a PC for offline analysis:

- The logic analyzer run functions are not available (because there is no acquisition hardware).
- You can save logic analyzer setups (including trigger sequences) to .ala format configuration files, and you can pass these files between personal computers (running the *Keysight Logic Analyzer* application).
- You can have more than one instance of the *Keysight Logic Analyzer* application running.
- Licensed tools require a license for each computer they run on.

NOTE

To open, copy, or save files directly from shared disk drives, make sure to configure all LAN connections (see [page 315](#)) to enable file sharing.

- See Also
- To open a configuration file (see [page 208](#))
 - To save a configuration file (see [page 198](#))
 - Minimum PC Requirements (see [page 218](#))

Minimum PC Requirements

Minimum PC requirements for offline analysis:

- Processor and Memory: 1 GHz processor, 512 M RAM.
- Operating System:
 - Windows Vista
 - Windows 7
 - Windows 8
 - Windows 8.1
 - Windows 10
 - Windows Server 2008
 - Windows Server 2008 R2
 - Windows Server 2012
 - Windows Server 2012 R2

Offline File Formats

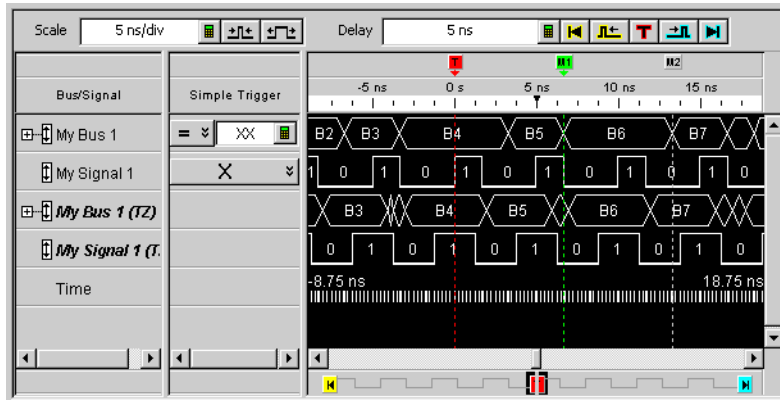
- .ala Format** You can open logic analyzer configuration files (.ala (see [page 504](#)) extension) for offline analysis. These files are saved by the *Keysight Logic Analyzer* application.

The logic analyzer configuration (.ala) file format is an internal format used by the *Keysight Logic Analyzer* application for saving and re-opening setups and data. ALA format configuration files contain everything that is needed to restore a session (in other words, the information necessary to reconstruct the display appearance, instrument settings, and optionally, captured data).

- .xml Format** Generic configuration files (".xml" (in the online help) extension) can be used when setting up the logic analysis system configuration. XML format configuration files are saved by the *Keysight Logic Analyzer* application.
- The generic configuration (.xml) file format is an eXtensible Markup Language format that can be edited (using an ASCII text editor) and post-processed by scripts (or other tools) and re-opened by the *Keysight Logic Analyzer* application. These files contain buses/signals, channels assigned to buses/signals, and user-defined symbols.
- CSV Format** You can export captured data to CSV (Comma-Separated Values) Format (see [page 504](#)) files (.csv extension) for offline analysis in other applications like Excel. These files contain buses/signals and data.

Analyzing Waveform Data

The Waveform window displays captured data as a digital waveform. You can configure the window to display selected buses and signals with time or pattern markers in the data. You can also set up bus pattern triggers and signal trigger options.



The Waveform window is accessed through the menu bar's **Window>Waveform** command. If you have Tabbed Windows (see [page 313](#)) turned on, you can also select a tab at the bottom of the window.

- To change the display scale (time/division) (see [page 221](#))
- To go to different locations in the captured data (see [page 222](#))
- To re-arrange waveforms (see [page 223](#))
- To overlay waveforms (see [page 223](#))
- To find a bus/signal row (see [page 224](#))
- To view bus data as a chart (see [page 224](#))
- To show/hide parts of the waveform display (see [page 226](#))
- To insert or delete buses/signals (see [page 227](#))
- To group signals into a bus (see [page 227](#))
- To expand/collapse buses (see [page 227](#))
- To insert separator rows (see [page 227](#))
- Changing Waveform Window Properties (see [page 227](#))
 - To change the waveform background color (see [page 228](#))
 - To change the overlaid waveform color (see [page 228](#))
 - To change the filtered data color (see [page 229](#))
 - To change the timing zoom background color (see [page 229](#))
 - To change the waveform font size (see [page 229](#))
 - To change the Fast Zoom In option (see [page 230](#))
 - To lock scrolling with other display windows (see [page 229](#))
 - To change the waveform tool tip display (see [page 230](#))
- Changing Bus/Signal Row Properties (see [page 230](#))
 - To change a waveform's color (see [page 231](#))
 - To change a waveform's height (see [page 232](#))
 - To change a bus/signal's number base (see [page 232](#))

- To show/hide a bus/signal's numeric data values (see [page 232](#))
- Changing Analog Signal Row Properties (see [page 233](#))
 - To change the analog properties (see [page 234](#))

See Also

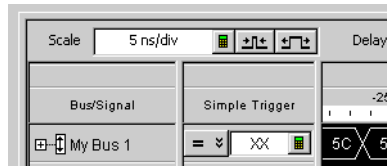
- Defining Buses and Signals (see [page 83](#))
- Setting Up Quick (Draw Box) Triggers (see [page 143](#))
- Specifying Simple Triggers (see [page 146](#))
- Marking, and Measuring Between, Data Points (see [page 245](#))
- Setting Up Symbols (see [page 132](#))
- Displaying Names (Symbols) for Bus/Signal Values (see [page 243](#))
- Searching the Captured Data (see [page 264](#))

To change the display scale (time/division)

The Waveform window displays data similarly to an oscilloscope, that is, waveforms on a horizontal time axis. Therefore, to zoom in or out on a waveform, you simply change the Scale (time/division) of the time axis that the waveform is viewed with.

To change the scale by clicking the zoom out/in buttons

- 1 Click the zoom out/in buttons to raise/lower the time/division scale.

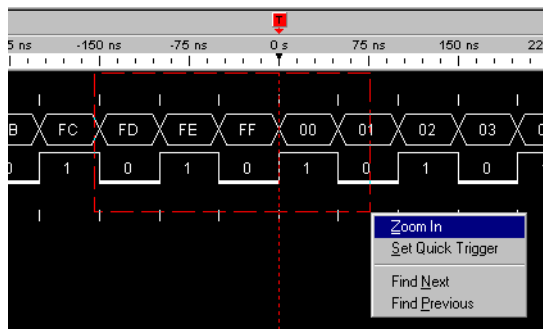


The scale ranges from 1 ps/div to 1 ks/div.

You can also change the time/division by entering a numeric value in the **Scale** field.

To zoom in by drawing a rectangle in the data

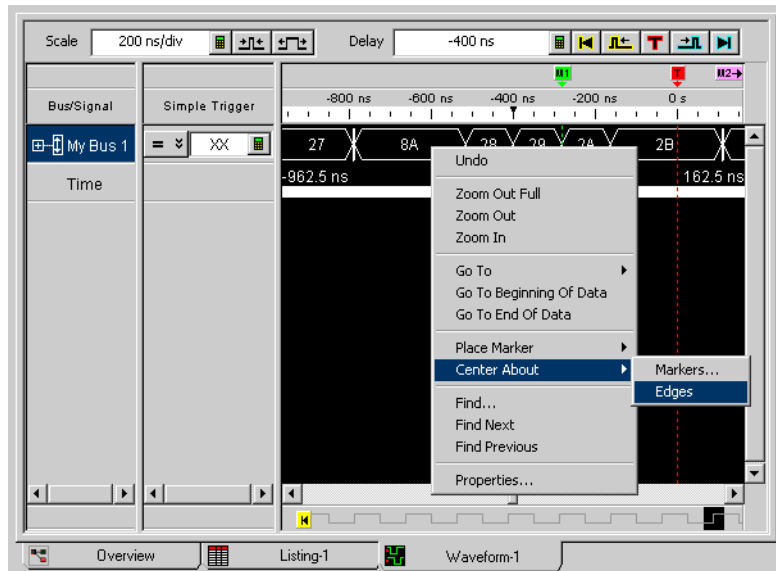
- 1 Point the mouse to the upper-left corner of the desired view area; then, click and hold while moving the mouse to the lower-right corner; then, release the mouse button.
- 2 If the Fast Zoom In (see [page 230](#)) option is not selected, choose **Zoom In** from the popup menu.



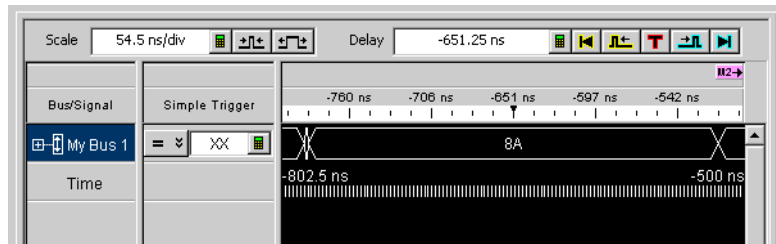
The new display scale is adjusted to the width of the box drawn.

To center about edges

- 1 Position the mouse cursor over a waveform, between the edges you want to center the display about.
- 2 Right-click and choose **Center About>Edges**.



The new display scale is adjusted to the width of the waveform edges.



To go to different locations in the captured data







In the Waveform display window, you can go to different locations in the captured data by using the horizontal scroll bars, by using the Delay field and buttons, or by choosing Go To commands from popup menus.

To go to different locations by changing the delay

- 1 Click one of the buttons in the Delay field or enter a delay value.

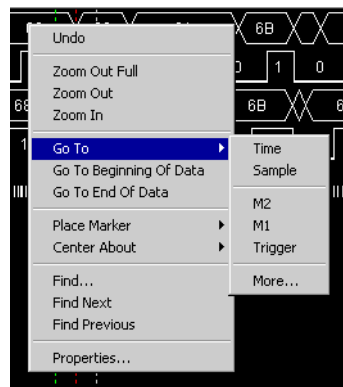


The delay adjusts the display window relative to the waveform data. The display window's relative position in time is dependent on the trigger point, and the beginning and end of data. Use the following delay controls to position the display window over the desired data.

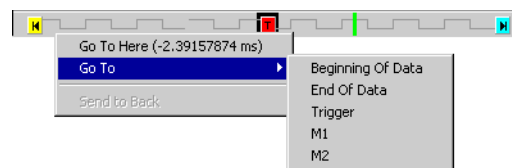
Menu	Description
	Use the keypad to enter a numeric value. If the value you enter is greater than or less than the time of the data range, the window will be moved to the beginning or end limit.
	Moves the window over the beginning of data.
	Scrolls the window towards the beginning of data.
	Moves the window over the trigger point.
	Scrolls the window towards the end of data.
	Moves the window over the end of data.

To go to different locations using popup menus

- 1 Right-click in the waveform display area, and choose one of the **Go To** commands.



Or, click in the marker overview bar, and choose one of the **Go To** commands.



You can choose **Beginning Of Data**, **End Of Data**, **Trigger**, a marker, a **Time**, or a **Sample**.

To re-arrange waveforms

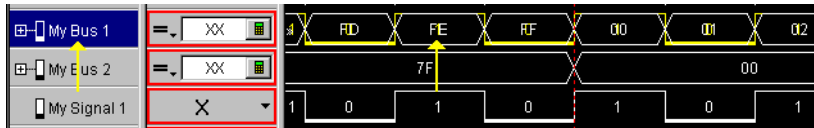
- 1 Position the mouse pointer over the bus/signal name associated with the waveform you want to move.
- 2 Click and hold the mouse button.
- 3 Drag-and-drop the bus/signal to its new position.
The name is placed above the red position indicator that appears.

See Also

- To overlay waveforms (see [page 223](#))
- To expand/collapse buses (see [page 227](#))

To overlay waveforms

Use the Overlay feature to place multiple bus or signals into one row of displayed data. When multiple signals are overlaid, you can see the relationships visually between all signals. The overlaid bus or signal is drawn first, then the main bus/signal is drawn last as to overwrite the overlaid bus/signals for clarity.



- 1 Right-click on the bus or signal you want to overlay another bus or signal onto, then select **Overlay....**
- 2 From the Overlay selection dialog that appears, select the bus or signal you want to overlay onto the highlighted bus or signal.
You can overlay analog signals (from an " external oscilloscope module" (in the online help)) with digital signals or with other analog signals.

NOTE

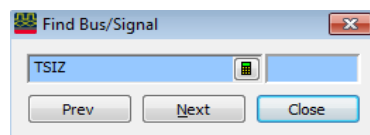
The scaling for an original analog signal is used for *all* signals overlaid onto its row; therefore, if user-defined scaling and offset values are used, it is possible that overlaid signals may not be visible. When automatic scaling is used, it will take into account the minimum and maximum voltages of *all* overlaid signals, and all signals will be visible.

- 3 If you want to change the color of the overlaid bus or digital signal, see To change the overlaid waveform color (see [page 228](#)).
When analog signals are overlaid onto other signals, the overlay signal color comes from the external oscilloscope module's setup options (which you can access by right-clicking the analog signal name, choosing **Assign Channels...**, and selecting the " Options tab" (in the online help)) instead of the Waveform window's overlay color setting.

To find a bus/signal row

When there are many bus/signal rows in the Waveform display window, you can search for a particular bus/signal row instead of scrolling through all the rows.

- 1 In the Waveform display window, right-click in the Bus/Signal column, and choose **Find Bus/Signal....**
- 2 In the Find Bus/Signal dialog, enter the name (or part of the name) of the bus/signal you wish to find.

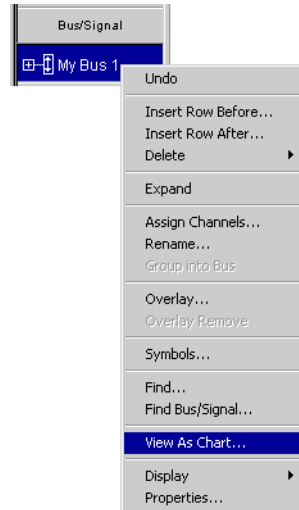


- 3 Then, click:
 - **Prev** – to search for the string backward in the bus/signal rows.
 - **Next** – to search for the string forward in the bus/signal rows.
 - **Close** – to close the Find Bus/Signal dialog.

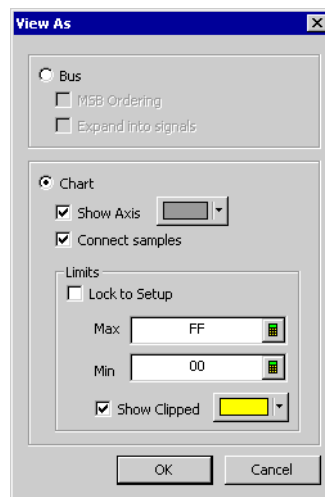
To view bus data as a chart

You can view bus data values as a chart instead of the conventional bus shape.

- 1 In the Waveform display window, right-click on a bus name, and choose **View As Chart....**
Or, in the Waveform Properties dialog's Row Properties tab, for the **Bus** property, click **View As....**



- 2 In the View As dialog, set the following options:



- **Show Axis** – causes a small axis, which represents the center of the range of values being displayed, to be drawn in the center of the waveform. When checked, you can also change the axis color.
- **Connect Samples** – causes lines to be drawn between samples.
- **Lock to Setup** – sets the range limits based on the width of the bus. For example, an 8-bit bus is set to a range of 0–255.
- **Max/Min** – sets the range limits of the displayed axis.
- **Show Clipped** – enables out-of-range data values to be displayed in a user-defined color.

NOTE

Because there is no hardware to accelerate chart drawing, Waveform windows that have charts draw slowly. You may want to place buses viewed as charts in a separate Waveform window.

To view bus data as a bus

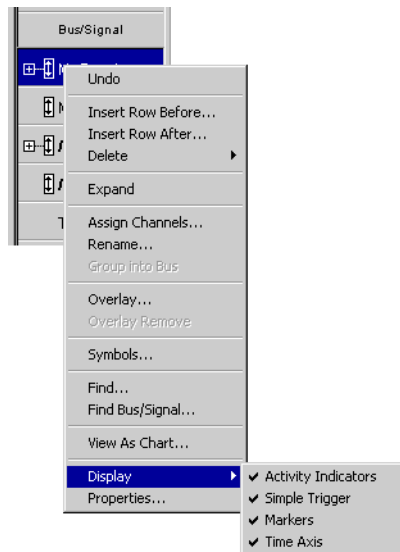
You can return the bus waveform appearance to a conventional bus shape.

- 1 In the Waveform display window, right-click on a bus name, and choose **View As Bus...**
Or, in the Waveform Properties dialog's Row Properties tab, for the **Bus** property, click **View As...**
- 2 In the View As dialog, check or uncheck the following options as desired:
 - **MSB Ordering** – the ordering of the signals in the bus are changed from least significant bit first to most significant bit first.
 - **Expand into signals** – expands the bus into individual signals (as if you selected the Expand (+) field to the left of the bus name).

See Also • Changing Bus/Signal Row Properties (see [page 230](#))

To show/hide parts of the waveform display

- 1 Right-click in the Bus/Signal column of the Waveform display, and choose **Display**>.



Then, check or uncheck one of the following to show or hide that part of the Waveform display window:

- **Activity Indicators** – either a low bar (low level), high bar (high level), or a transition arrow (transitioning signal) displayed to the left of bus/signal names.
- **Simple Trigger** – the Simple Trigger column.
- **Markers** – the markers display bar (see [page 407](#)).
- **Time Axis** – the time axis (and column headings).

You can also make these selections in the **Display Options** area of the Waveform Properties dialog's Window Properties tab.

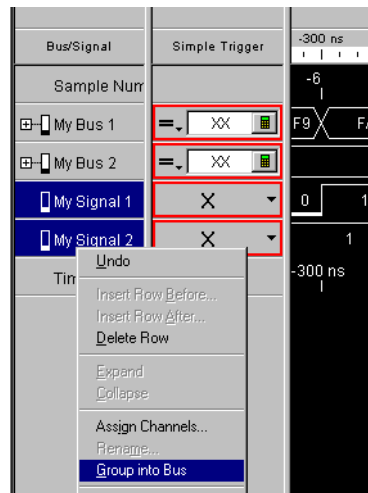
See Also • Changing Waveform Window Properties (see [page 227](#))

To insert or delete buses/signals

- | | |
|----------------------------------|--|
| To insert buses/signals | <ol style="list-style-type: none"> 1 In the Waveform display window, right-click in the Bus/Signal column; then, choose Insert Row. 2 In the Insert dialog, select the buses/signals you want to insert; then, click OK. |
| To delete selected buses/signals | <ol style="list-style-type: none"> 1 In the Bus/Signal column, highlight the buses/signals you want to delete (by clicking, Shift-clicking, or Ctrl-clicking the bus/signal names). 2 Right-click in the Bus/Signal column; then, choose Delete>Row. |
| See Also | <ul style="list-style-type: none"> • Defining Buses and Signals (see page 83) |

To group signals into a bus

- 1 While holding the shift key down, click on all desired signals.
- 2 With the mouse pointer over any one of the highlighted signals, right-click and select **Group into Bus**.



To expand/collapse buses

In the Waveform display window's Bus/Signal column:

- Click the "+" or "-" symbol associated with a bus.
- Right-click the bus, and choose **Expand** or **Collapse**.

See Also • Defining Buses and Signals (see [page 83](#))

To insert separator rows

To add distance between waveforms, you can add separator rows to the Waveform display window.

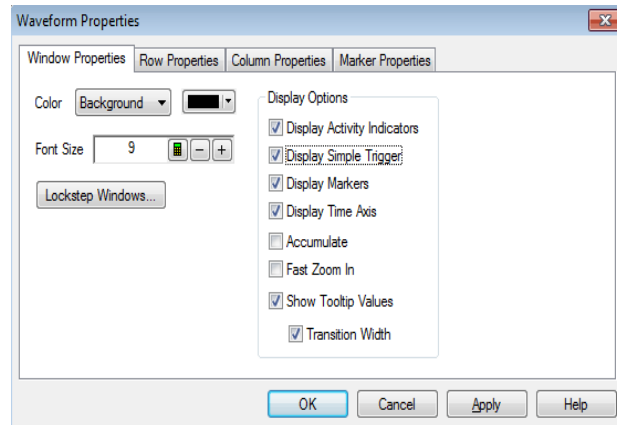
- 1 In the Waveform display window, right-click in the Bus/Signal column; then, choose **Insert Separator**.

Separator rows can be sized, colored, re-arranged, and deleted just like bus/signal waveform rows.

Changing Waveform Window Properties

You can change properties that affect the entire Waveform display window.

- 1 Right-click in a blank area of the waveform display, and choose **Properties...**
Or, with no bus/signal names selected, choose **Edit>Window Properties...** from the main menu.
- 2 In the Waveform Properties dialog's Window Properties tab:



You can:

- Change the waveform background color (see [page 228](#))
 - Change the overlaid waveform color (see [page 228](#))
 - Change the filtered data color (see [page 229](#))
 - Change the timing zoom background color (see [page 229](#))
 - Change the waveform font size (see [page 229](#))
 - Lock scrolling with other display windows (see [page 229](#))
 - Show/hide parts of the waveform display (see [page 226](#))
 - Change the accumulate waveforms option (see [page 229](#))
 - Change the Fast Zoom In option (see [page 230](#))
 - Change the waveform tool tip display (see [page 230](#))
- 3 Click **OK** to apply the changes and close the Waveform Properties dialog.

See Also • [Changing Bus/Signal Row Properties \(see page 230\)](#)

To change the waveform background color

- 1 In the Waveform Properties dialog's Window Properties tab, select the **Background** color, click the selection button, and select the desired background color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

To change the overlaid waveform color

When buses/signals are overlaid (see [To overlay waveforms \(see page 223\)](#)), the overlay property specifies the color used for the overlaid waveforms.

- 1 In the Waveform Properties dialog's Window Properties tab, select the **Overlay** color, click the selection button, and select the desired overlaid waveform color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

To change the filtered data color

When a filter tool is used to hide data from the Waveform display window, cross-hatching appears at locations where data is hidden; the filter property specifies the color used for the cross-hatched areas.

- 1 In the Waveform Properties dialog's Window Properties tab, select the **Filter** color, click the selection button, and select the desired filter color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

To change the timing zoom background color

You can give waveforms from the timing zoom feature a different background color than other waveforms.

- 1 In the Waveform Properties dialog's Window Properties tab, select the **TimingZoom** color, click the selection button, and select the desired timing zoom background color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

To change the waveform font size

The font size property adjusts the data display, bus/signal, and simple trigger text size.

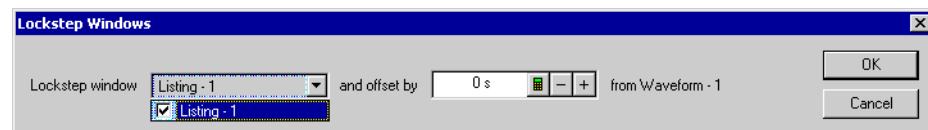
- 1 In the Waveform Properties dialog's Window Properties tab, enter the desired **Font Size**.
Fonts can range from size 6 through 72 points.
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

As the font size is changed, the row height may be automatically increased to fit the new text size.

To lock scrolling with other display windows

You can lock display windows (for example, Waveform, Listing, Compare, etc.) so that when one window is scrolled, others are scrolled as well, such that the same time is centered in each display.

- 1 In the Waveform Properties dialog's Window Properties tab, click **Lockstep Windows...**
- 2 In the Lockstep Windows dialog, select the display windows whose scrolling should be locked with this window and specify any offset from this window.

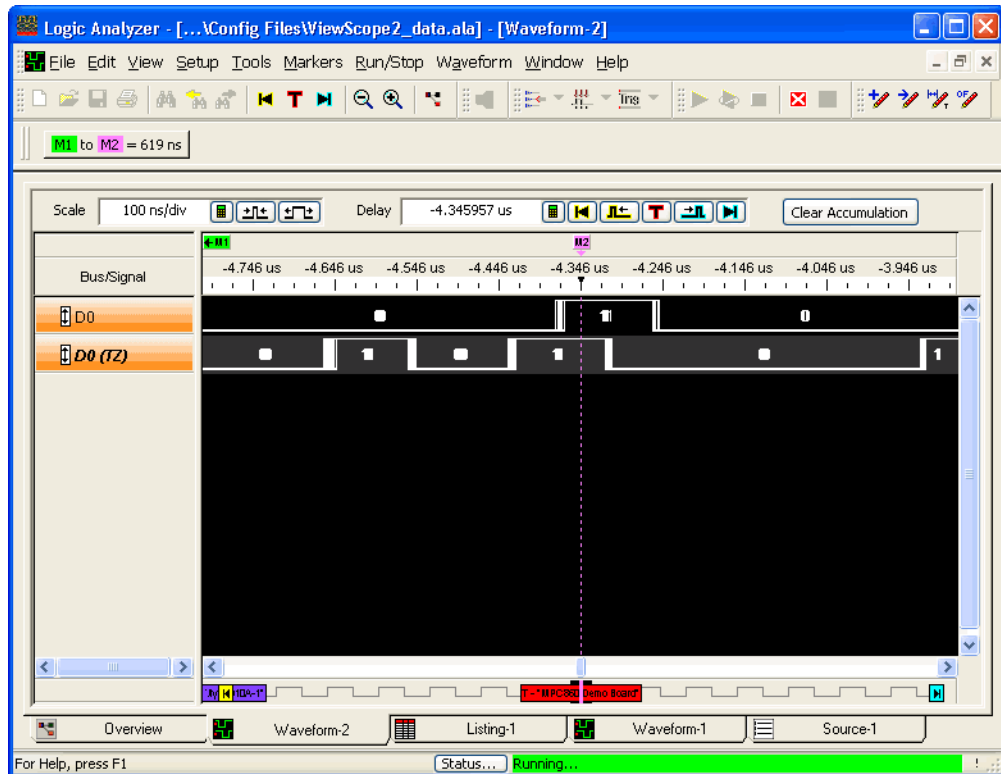


- 3 Click **OK** to close the Lockstep Windows dialog.
- 4 Click **OK** to apply the changes and close the Waveform Properties dialog.

To accumulate waveforms

- 1 In the Waveform Properties dialog's Window Properties tab's Display Options area, check or uncheck **Accumulate** to specify whether waveforms are accumulated on the display.
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

When the accumulate option is enabled and you run repetitively, waveforms overlay previously captured waveforms.



To clear accumulated waveforms, click **Clear Accumulation** at the top of the Waveform window.

To change the Fast Zoom In option

When the Fast Zoom In option is selected, you can drag the mouse cursor over the area you want to zoom in on, and when you release the mouse button, the zoom happens immediately, without having to select **Zoom In** from a popup menu.

- 1 In the Waveform Properties dialog's Window Properties tab, select or deselect the **Fast Zoom In** option.
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

- See Also
- To change the display scale (time/division) (see [page 221](#))
 - To set a Quick Trigger in the Waveform window (see [page 143](#))
 - To quickly find bus signal patterns (see [page 264](#))

To change the waveform tool tip display

A *tool tip* (that is, a small box with text) can appear when the mouse pointer is over a waveform and held motionless for a second.

- 1 In the Waveform Properties dialog's Window Properties tab's Display Options area, check or uncheck **Show Tooltip Values** to specify whether bus/signal values are shown as tool tips.
If **Show Tooltip Values** is checked, check or uncheck **Transition Width** to specify whether transition (or pulse) width values are included in the tool tips.
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

Changing Bus/Signal Row Properties

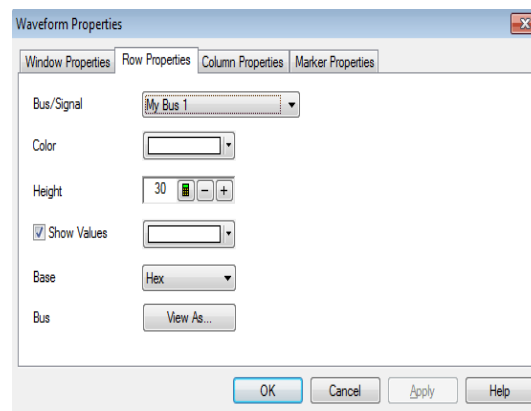
In the Waveform display window, you can change the color or size of a waveform, and you can choose whether numeric values are displayed with the waveform.

NOTE

Property changes to a bus affect all signals within the bus. For example, if you change the color of a bus and then expand the bus, you will see that the color is changed for all signals in the bus.

To change the properties of a waveform in the Waveform window:

- 1 Right-click on a bus/signal name or on a waveform, and choose **Properties...**
Or, highlight the buses/signals whose properties you want to change (by clicking, Shift-clicking, or Ctrl-clicking the bus/signal names), and choose **Edit>Window Properties...** from the main menu.
- 2 In the Waveform Properties dialog's Row Properties tab:



You can:

- Select the **Bus/Signal** to which the property changes apply. You can select:
 - Any bus/signal name that has been assigned (see Defining Buses and Signals (see [page 83](#))).
 - **<all>** buses/signals.
 - **<selected>** buses/signals if more than one is highlighted in the Bus/Signal column.
 - Change a waveform's color (see [page 231](#))
 - Change a waveform's height (see [page 232](#))
 - Change a bus/signal's number base (see [page 232](#))
 - Show/hide a bus/signal's numeric data values (see [page 232](#))
 - View bus data as a chart or a bus (see [page 224](#))
- 3 Click **OK** to apply the changes and close the Waveform Properties dialog.

See Also • [Changing Waveform Window Properties \(see page 227\)](#)

To change a waveform's color

- 1 In the Waveform Properties dialog's Row Properties tab, click the **Color** selection button and select the desired waveform color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

See Also · Changing Analog Signal Row Properties (see [page 233](#))

To change a waveform's height

- 1 In the Bus/Signal column of the Waveform display window, position the mouse pointer over a row separator line; when the cursor changes to a resizing cursor, drag the row border.

Or:

- 1 In the Waveform Properties dialog's Row Properties tab, enter the **Height** value in pixels. The minimum row height is set by the font size (see To change the waveform font size (see [page 229](#))). The maximum height is 1000 pixels.
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

See Also · Changing Analog Signal Row Properties (see [page 233](#))

To change a bus/signal's number base

When a bus/signal's numeric data values are displayed (see To show/hide a bus/signal's numeric data values (see [page 232](#))), the base property specifies the number base to use.

- 1 In the Waveform Properties dialog's Row Properties tab, select the desired number **Base** from:
 - **Binary**
 - **Hex**
 - **Octal**
 - **Decimal**
 - **Signed Decimal** (two's complement)
 - **Ascii**
 - **Symbol** (see Displaying Names (Symbols) for Bus/Signal Values (see [page 243](#)))
- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

NOTE

If an analog signal from an external oscilloscope module (see External Oscilloscope Time Correlation and Data Display help) column has been selected instead of a data column, then:

- **Amperage** is used as the Base for the signal's data if you have enabled the Current Probe checkbox for this signal in the External Oscilloscope Setup dialog box.
- **Voltage** is used as the Base for the signal's data if you have NOT enabled the Current Probe checkbox for this signal in the External Oscilloscope Setup dialog box.

To show/hide a bus/signal's numeric data values

You can display (and specify the color of) numeric data values with a waveform.

NOTE

If the waveform time scale is small, "..." may appear in the data value to indicate that more text will be displayed if you expand the scale.

- 1 In the Waveform Properties dialog's Row Properties tab, check or uncheck **Show Values** to show or hide numeric data values with the waveform.

If **Show Values** is checked, click the color selection button and select the desired data value color from the palette.

If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.

- 2 Click **OK** to apply the changes and close the Waveform Properties dialog.

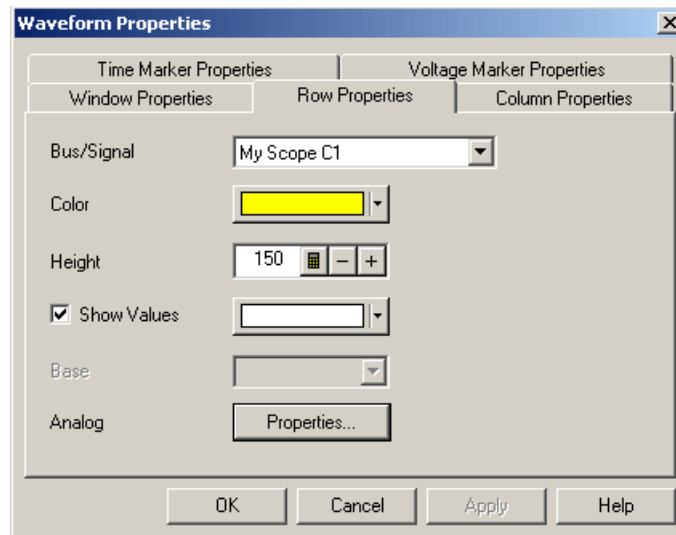
See Also • Changing Analog Signal Row Properties (see [page 233](#))

Changing Analog Signal Row Properties

In the Waveform display window, you can change the color or size of an analog signal waveform, and you can choose whether voltage and volts/division values are displayed with the waveform. Analog signals come from an external oscilloscope module (see " External Oscilloscope Time Correlation and Data Display" (in the online help)).

To change the properties of an analog signal waveform in the Waveform window:

- 1 Right-click on an analog signal name or on a waveform, and choose **Properties...**
Or, highlight the analog signals whose properties you want to change (by clicking, Shift-clicking, or Ctrl-clicking the analog signal names), and choose **Edit > Window Properties...** from the main menu.
- 2 In the Waveform Properties dialog's Row Properties tab:



You can:

- Select the **Bus/Signal** to which the property changes apply. You can select:
 - Any bus/signal name that has been assigned (see Defining Buses and Signals (see [page 83](#))).
 - **<all>** buses/signals.
 - **<selected>** buses/signals if more than one is highlighted in the Bus/Signal column.
- Change a waveform's color (see [page 231](#))
- Change a waveform's height (see [page 232](#))
- Show/hide a signal's numeric data values (see [page 232](#))
- Change the analog properties (see [page 234](#))

NOTE

The Base field is disabled because for analog signals:

- Amperage is used as the Base for the signal's data if you have enabled the Current Probe checkbox for this signal in the External Oscilloscope Setup dialog box.
- Voltage is used as the Base for the signal's data if you have NOT enabled the Current Probe checkbox for this signal in the External Oscilloscope Setup dialog box.

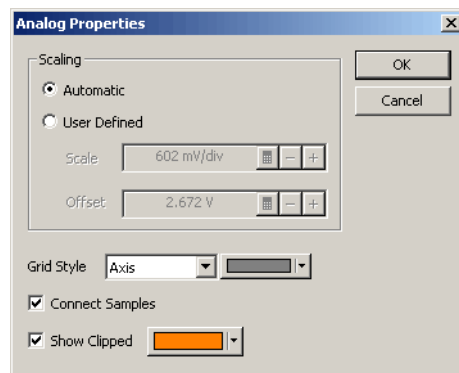
- 3 Click **OK** to apply the changes and close the Waveform Properties dialog.

See Also • To change Y-Axis Marker properties (see [page 262](#))

To change the analog properties

You can change the properties of an analog signal in the Waveform display window.

- 1 In the Waveform Properties dialog's Row Properties tab, for the **Analog** property, click **Properties...**
- 2 In the Analog Properties dialog, set the following options:



- **Scaling** – you can select **Automatic** (to have the voltage/Amperage scale and offset automatically set) or **User Defined** (to be able to set your own voltage/amperage scale and offset values). Be careful not to enter scale and offset values that will move the waveform out of the display area. In this section, Amperage is used as the unit if you have enabled the Current Probe checkbox for this signal in the External Oscilloscope Setup dialog box. Voltage is used as the unit if you have NOT enabled the Current Probe checkbox for this signal in the External Oscilloscope Setup dialog box.
- **Grid Style/Color** – you can select **None**, **Axis** (to have axis lines drawn through the center of the waveform display area), or **Grid** (to have grid lines drawn for voltage and time divisions). If you choose axis or grid, you can specify its color.
- **Connect Samples** – specifies whether lines are drawn between waveform data sample points.
- **Show Clipped** – enables out-of-range data values to be displayed in a user-defined color.

Analyzing Listing Data

The Listing window displays your captured data as a state listing. You configure the window to display selected buses and signals in columns. Within the listed data, you can insert time or pattern markers. You can also configure the bus pattern triggers and signal trigger options.

Sample Number	Bus 1	Bus 2	Signal 1	Signal 2	Time
-7	F9	3FF9	1	1	-350
-6	FA	3FFA	1	1	-300
-5	FB	3FFB	1	1	-250
-4	FC	3FFC	1	1	-200
-3	FD	3FFD	1	1	-150
-2	FE	3FFE	1	1	-100
-1	FF	3FFF	1	1	-50
0	00	0000	0	0	0
1	01	0001	0	0	50
2	02	0002	0	0	100
3	03	0003	0	0	150
4	04	0004	0	0	200
5	05	0005	0	0	250
6	06	0006	0	0	300
7	07	0007	0	0	350

The Listing window is accessed through the menu bar's **Window>Listing**. If you have Tabbed Windows (see [page 313](#)) turned on, you can also select a tab at the bottom of the window.

- To go to different locations in the captured data (see [page 236](#))
- To re-arrange bus/signal columns (see [page 237](#))
- To find a bus/signal column (see [page 237](#))
- To show/hide parts of the listing display (see [page 237](#))
- To insert or delete buses/signals (see [page 238](#))
- Changing Listing Window Properties (see [page 238](#))
 - To change the listing background color (see [page 239](#))
 - To change the timing zoom background color (see [page 239](#))
 - To change the listing font size (see [page 239](#))
 - To lock scrolling with other display windows (see [page 239](#))
 - To show/hide the center rectangle (see [page 239](#))
- Changing Bus/Signal Column Properties (see [page 240](#))
 - To change a bus/signal's data color (see [page 240](#))
 - To change the width of a bus/signal column (see [page 241](#))
 - To change the alignment of a bus/signal column (see [page 241](#))
 - To change a bus/signal's number base (see [page 241](#))
 - To select the marker for marker-relative times (see [page 242](#))
 - To select fixed time units (see [page 242](#))

- See Also
- Defining Buses and Signals (see [page 83](#))
 - Setting Up Quick (Draw Box) Triggers (see [page 143](#))
 - Specifying Simple Triggers (see [page 146](#))
 - Marking, and Measuring Between, Data Points (see [page 245](#))
 - Setting Up Symbols (see [page 132](#))
 - Displaying Names (Symbols) for Bus/Signal Values (see [page 243](#))

- Searching the Captured Data (see [page 264](#))

To go to different locations in the captured data

In the Listing display window, you can go to different locations in the captured data by using the vertical scroll bars, by using the Go To buttons on the standard toolbar, or by choosing Go To commands from popup menus.

To go to different locations using toolbar buttons

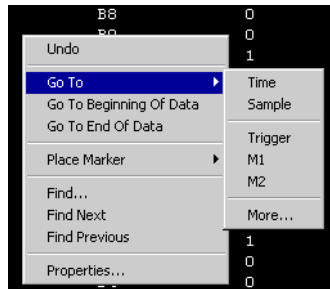
- 1 Click one of the Go To buttons in the standard toolbar.



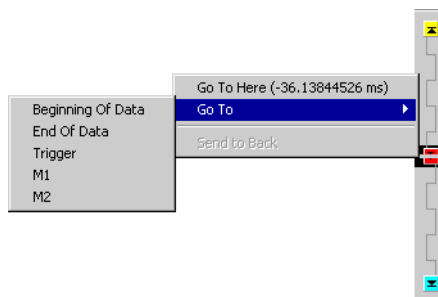
Menu	Description
	Go to Beginning – centers the beginning of the acquisition data.
	Go to Trigger – centers the trigger point of the acquisition.
	Go to End – centers the end of the acquisition data.

To go to different locations using popup menus

- 1 Right-click in the waveform display area, and choose one of the **Go To** commands.



Or, click in the marker overview bar, and choose one of the **Go To** commands.



You can choose **Beginning Of Data**, **End Of Data**, **Trigger**, a marker, a **Time**, or a **Sample**.

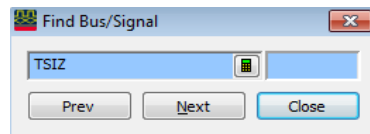
To re-arrange bus/signal columns

- 1 Position the mouse pointer over the bus/signal name associated with the column you want to move.
- 2 Click and hold the mouse button.
- 3 Drag-and-drop the bus/signal to its new position.
The name is placed to the left of the red position indicator that appears.

To find a bus/signal column

When there are many bus/signal columns in the Listing display window, you can search for a particular bus/signal column instead of scrolling through all the columns.

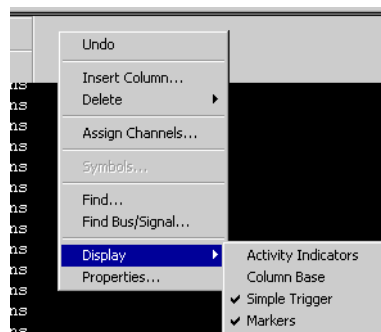
- 1 In the Listing display window, right-click in any Bus/Signal column heading, and choose **Find Bus/Signal...**
- 2 In the Find Bus/Signal dialog, enter the name (or part of the name) of the bus/signal you wish to find.



- 3 Then, click:
 - **Prev** – to search for the string backward in the bus/signal columns.
 - **Next** – to search for the string forward in the bus/signal columns.
 - **Close** – to close the Find Bus/Signal dialog.

To show/hide parts of the listing display

- 1 Right-click in the Bus/Signal column heading of the Listing display, and choose **Display>**.



Then, check or uncheck one of the following to show or hide that part of the Listing display window:

- **Activity Indicators** – either a low bar (low level), high bar (high level), or a transition arrow (transitioning signal) displayed to the left of bus/signal names.
- **Column Base** – the number base row in the column headings.
- **Simple Trigger** – the Simple Trigger row in the column headings (see Specifying Simple Triggers (see [page 146](#))).
- **Markers** – the markers display bar (see [page 407](#)).

You can also make these selections in the **Display Options** area of the Listing Properties dialog's Window Properties tab.

See Also • Changing Listing Window Properties (see [page 238](#))

To insert or delete buses/signals

- To insert buses/signals
- 1 In the Listing display window, right-click in the Bus/Signal column headings; then, choose **Insert Column**.
 - 2 In the Insert dialog, select the buses/signals you want to insert; then, click **OK**.

- To delete selected buses/signals
- 1 Highlight the headings of the bus/signal columns you want to delete (by clicking, Shift-clicking, or Ctrl-clicking the bus/signal names).
 - 2 Right-click in an empty area of the column headings row; then, choose **Delete>Column**.

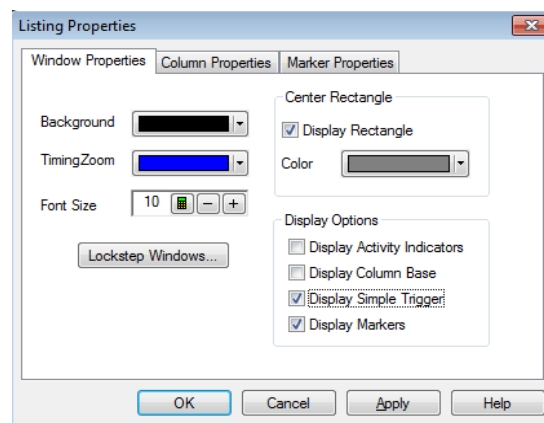
- To delete all buses/signals
- 1 Right-click anywhere in the column headings row; then, choose **Delete>All Columns**.

See Also • Defining Buses and Signals (see [page 83](#))

Changing Listing Window Properties

You can change properties that affect the entire Listing display window.

- 1 Right-click in a blank area of the listing display, and choose **Properties...**
Or, with no bus/signal names selected, choose **Edit>Window Properties...** from the main menu.
- 2 In the Listing Properties dialog's Window Properties tab:



You can:

- Change the listing background color (see [page 239](#))
 - Change the timing zoom background color (see [page 239](#))
 - Change the listing font size (see [page 239](#))
 - Lock scrolling with other display windows (see [page 239](#))
 - Show/hide the center rectangle (see [page 239](#))
 - Show/hide parts of the listing display (see [page 237](#))
- 3 Click **OK** to apply the changes and close the Listing Properties dialog.

See Also • Changing Bus/Signal Column Properties (see [page 240](#))

To change the listing background color

- 1 In the Listing Properties dialog's Window Properties tab, click the **Background** color selection button and select the desired background color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Listing Properties dialog.

To change the timing zoom background color

In the Listing display window, you can give columns from the timing zoom feature a different background color than other bus/signal data columns.

- 1 In the Listing Properties dialog's Window Properties tab, click the **TimingZoom** color selection button and select the desired timing zoom background color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Listing Properties dialog.

To change the listing font size

The font size property adjusts the data display, bus/signal, and simple trigger text size.

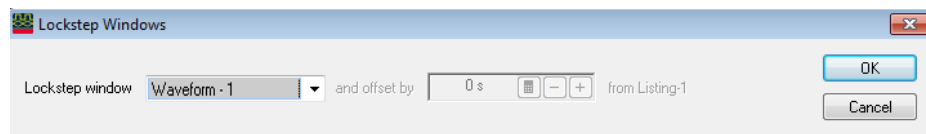
- 1 In the Listing Properties dialog's Window Properties tab, enter the desired **Font Size**.
Fonts can range from size 6 through 72 points.
- 2 Click **OK** to apply the changes and close the Listing Properties dialog.

As the font size is changed, the column width may be automatically increased to fit the new text size.

To lock scrolling with other display windows

You can lock display windows (for example, Waveform, Listing, Compare, etc.) so that when one window is scrolled, others are scrolled as well, such that the same time is centered in each display.

- 1 In the Listing Properties dialog's Window Properties tab, click **Lockstep Windows...**
- 2 In the Lockstep Windows dialog, select the display windows whose scrolling should be locked with this window and specify any offset from this window.



- 3 Click **OK** to close the Lockstep Windows dialog.
- 4 Click **OK** to apply the changes and close the Listing Properties dialog.

To show/hide the center rectangle

The center rectangle is the box that is drawn around the one sample displayed at center of the screen.

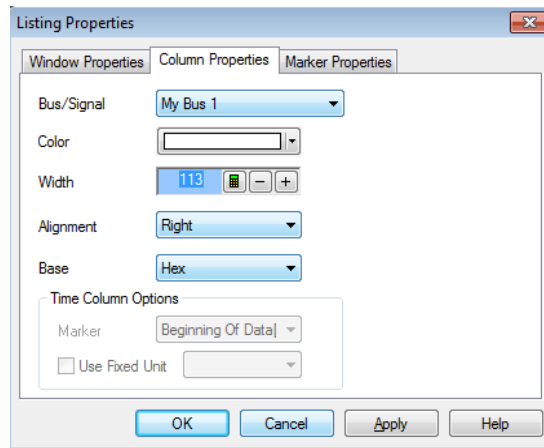
- 1 In the Listing Properties dialog's Window Properties tab's Center Rectangle area, check or uncheck **Display Rectangle** to specify whether the center rectangle is shown or hidden.
If **Display Rectangle** is checked, click the color selection button and select the desired center rectangle color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Listing Properties dialog.

Changing Bus/Signal Column Properties

In the Listing display window, you can change the color, width, alignment, or number base of bus/signal data columns.

To change the properties of a bus/signal data column in the Listing window:

- 1 Right-click on a bus/signal name or on a waveform, and choose **Properties...**
Or, highlight the buses/signals whose properties you want to change (by clicking, Shift-clicking, or Ctrl-clicking the bus/signal names), and choose **Edit>Window Properties...** from the main menu.
- 2 In the Listing Properties dialog's Column Properties tab:



You can:

- Select the **Bus/Signal** to which the property changes apply. You can select:
 - Any bus/signal name that has been assigned (see Defining Buses and Signals (see [page 83](#))).
 - **<all>** buses/signals.
 - **<selected>** buses/signals if more than one column is highlighted.
 - Change a bus/signal's data color (see [page 240](#))
 - Change the width of a bus/signal column (see [page 241](#))
 - Change the alignment of a bus/signal column (see [page 241](#))
 - Change a bus/signal's number base (see [page 241](#))
 - Select the marker for marker-relative times (see [page 242](#))
 - Select fixed time units (see [page 242](#))
- 3 Click **OK** to apply the changes and close the Listing Properties dialog.

See Also • Changing Listing Window Properties (see [page 238](#))

To change a bus/signal's data color

- 1 In the Listing Properties dialog's Column Properties tab, click the **Color** selection button and select the desired bus/signal data color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Listing Properties dialog.

To change the width of a bus/signal column

TIP

You can autosize individual columns by placing the mouse pointer over the right border of the column header box; then, when the pointer icon changes to a resizing pointer, double-click.

TIP

If your keyboard has a numeric keypad, you can autosize all columns by selecting any column header box (to highlight it) and by pressing Ctrl and "+" on the numeric keypad.

1 In the bus/signal headings row of the listing display window, position the mouse pointer over a column separator line; when the cursor changes to a resizing cursor, drag the column border.

Or:

- 1 In the Listing Properties dialog's Column Properties tab, enter the **Width** value in pixels. The minimum column width is 1 pixel, while the maximum width is 1000 pixels.
- 2 Click **OK** to apply the changes and close the Listing Properties dialog.

To change the alignment of a bus/signal column

The Alignment property sets the display of data to be left-justified, right-justified, or centered within the column.

- 1 In the Listing Properties dialog's Column Properties tab, select the **Alignment** from:
 - **Left**
 - **Center**
 - **Right**
- 2 Click **OK** to apply the changes and close the Listing Properties dialog.

To change a bus/signal's number base

The base property specifies the number base to use when displaying the captured data.

- 1 In the Listing Properties dialog's Column Properties tab, select the desired number **Base** from:
 - **Binary**
 - **Hex**
 - **Octal**
 - **Decimal**
 - **Signed Decimal** (two's complement)
 - **Ascii**
 - **Symbol** (see Displaying Names (Symbols) for Bus/Signal Values (see [page 243](#)))

NOTE

If the **Time** column has been selected instead of a data column, your choices change from a numeric format to **Absolute**, **Relative Previous**, or **Relative Marker**.

NOTE

If an analog signal from an external oscilloscope module (see External Oscilloscope Time Correlation and Data Display help) column has been selected instead of a data column, then:

- **Amperage** is used and displayed as the Base for the signal's data if you have enabled the Current Probe checkbox for this signal in the External Oscilloscope Setup dialog box.
 - **Voltage** is used and displayed as the Base for the signal's data if you have NOT enabled the Current Probe checkbox for this signal in the External Oscilloscope Setup dialog box.
-

- 2 Click **OK** to apply the changes and close the Listing Properties dialog.

To select the marker for marker-relative times

In the Listing window, you can display times relative to a marker.

- 1 In the Listing Properties dialog's Column Properties tab, use the **Bus/Signal** selection to select the **Time** column.
- 2 For the **Base** property, select **Relative Marker**.
- 3 For the **Marker** property, select the marker to which relative times should be displayed.
- 4 Click **OK** to apply the changes and close the Listing Properties dialog.

To select fixed time units

In the Listing window, you can display time column values with a fixed unit.

- 1 In the Listing Properties dialog's Column Properties tab, use the **Bus/Signal** selection to select the **Time** column.
- 2 In the Time Column Properties box, check **Use Fixed Unit**; then, select the desired time unit from the drop-down list.
- 3 Click **OK** to apply the changes and close the Listing Properties dialog.

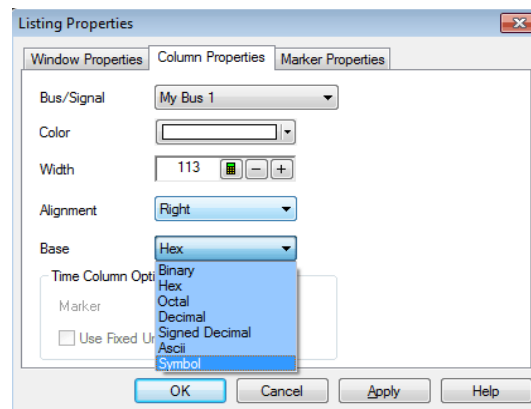
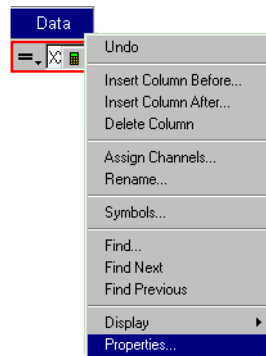
Displaying Names (Symbols) for Bus/Signal Values

You can display a bus or signal using meaningful names rather than numeric values.

Symbols can be displayed in Waveform, Listing, Compare, and Source windows.

To display symbols:

- 1 Set up the symbols (see [page 132](#)).
- 2 Change the number base (see [page 241](#)) of the bus or signal to Symbols.



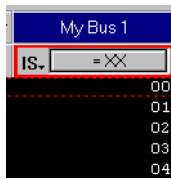
If the symbol is defined as a range, values in the range will be displayed with an offset from the lowest end of the range.

If the definitions of several symbols overlap, the first one listed in the Symbols dialog has precedence over the others.

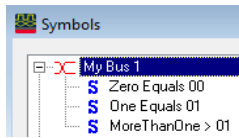
In the Waveform display, "..." will be shown when the full name of the symbol will not fit into the space available.

Once you have set up symbols, it's usually a good idea to save (see [page 198](#)) the logic analyzer configuration. The symbol definitions will be stored as part of the configuration.

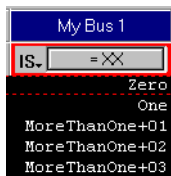
Example Here is what "My Bus 1" looks like before defining any symbols:



When the symbols have been defined, they are shown in the Symbols dialog:



Here is what the bus looks like after the symbols are defined:



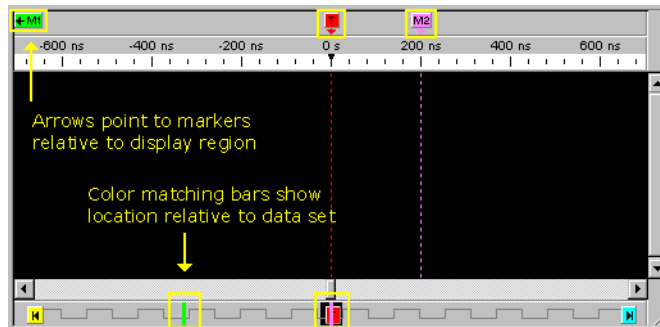
Marking, and Measuring Between, Data Points

Once a marker is created, you can use it as a reference point in the data when measuring intervals or viewing the data value at the marker.

- To read the markers display and overview bars (see [page 245](#))
- To create new markers (see [page 246](#))
- To place markers in data (see [page 247](#))
- To go to a marker (see [page 249](#))
- To center the display about a marker pair (see [page 249](#))
- To change a marker's snap to edge setting (see [page 250](#))
- To delete a marker (see [page 250](#))
- To create a new time interval measurement (see [page 251](#))
- To create a new sample interval measurement (see [page 252](#))
- To create a new value at measurement (see [page 253](#))
- To rename a marker (see [page 254](#))
- To send a marker to the back (see [page 254](#))
- Changing Marker Properties (see [page 255](#))
 - To change a marker's background color (see [page 256](#))
 - To change a marker's foreground color (see [page 256](#))
 - To hide/show a marker (see [page 256](#))
 - To change a marker's lock in viewer setting (see [page 256](#))
 - To lock a marker relative to another marker (see [page 257](#))
 - To add comments to a marker (see [page 257](#))
- Using Y-Axis Markers for Analog Signals (in the Waveform Display) (see [page 257](#))
 - To create new Y-Axis markers (see [page 258](#))
 - To place Y-Axis markers (see [page 259](#))
 - To delete Y-Axis markers (see [page 260](#))
 - To create a new Y-Axis interval measurement (see [page 260](#))
 - To rename a Y-Axis marker (see [page 261](#))
 - To send a Y-Axis marker to the back (see [page 261](#))
 - To change Y-Axis marker properties (see [page 262](#))

- See Also
- Markers Display Bar (see [page 407](#))
 - Marker Measurement Display Bar (see [page 403](#))
 - Markers Menu (see [page 391](#))
 - Markers Toolbar (see [page 400](#))

To read the markers display and overview bars



In the upper markers display bar (see [page 407](#)), markers are color coded and displayed with arrows that point to the marker's location relative to the displayed data.

In the lower markers overview bar (see [page 408](#)), markers are displayed as color coded bars that show the location relative to the complete captured data set.

In the Waveform window (as shown above), the markers display and overview bars appear on the top and bottom of the window. In the Listing window, the markers display and overview bars appear on the left and right sides of the window in a similar way.

TIP

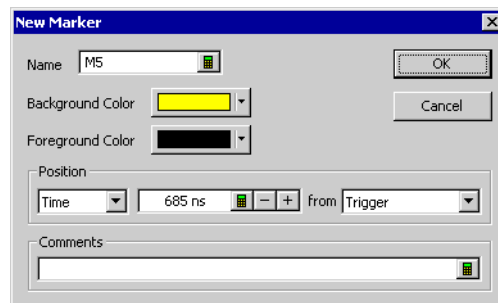
You can quickly display a different region of data by clicking on the markers overview bar at the bottom (waveform) or right side (listing) and selecting **Go To Here** from the popup menu.

To create new markers

When creating a new marker, you can give it a name, specify its color, position it in the data, and add comments. Up to 1024 markers can be created.

- 1 From the menu bar, select **Markers>New...**
- 2 In the New Marker dialog, enter the marker name.

You can specify both a *long name* and an *abbreviated name* by using the "Long name[abbreviated name]" syntax; for example, "Location A[A]". When an abbreviated name is used, it appears on the marker in the marker display bar (see [page 407](#)) while the long name appears in the marker *tool tip* (see [page 619](#)).



- 3 Select the marker's background and foreground colors.
- 4 Specify the position of the new marker in the data by:
 - **Time** - positions the marker by a time value from a reference point. Reference points are the Trigger, Beginning of Data, End of Data, or another marker.

- **Sample** - positions the marker by a number of samples from a reference point. Reference points are the Trigger, Beginning of Data, End of Data, or another marker.
- **Value** - positions the marker at an occurrence of a bus/signal pattern. Click **Occurs...** to specify the bus/signal pattern value.

Bus/signal pattern specification is the same as when searching the captured data (see [page 264](#)).

- 5 Enter comments for the marker.
Comments appear in the marker's *tool tip* (see [page 619](#)).
- 6 Click **OK**.

- See Also
- To place markers in data (see [page 247](#))
 - To go to a marker (see [page 249](#))
 - To read the markers display and overview bars (see [page 245](#))
 - To center the display about a marker pair (see [page 249](#))
 - To delete a marker (see [page 250](#))
 - To rename a marker (see [page 254](#))
 - To send a marker to the back (see [page 254](#))
 - To change a marker's snap to edge setting (see [page 250](#))
 - Changing Marker Properties (see [page 255](#))

To place markers in data

Use Place Markers to quickly position a marker in the data. Depending on how you access the Place Markers feature, the marker is placed in the data a little differently. You can also move markers by dragging them with the mouse or by using the front-panel knobs.

NOTE

An enabled **Snap to Edge** property affects a marker's placement in the Waveform window if the mouse cursor is over a waveform when dragging and dropping or when placing at the mouse cursor.

- To drag and drop markers in data (see [page 247](#))
- To place marker at the mouse cursor (see [page 248](#))
- To place marker at center screen (see [page 248](#))
- To change a marker's position property (see [page 248](#))
- You can also place markers where data is found when searching (see [To specify "found" marker placement \(see page 272\)](#)).

To drag and drop markers in data

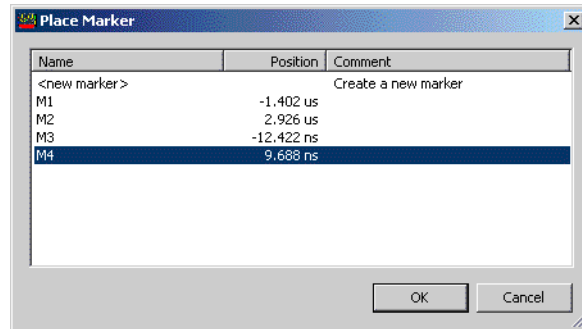
Using the drag and drop feature you can move markers to new positions in the data.

- 1 Click and hold down the mouse button on the marker you wish to move.
- 2 Move the mouse cursor to the new position.

When moving a marker in the Waveform display window, if the mouse cursor is over a waveform and the marker's **Snap to Edge** property is enabled, the cursor changes to a green "direction arrow" indicating the direction of the next valid edge. A yellow "cross hair" target is placed on the edge at which the marker will be placed if you decide to release the mouse button. If you don't want the marker to snap to an edge, move the mouse cursor so that it is not over any waveforms before releasing the mouse button.

- 3 Release the mouse button to reposition the marker.

- To place marker at the mouse cursor
- 1 Point the mouse to the desired data point in the display.
 - 2 Right-click, and select **Place Marker>Time>(desired marker)**.
If the mouse cursor is over a waveform and the marker's **Snap to Edge** property is enabled, the marker is placed at nearest waveform edge; otherwise, the marker is placed at the mouse cursor location.
- To place marker at center screen
- 1 From the menu bar click **Markers>Place On Screen...**
 - 2 In the Place Marker dialog, select the desired marker.



You can sort the list of markers by clicking on the Name, Position, or Comment column headings.

- 3 Click **OK**.
The marker will be placed at mid-screen.
- To change a marker's position property
- 1 Right-click on a marker, and choose **Properties...**
Or, when viewing a display window that has markers, choose **Markers>Properties...** from the main menu.
 - 2 In the display window properties dialog's Time Marker Properties tab, select the **Marker** to which the property changes apply.
 - 3 In the **Position** box, select what to position the marker by:
 - **Time** - positions the marker by a time value from a reference point. Reference points are the Trigger, Beginning of Data, End of Data, or another marker.
 - **Sample** - positions the marker by a number of samples from a reference point. Reference points are the Trigger, Beginning of Data, End of Data, or another marker.
 - **Value** - positions the marker at an occurrence of a bus/signal pattern. Click **Occurs...** to specify the bus/signal pattern value.
Bus/signal pattern specification is the same as when searching the captured data (see [page 264](#)), except you can click **Properties...** to open the Value Properties dialog:
 - Check **Stop repetitive run** if you want to stop a repetitive run when the specified bus/signal pattern is found (or not found).
 - Check **Send e-mail** if you want to send an e-mail when the specified bus/signal pattern is found (or not found); then, click the **E-mail...** button. In the E-mail dialog (see [page 432](#)), enter the address to which e-mail will be sent, the subject, and the text of the message.
 - 4 Click **OK** to apply the changes and close the properties dialog.

- See Also
- To specify "found" marker placement (see [page 272](#)) (for placing markers where data is found)
 - To create new markers (see [page 246](#))
 - To go to a marker (see [page 249](#))


- To read the markers display and overview bars (see [page 245](#))
- To center the display about a marker pair (see [page 249](#))
- To delete a marker (see [page 250](#))
- To rename a marker (see [page 254](#))
- To send a marker to the back (see [page 254](#))
- To change a marker's snap to edge setting (see [page 250](#))
- Changing Marker Properties (see [page 255](#))

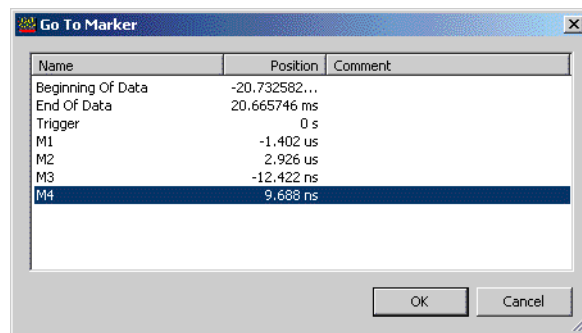
To go to a marker

To quickly find a previously set marker in the data, or to go to the beginning of data, end of data, or the trigger point:

- Click in the markers display bar (see [page 407](#)) or the markers overview bar (see [page 408](#)), and choose **Go To** from the popup menu.

Or:

- 1 From the menu bar, select **Markers>Go To...** or select the  icon in the markers toolbar (see [page 400](#)).
- 2 In the Go To Marker dialog, select the marker you wish to find from the list provided.



You can sort the list of markers by clicking on the Name, Position, or Comment column headings.

- 3 Click **OK**.

The selected marker appears at the center of the display.

See Also

- To create new markers (see [page 246](#))
- To place markers in data (see [page 247](#))
- To center the display about a marker pair (see [page 249](#))
- To delete a marker (see [page 250](#))
- To rename a marker (see [page 254](#))
- To send a marker to the back (see [page 254](#))
- To change a marker's snap to edge setting (see [page 250](#))
- Changing Marker Properties (see [page 255](#))

To center the display about a marker pair

Use the center about feature to center the display around a selected marker pair. If the marker pair is separated by a large time or sample amount, the scale of the display is automatically changed so both markers appear on screen.

Since the center about feature centers the display around a pair (two) markers, if you have three or more markers defined, you will have available choices for all possible combinations of two.

- 1 From the menu bar, select **Markers>Center About...**
- 2 In the Center About dialog, select the desired marker combination.
- 3 Click **OK**.

The data between the two markers is displayed.

- See Also
- To create new markers (see [page 246](#))
 - To place markers in data (see [page 247](#))
 - To go to a marker (see [page 249](#))
 - To delete a marker (see [page 250](#))
 - To rename a marker (see [page 254](#))
 - To send a marker to the back (see [page 254](#))
 - To change a marker's snap to edge setting (see [page 250](#))
 - Changing Marker Properties (see [page 255](#))

To change a marker's snap to edge setting

- 1 In a display window with markers, right-click on the marker, and choose **Snap to Edge**.
Or, in the display window properties dialog's Time Marker Properties tab, check or uncheck **Snap to Edge** to enable or disable the marker's snap to edge behavior.

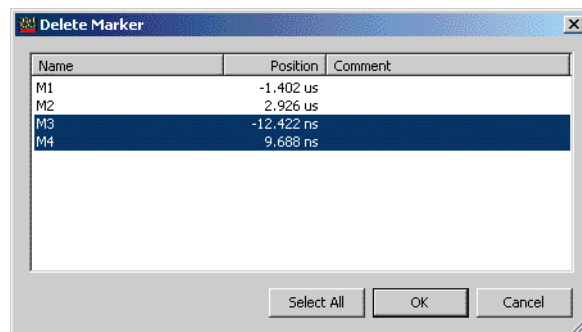
- See Also
- To place markers in data (see [page 247](#))
 - Changing Marker Properties (see [page 255](#))

To delete a marker

- In the markers display bar (see [page 407](#)), click the marker you want to delete, and choose **Delete** from the popup menu (or choose **Delete All** to delete all markers).

Or:

- 1 From the menu bar, select **Markers>Delete...**
- 2 In the Delete Marker dialog, select the markers you wish to delete.



You can sort the list of markers by clicking on the Name, Position, or Comment column headings.


- 3 Click **OK**.

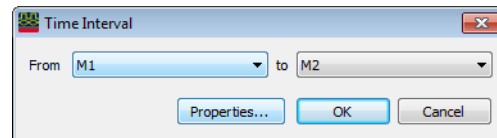
- See Also
- To create new markers (see [page 246](#))
 - To place markers in data (see [page 247](#))

- To go to a marker (see [page 249](#))
- To center the display about a marker pair (see [page 249](#))
- To rename a marker (see [page 254](#))
- To send a marker to the back (see [page 254](#))
- To change a marker's snap to edge setting (see [page 250](#))
- Changing Marker Properties (see [page 255](#))

To create a new time interval measurement

Use the new time interval measurement feature to measure a time interval between two specified points in the captured data. Measurement results are displayed in the marker measurement display bar (see [page 403](#)).

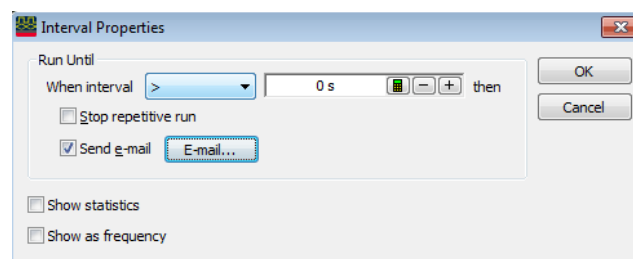
- 1 From the menu bar select **Markers>New Time Interval Measurement**, or click the  icon in the markers toolbar (see [page 400](#)).
- 2 In the Time Interval dialog, select the markers you want to measure time between.



NOTE

If you have selected a "System Trigger - (module)" or "Trigger - (module)" marker, make sure you uncheck the **First module to trigger designates the System Trigger** option in the Module Skew and System Trigger dialog (see [page 446](#)). Otherwise, during a run, the system trigger could switch from one module to another, causing the module markers to be re-assigned and any measurements using these markers to be deleted.

- 3 To specify interval properties, click **Properties...**



In the Interval Properties dialog:

- Check **Stop repetitive run** if you want to stop a repetitive run when the specified interval value is measured.
- Check **Send e-mail** if you want to send an e-mail when the specified interval value is measured; then, click the **E-mail...** button. In the E-mail dialog (see [page 432](#)), enter the address to which e-mail will be sent, the subject, and the text of the message.
- Check **Show statistics** if you want to show repetitive run statistics.

CAUTION

If you have selected a "System Trigger - (module)" or "Trigger - (module)" marker as one of the markers you want to measure time between, make sure you uncheck the **First module to trigger designates the System Trigger** option in the Module Skew and System Trigger dialog (see [page 446](#)). Otherwise, during the repetitive run, the system trigger could switch from one module to another, causing you to lose all the statistical data you have accumulated.

- Check **Show as frequency** if you want to show the measured frequency of changes in the interval.
 - Click **OK** when you are done specifying interval properties.
- 4 Click **OK** to close the Time Interval dialog.

The result of the interval measurement is displayed in the marker measurements display bar:


Beginning Of Data to Trigger = 3.860812 ms (511.164 us / 3.860812 ms / 2.279915 ms)

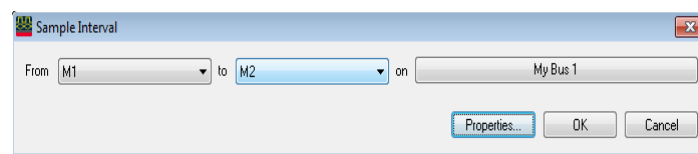
If statistics are shown, the low, high, and average interval measurements are included.

- See Also
- To create a new sample interval measurement (see [page 252](#))
 - To create a new value at measurement (see [page 253](#))

To create a new sample interval measurement

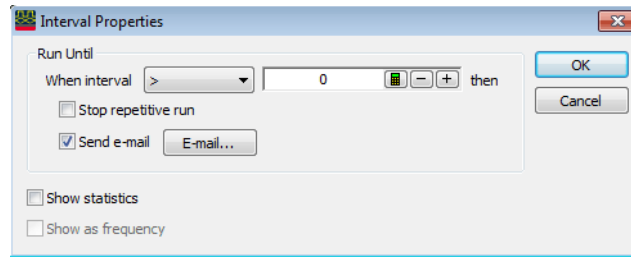
Use the new sample interval measurement feature to measure the number of samples between two specified points in the captured data. Measurement results are displayed in the marker measurement display bar (see [page 403](#)).

- 1 From the menu bar select **Markers>New Sample Interval Measurement**, or click the  icon in the markers toolbar (see [page 400](#)).
- 2 In the Sample Interval dialog, select the markers you want to measure samples between, and select the bus/signal.

**NOTE**

If you have selected a "System Trigger - (module)" or "Trigger - (module)" marker, make sure you uncheck the **First module to trigger designates the System Trigger** option in the Module Skew and System Trigger dialog (see [page 446](#)). Otherwise, during a run, the system trigger could switch from one module to another, causing the module markers to be re-assigned and any measurements using these markers to be deleted.

- 3 To specify interval properties, click **Properties....**



In the Interval Properties dialog:

- Check **Stop repetitive run** if you want to stop a repetitive run when the specified interval value is measured.
- Check **Send e-mail** if you want to send an e-mail when the specified interval value is measured; then, click the **E-mail...** button. In the E-mail dialog (see [page 432](#)), enter the address to which e-mail will be sent, the subject, and the text of the message.
- Check **Show statistics** if you want to show repetitive run statistics.

CAUTION

If you have selected a "System Trigger - (module)" or "Trigger - (module)" marker as one of the markers you want to measure samples between, make sure you uncheck the **First module to trigger designates the System Trigger** option in the Module Skew and System Trigger dialog (see [page 446](#)). Otherwise, during the repetitive run, the system trigger could switch from one module to another, causing you to lose all the statistical data you have accumulated.

- Click **OK** when you are done specifying interval properties.

4 Click **OK** to close the Sample Interval dialog.

The result of the interval measurement is displayed in the marker measurements display bar:


```
MI to M2 = 1442011 (393733 / 1442011) [My Bus 1]
```

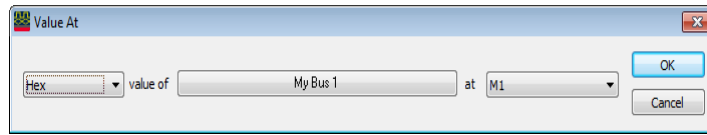
If statistics are shown, the low and high interval measurements are included.

- See Also
- To create a new time interval measurement (see [page 251](#))
 - To create a new value at measurement (see [page 253](#))

To create a new value at measurement

Use the new value at measurement feature to measure the value of a bus/signal at a specified marker location in the captured data. Measurement results are displayed in the marker measurement display bar (see [page 403](#)).

- 1 From the menu bar select **Markers>New Value At Measurement**, or click the  icon in the markers toolbar (see [page 400](#)).
- 2 In the Value At dialog, select the numeric base of the data, the bus/signal, and the marker.

**NOTE**

If you have selected a "System Trigger - (module)" or "Trigger - (module)" marker, make sure you uncheck the **First module to trigger designates the System Trigger** option in the Module Skew and System Trigger dialog (see [page 446](#)). Otherwise, during a run, the system trigger could switch from one module to another, causing the module markers to be re-assigned and any measurements using these markers to be deleted.

- 3 Click **OK**.

The result of the value at measurement `My Bus 1@M1 = 4B` is displayed in the marker measurement display bar.

- See Also
- To create a new time interval measurement (see [page 251](#))
 - To create a new sample interval measurement (see [page 252](#))

To rename a marker

You can give markers any name you choose.

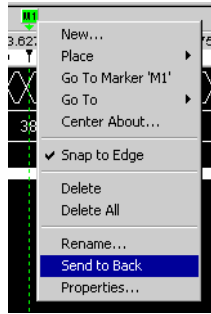
- 1 In a display window with markers, right-click on the marker, and choose **Rename...**
Or, in the display window properties dialog's Time Marker Properties tab, select the **Marker**, and click **Rename...**
- 2 In the Rename dialog, enter the new marker name.
You can specify both a *long name* and an *abbreviated name* by using the "Long name[abbreviated name]" syntax; for example, "Location A[A]". When an abbreviated name is used, it appears on the marker in the marker display bar (see [page 407](#)) while the long name appears in the marker *tool tip* (see [page 619](#)).
- 3 Click **OK**.

- See Also
- To create new markers (see [page 246](#))
 - To place markers in data (see [page 247](#))
 - To go to a marker (see [page 249](#))
 - To center the display about a marker pair (see [page 249](#))
 - To delete a marker (see [page 250](#))
 - To send a marker to the back (see [page 254](#))
 - To change a marker's snap to edge setting (see [page 250](#))
 - Changing Marker Properties (see [page 255](#))

To send a marker to the back

When markers overlap on the display, you can send the visible marker to the back in order to see the marker underneath.

- 1 Click the marker you wish to send to the back, and choose **Send to Back** from the pop-up menu.

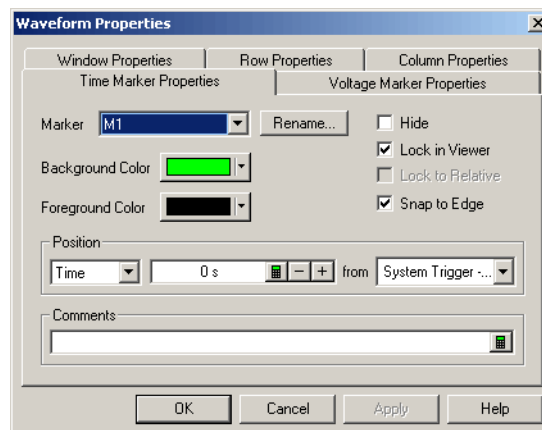


- See Also
- To create new markers (see [page 246](#))
 - To place markers in data (see [page 247](#))
 - To go to a marker (see [page 249](#))
 - To center the display about a marker pair (see [page 249](#))
 - To delete a marker (see [page 250](#))
 - To rename a marker (see [page 254](#))
 - To change a marker's snap to edge setting (see [page 250](#))
 - Changing Marker Properties (see [page 255](#))

Changing Marker Properties

Once a marker is created, you can modify any of its attributes from the Time Marker Properties tab..

- 1 Right-click on a marker, and choose **Properties...**
Or, when viewing a display window that has markers, choose **Markers>Properties...** from the main menu.
- 2 In the display window properties dialog's Time Marker Properties tab:



You can:

- Select the **Marker** to which the property changes apply.

- Rename a marker (see [page 254](#))
 - Change a marker's background color (see [page 256](#))
 - Change a marker's foreground color (see [page 256](#))
 - Change a marker's position property (see [page 248](#))
 - Hide/show a marker (see [page 256](#))
 - Change a marker's lock in viewer setting (see [page 256](#))
 - Lock a marker relative to another marker (see [page 257](#))
 - Change a marker's snap to edge setting (see [page 250](#))
 - Add comments to a marker (see [page 257](#))
- 3 Click **OK** to apply the changes and close the properties dialog.

- See Also
- To create new markers (see [page 246](#))
 - To place markers in data (see [page 247](#))
 - To go to a marker (see [page 249](#))
 - To center the display about a marker pair (see [page 249](#))
 - To delete a marker (see [page 250](#))
 - To rename a marker (see [page 254](#))
 - To send a marker to the back (see [page 254](#))
 - To read the markers display and overview bars (see [page 245](#))

To change a marker's background color

- 1 In the Marker Properties tab, click the **Background Color** selection button and select the desired color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the properties dialog.

- See Also
- To change Y-Axis marker properties (see [page 262](#))

To change a marker's foreground color

- 1 In the Marker Properties tab, click the **Foreground Color** selection button and select the desired color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the properties dialog.

- See Also
- To change Y-Axis marker properties (see [page 262](#))

To hide/show a marker

- 1 In the Marker Properties tab, check or uncheck **Hide** to hide or show the marker.
When a marker is hidden, all other marker properties are retained; the marker is just hidden from view in the display.
- 2 Click **OK** to apply the changes and close the properties dialog.

- See Also
- To change Y-Axis marker properties (see [page 262](#))

To change a marker's lock in viewer setting

When a marker's **Lock in Viewer** setting is enabled, moving or placing the marker in one display window causes other display windows to be updated so that the marker appears in them as well.

NOTE

The lock in viewer behavior applies only when a marker is dragged within the immediate data viewing area. If a marker is moved by defining a new position in the Time Markers Properties tab, the marker is not guaranteed to be visible in other display windows.

- 1 In the Time Marker Properties tab, check or uncheck **Lock in Viewer** to enable or disable the setting.
- 2 Click **OK** to apply the changes and close the properties dialog.

To lock a marker relative to another marker

When a marker is positioned relative to another marker and the marker's **Lock to Relative** setting is enabled, moving or placing either marker causes both to move such that the time between the markers remains the same. Both markers must be movable.

- 1 In the Marker Properties tab's **Position** box:
 - a Select **Time**.
 - b Select the relative marker.
 - c Enter the relative time between markers.
- 2 Check or uncheck **Lock to Relative** to lock or unlock relative marker movements.
- 3 Click **OK** to apply the changes and close the properties dialog.

To add comments to a marker

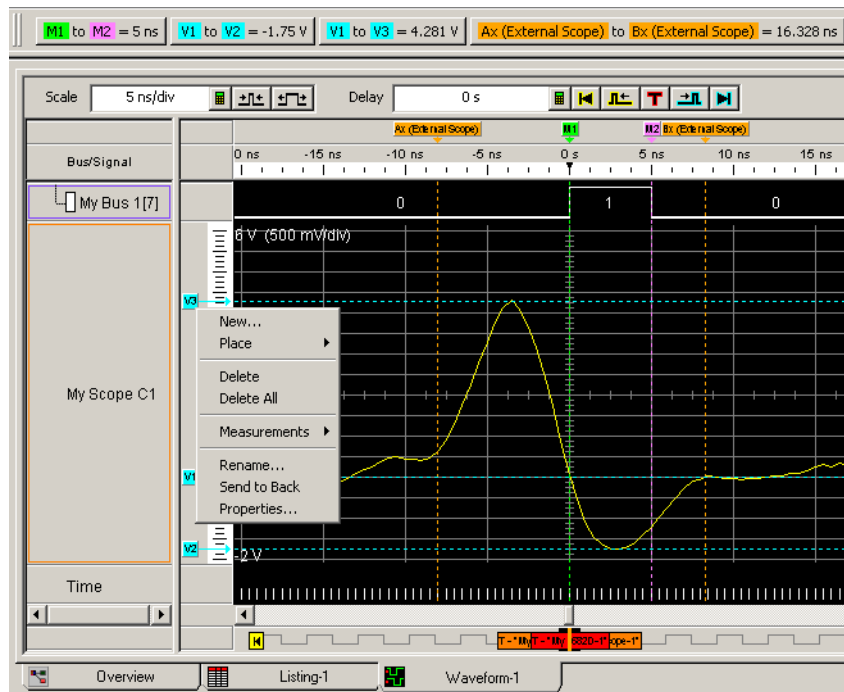
You can add comments to a marker that appear in the marker's *tool tip* (see [page 619](#)).

- 1 In the Marker Properties tab's **Comments** box, enter your comments.
- 2 Click **OK** to apply the changes and close the properties dialog.

See Also • To change Y-Axis marker properties (see [page 262](#))

Using Y-Axis Markers for Analog Signals (in the Waveform Display)

When analog signals are added to the Waveform display window (from an external oscilloscope module), you can add Y-Axis (Voltage / Amperage) markers and interval measurements. If the signal's data is displayed in Amperage, then Amperage is used in the creation of a Y-Axis marker for the signal. If the signal's data is displayed in Voltage, then Voltage is used in the creation of a Y-Axis marker for the signal.



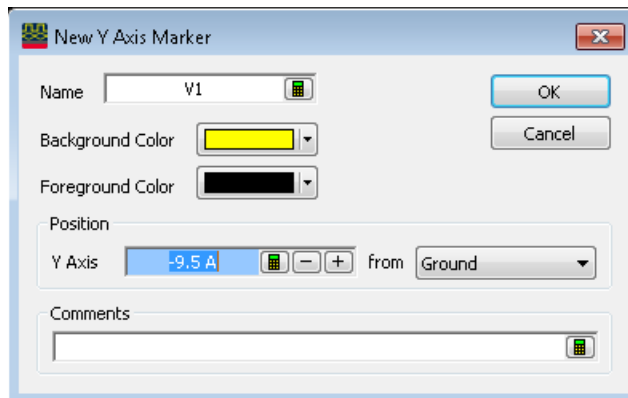
- To create new Y-Axis markers (see [page 258](#))
- To place Y-Axis markers (see [page 259](#))
- To delete Y-Axis markers (see [page 260](#))
- To create a new voltage interval measurement (see [page 260](#))
- To rename a Y-Axis marker (see [page 261](#))
- To send a Y-Axis marker to the back (see [page 261](#))
- To change Y-Axis marker properties (see [page 262](#))

See Also • " External Oscilloscope Time Correlation and Data Display" (in the online help)

To create new Y-Axis markers

When creating a new Y-Axis marker, you can give it a name, specify its color, position it, and add comments.

- 1 In an analog signal row's Y-Axis marker/vertical scale display bar (to the left of the waveform), click (where you would like to place the marker) and choose **New...**
Or, right-click on an analog signal waveform (where you would like to place the marker) and choose **Place Marker>Y-Axis>New Marker...**
- 2 In the New Y-Axis Marker dialog, enter the marker name.
You can specify both a *long name* and an *abbreviated name* by using the "Long name[abbreviated name]" syntax; for example, "Location A[A]". When an abbreviated name is used, it appears on the marker in the Y-Axis marker display bar while the long name appears in the marker *tool tip* (see [page 619](#)).



- 3 Select the marker's background and foreground colors.
- 4 Specify the position of the new Y-Axis marker by its voltage / amperage from ground or another Y-Axis marker. If the signal's data is displayed in Amperage, then Amperage is used in the creation of the Y-Axis marker for the signal. If the signal's data is displayed in Voltage, then Voltage is used in the creation of the Y-Axis marker for the signal.
- 5 Enter comments for the Y-Axis marker.
Comments appear in the marker's *tool tip* (see [page 619](#)).
- 6 Click **OK**.

- See Also
- To place Y-Axis markers (see [page 259](#))
 - To delete Y-Axis markers (see [page 260](#))
 - To create a new Y-Axis interval measurement (see [page 260](#))
 - To rename a Y-Axis marker (see [page 261](#))
 - To send a Y-Axis marker to the back (see [page 261](#))
 - To change Y-Axis marker properties (see [page 262](#))

To place Y-Axis markers

Use Place Markers to quickly position a Y-Axis marker. Depending on how you access the Place Markers feature, the marker is placed in the data a little differently. You can also move markers by dragging them with the mouse or by using the front-panel knobs. Where Y-Axis markers intersect time markers, you can drag both markers at the same time.

- To drag and drop Y-Axis markers (see [page 259](#))
- To place a Y-Axis marker at the mouse cursor (see [page 259](#))
- To change a Y-Axis marker's position property (see [page 259](#))

To drag and drop Y-Axis markers Using the drag and drop feature you can move Y-Axis markers to new positions in the data.

- 1 Click and hold down the mouse button on the marker you wish to move.
- 2 Move the mouse cursor to the new position.
- 3 Release the mouse button to reposition the marker.

To place a Y-Axis marker at the mouse cursor

- 1 Point the mouse to the desired data point in the display.
- 2 Right-click, and select **Place Marker>Y-Axis>(desired marker)**.

To change a Y-Axis marker's position property

- 1 Right-click on a Y-Axis marker, and choose **Properties...**
Or, when viewing a display window that has Y-Axis markers, choose **Markers>Properties...** from the main menu.

- 2 In the Waveform Properties dialog's Y-Axis Marker Properties tab, select the **Marker** to which the property changes apply.
- 3 In the **Position** box, position the marker by its voltage / amperage from ground or another Y-Axis marker.
- 4 Click **OK** to apply the changes and close the properties dialog.

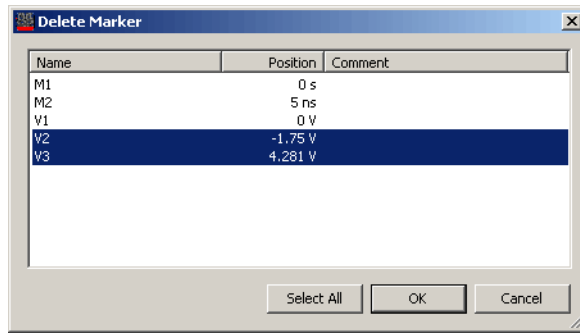
- See Also
- To create new Y-Axis markers (see [page 258](#))
 - To delete Y-Axis markers (see [page 260](#))
 - To create a new Y-Axis interval measurement (see [page 260](#))
 - To rename a Y-Axis marker (see [page 261](#))
 - To send a Y-Axis marker to the back (see [page 261](#))
 - To change Y-Axis marker properties (see [page 262](#))

To delete Y-Axis markers

- In an analog signal's Y-Axis markers/vertical scale display bar (to the left of the waveform), click the Y-Axis marker you want to delete, and choose **Delete** from the popup menu (or choose **Delete All** to delete all Y-Axis markers).

Or:

- 1 From the menu bar, select **Markers>Delete...**
- 2 In the Delete Marker dialog, select the markers you wish to delete.



You can sort the list of markers by clicking on the Name, Position, or Comment column headings.

- 3 Click **OK**.

- See Also
- To create new Y-Axis markers (see [page 258](#))
 - To place Y-Axis markers (see [page 259](#))
 - To create a new Y-Axis interval measurement (see [page 260](#))
 - To rename a Y-Axis marker (see [page 261](#))
 - To send a Y-Axis marker to the back (see [page 261](#))
 - To change Y-Axis marker properties (see [page 262](#))

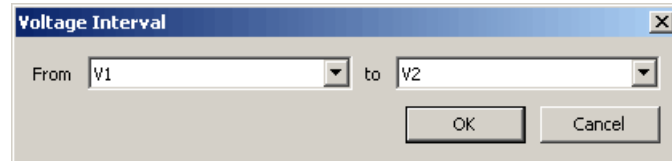
To create a new Y-Axis interval measurement

Use the Y-Axis interval measurement feature to measure a voltage / amperage between two Y-Axis markers. Measurement results are displayed in the marker measurement display bar (see [page 403](#)).

- 1 In an analog signal row's Y-Axis marker/vertical scale display bar (to the left of the waveform), click and choose **Measurements>New Y-Axis Interval Measurement**.

Or, right-click in the marker measurement display bar and choose **New Y-Axis Interval Measurement>(analog signal name)**.

- 2 In the Y-Axis Interval dialog, select the markers you want to measure voltage between.



- 3 Click **OK** to close the Y-Axis Interval dialog.

The result of the interval measurement is displayed in the marker measurements display bar:

V1 to V2 = -1.75 V

To rename a Y-Axis marker

You can give Y-Axis markers any name you choose.

- 1 In a Waveform window with voltage markers, right-click on the marker, and choose **Rename...**

Or, in the display window properties dialog's Y-Axis Marker Properties tab, select the **Marker**, and click **Rename...**

- 2 In the Rename dialog, enter the new marker name.

You can specify both a *long name* and an *abbreviated name* by using the "Long name[abbreviated name]" syntax; for example, "Location A[A]". When an abbreviated name is used, it appears on the marker in the Y-Axis marker display bar while the long name appears in the marker *tool tip* (see [page 619](#)).

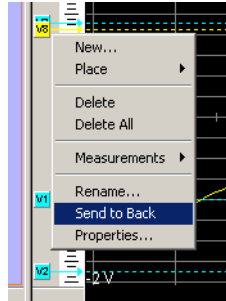
- 3 Click **OK**.

- See Also
- To create new Y-Axis markers (see [page 258](#))
 - To place Y-Axis markers (see [page 259](#))
 - To delete Y-Axis markers (see [page 260](#))
 - To create a new Y-Axis interval measurement (see [page 260](#))
 - To send a Y-Axis marker to the back (see [page 261](#))
 - To change Y-Axis marker properties (see [page 262](#))

To send a Y-Axis marker to the back

When Y-Axis markers overlap on the display, you can send the visible marker to the back in order to see the marker underneath.

- 1 Click the Y-Axis marker you wish to send to the back, and choose **Send to Back** from the pop-up menu.

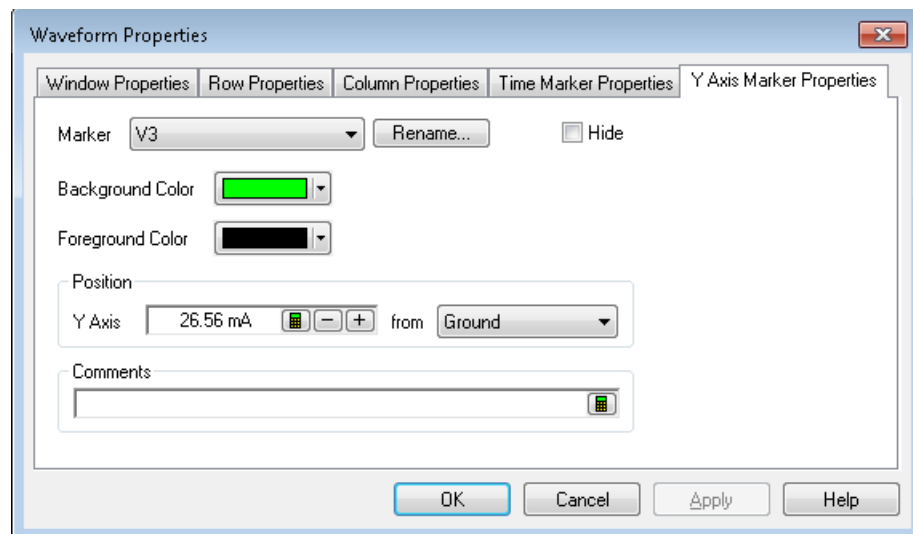


- See Also
- To create new voltage markers (see [page 258](#))
 - To place Y-Axis markers (see [page 259](#))
 - To delete Y-Axis markers (see [page 260](#))
 - To create a new Y-Axis interval measurement (see [page 260](#))
 - To rename a Y-Axis marker (see [page 261](#))
 - To change Y-Axis marker properties (see [page 262](#))

To change Y-Axis marker properties

Once a Y-Axis marker is created, you can modify any of its attributes from the Y-Axis Marker Properties tab.

- 1 Right-click on a Y-Axis marker, and choose **Properties...**
Or, when viewing a Waveform display window that has Y-Axis markers, choose **Markers>Properties...** from the main menu.
- 2 In the Waveform Properties dialog's Y-Axis Marker Properties tab:



You can:

- Select the **Marker** to which the property changes apply.

- Rename a Y-Axis marker (see [page 261](#))
 - Change a marker's background color (see [page 256](#))
 - Change a marker's foreground color (see [page 256](#))
 - Change a Y-Axis marker's position property (see [page 259](#))
 - Hide/show a marker (see [page 256](#))
 - Add comments to a marker (see [page 257](#))
- 3 Click **OK** to apply the changes and close the properties dialog.

- See Also
- To create new Y-Axis markers (see [page 258](#))
 - To place Y-Axis markers (see [page 259](#))
 - To delete Y-Axis markers (see [page 260](#))
 - To create a new Y-Axis interval measurement (see [page 260](#))
 - To rename a Y-Axis marker (see [page 261](#))
 - To send a Y-Axis marker to the back (see [page 261](#))

Searching the Captured Data

You can search for bus/signal patterns in the captured data.

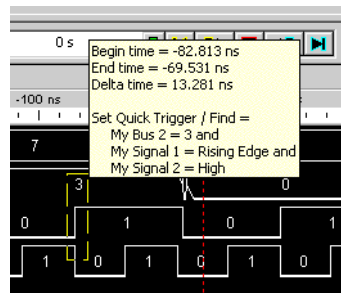
- To quickly find bus/signal patterns (see [page 264](#))
- To find bus/signal patterns in the captured data (see [page 265](#))
- To find packet patterns in the captured data (see [page 267](#))
- To find complex patterns in the captured data (see [page 270](#))
- To store, recall, or delete favorite find patterns (see [page 271](#))
- To specify "found" marker placement (see [page 272](#))

To quickly find bus/signal patterns

In the Waveform or Listing windows, you can quickly draw a rectangle and find the next or previous occurrence of that bus/signal pattern.

- 1 In the Waveform window, make sure the Fast Zoom In (see [page 230](#)) option is not selected.
- 2 Using the mouse, point to the upper-left corner of your desired bus/signal pattern.
- 3 While holding down the mouse button, drag the mouse pointer to the lower-right corner of your bus/signal pattern.

As you draw the rectangle, a tool tip shows the selected bus/signal pattern.



As you move the mouse left-to-right and top-to-bottom, the signal edge/level or bus value in contact with the **left of the rectangle** becomes the bus/signal pattern.

Only one edge can be set.

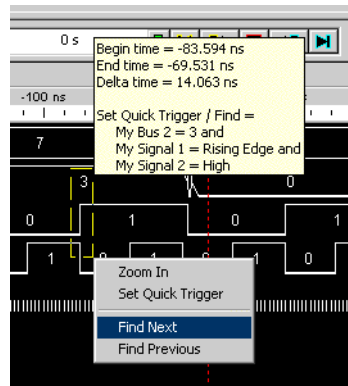
If a bus is expanded into its separate signals, three conditions apply:

- a If drawing starts on a bus, none of its expanded signals can be included.
- b If drawing starts on a signal, the bus cannot be included.
- c Edges and levels are mutually exclusive. That is, either one edge can be set, or all levels can be set, but not both at the same time.

NOTE

In the Waveform display window, it may be necessary to redraw the rectangle if you do not get your desired bus/signal pattern dictated by the left-side line of the rectangle. You could also try drawing the rectangle backwards leaving the left-side rectangle line set last.

- 4 When the desired bus/signal pattern has been selected, release the mouse button, and select **Find Next** or **Find Previous** from the popup menu.



- 5 Click the Previous  or Next  icons to see more occurrences.

General Guidelines


- Any bus/signals with overlapping bits are not included within the bus/signal pattern.
Example: Bus_1 has channels 0 through 7 of pod 1 assigned and Bus_2 has channels 3 through 6 of pod 1 assigned. At this point, you have the same probed signals (channels 3 through 6 of pod 1) assigned in both Bus_1 and Bus_2. Now, you draw the rectangle over both bus_1 and bus_2. Because Bus_1 channels 3 through 6 are repeated (overlapped) on Bus_2, they will not be included in the bus/signal pattern.

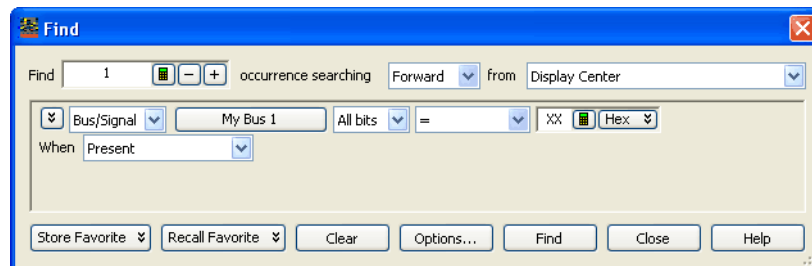
See Also

- To find bus/signal patterns in the captured data (see [page 265](#))
- To find packet patterns in the captured data (see [page 267](#))
- To find complex patterns in the captured data (see [page 270](#))
- To store, recall, or delete favorite find patterns (see [page 271](#))

To find bus/signal patterns in the captured data



This search option locates a specified data pattern. You can qualify your search by specific bits, data patterns, equality, and range operators. The search result is placed at the center of the display.

- 1 From the menu bar, choose **Edit>Find...**, or click the  icon in the standard toolbar (see [page 398](#)).
- 2 In the **Find** dialog, enter the number of the occurrences you wish to find, select whether you want to search forward or backward from the start location; then, select the start location.



- 3 In the event specification area, select the **Bus/Signal** pattern type.
- 4 Specify the bus/signal pattern event you wish to locate.

In addition to the usual pattern matching operators (=, !=, <, >, <=, >=, **In Range**, and **Not In Range**), there are three additional operators you can use:

- **Entering** – the first sample of one or more consecutive samples that match the pattern. (By comparison, the "=" equals operator considers every sample that matches the pattern as an occurrence.)
 - **Exiting** – the sample after one or more consecutive samples that match the pattern.
 - **Transitioning** – entering or exiting one or more consecutive samples that match the pattern.
- You can find analog signal values as well as digital signal values.
- 5 Select the desired **When** find qualifier (which further qualifies the find criteria with a time duration or other operator):
 - **Present**
 - **Not Present**
 - **Present**> (time duration)
 - **Present**>= (time duration)
 - **Present**< (time duration)
 - **Present**<= (time duration)
 - **Present for Range** (of time)
 - **Not Present for Range** (of time)
 - **Entering**
 - **Exiting**
 - **Transitioning**
 - 6 Click **Find**.
 - 7 Click the Previous  or Next  icons to see more occurrences.

TIP

As you configure the find function, try to think of it as constructing a sentence that reads left-to-right. For example: "Find 1 occurrence Forward from the Display Center of a bus named My Bus 1, and on All bits a pattern that Equals XX Hex, display the event When all criteria is Present."

NOTE

The find qualifiers:

- **Present**>
- **Present**>=
- **Present**<
- **Present**<=
- **Present for Range**
- **Not Present for Range**

allow you to specify a time duration. This means that the find event specified in the expression area will be found based upon the given time and operator.

The other qualifiers:

- **Present**
- **Not Present**
- **Entering**
- **Exiting**
- **Transitioning**


do not allow a time duration.

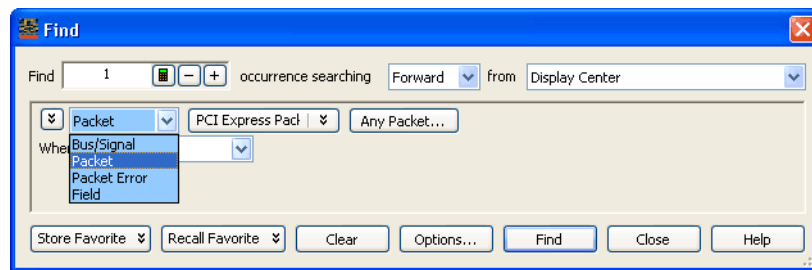
- See Also
- To quickly find bus/signal patterns (see [page 264](#))
 - To find packet patterns in the captured data (see [page 267](#))
 - To find complex patterns in the captured data (see [page 270](#))

- To store, recall, or delete favorite find patterns (see [page 271](#))
- To specify "found" marker placement (see [page 272](#))

To find packet patterns in the captured data

In the Protocol Viewer window, you can search for packets, packet errors, and field values.

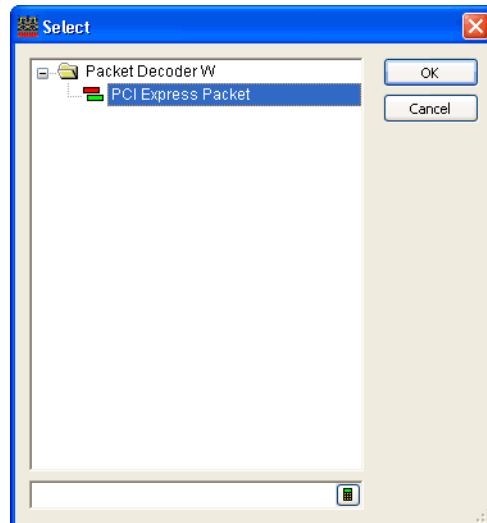
- 1 With the Protocol Viewer window open, choose **Edit>Find...** from the menu bar or click the  icon in the standard toolbar (see [page 398](#)).
- 2 In the **Find** dialog, enter the number of the occurrences you wish to find, select whether you want to search forward or backward from the start location; then, select the start location.
- 3 In the event specification area, select the **Packet**, **Packet Error**, or **Field** pattern type.



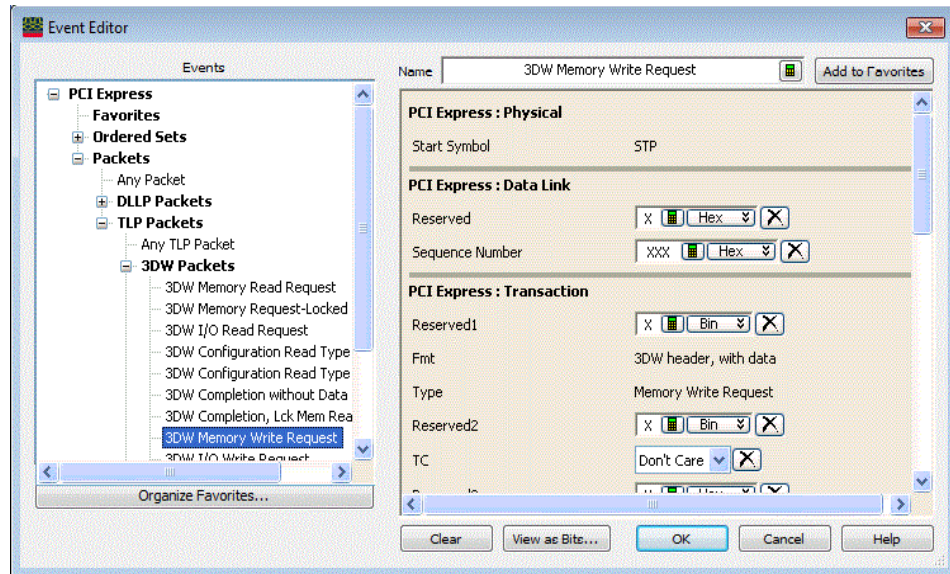
- To find packet events (see [page 267](#))
- To find packet errors (see [page 268](#))
- To find field values (see [page 269](#))

To find packet events

- 1 You can click packet type button to open a selection dialog.



- 2 Click the packet event button to open the Event Editor dialog.

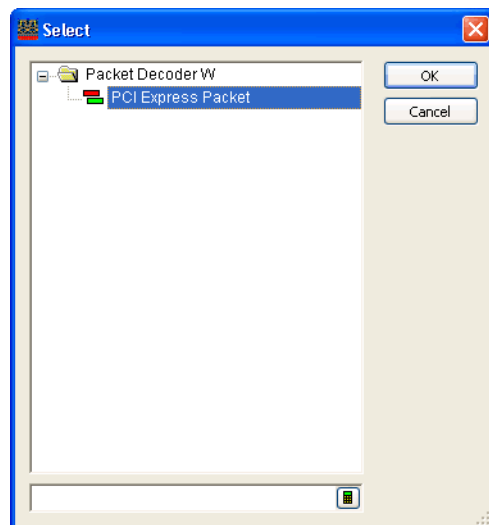


For more information, see Using the Packet Event Editor (see [page 164](#))

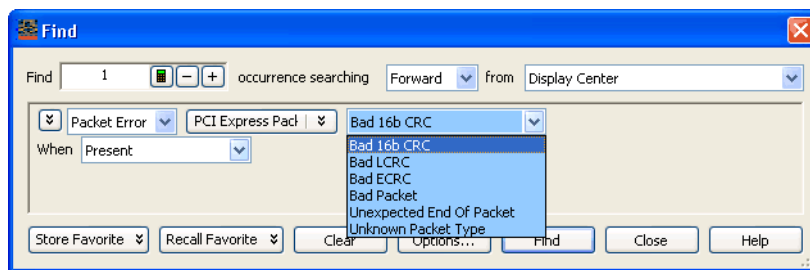
- 3 Select the desired **When** find qualifier (which further qualifies the find criteria with a time duration or other operator). For more information, see [To find bus/signal patterns in the captured data \(see page 265\)](#).
- 4 Click **Find**.
- 5 Click the Previous or Next icons to see more occurrences.

To find packet errors

- 1 You can click packet type button to open a selection dialog.



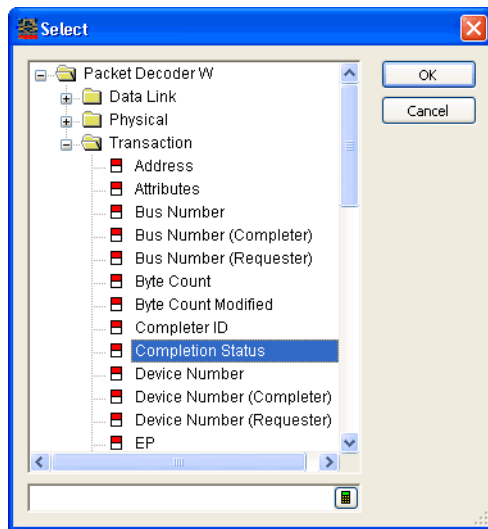
- 2 Select the packet error value:



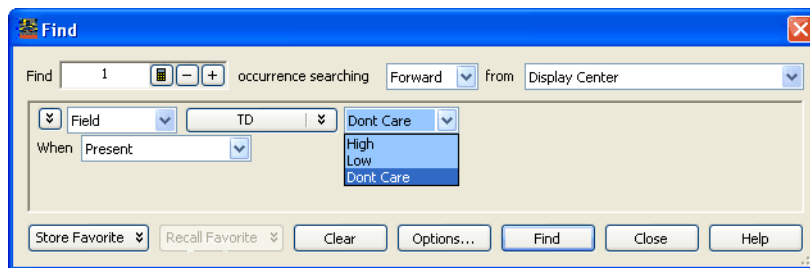
- 3 Select the desired **When** find qualifier (which further qualifies the find criteria with a time duration or other operator). For more information, see [To find bus/signal patterns in the captured data](#) (see [page 265](#)).
- 4 Click **Find**.
- 5 Click the Previous or Next icons to see more occurrences.

To find field values

- 1 Select the field name.
Clicking lets you select from recently used field names. Clicking elsewhere on a field name button opens a Select dialog for selecting a different name.

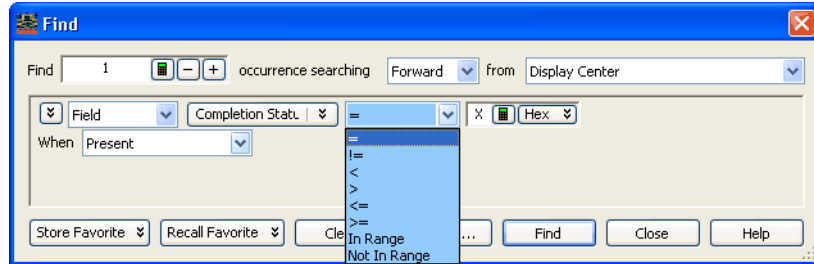


- 2 Specify the field value:
If a single-bit field has been selected, select the signal pattern value (**High**, **Low**, or **Dont Care**).

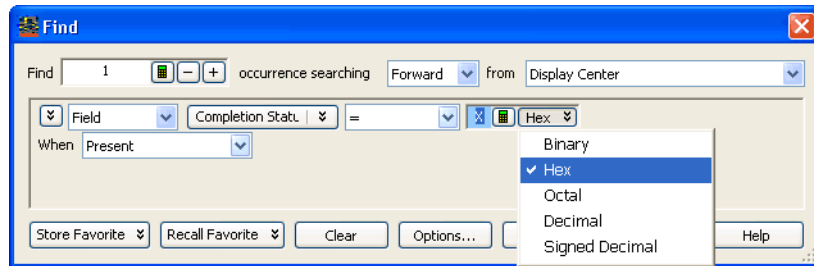




If a multiple-bit field has been selected:

- a Select one of the operators: = (equal to), != (not equal to), < (less than), > (greater than), <= (less than or equal to), >= (greater than or equal to), **In Range**, or **Not In Range**.



- b Select the number base (**Binary**, **Hex**, **Octal**, **Decimal**, or **Signed Decimal**, also known as two's complement).




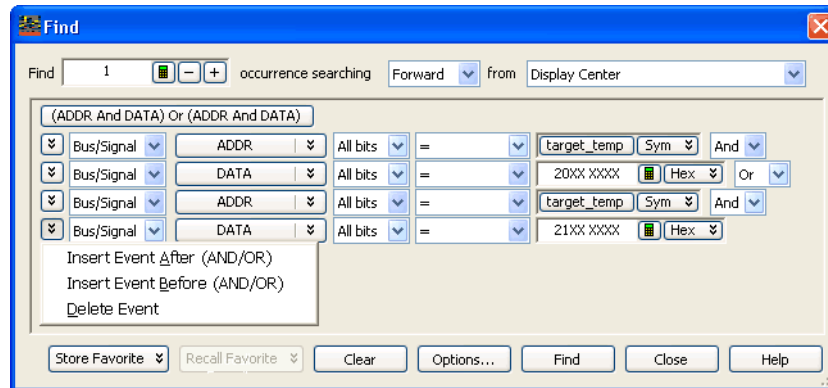
- c Enter the pattern value(s).
- 3 Select the desired **When** find qualifier (which further qualifies the find criteria with a time duration or other operator). For more information, see To find bus/signal patterns in the captured data (see [page 265](#)).
- 4 Click **Find**.
- 5 Click the Previous  or Next  icons to see more occurrences.

- See Also
- To quickly find bus/signal patterns (see [page 264](#))
 - To find bus/signal patterns in the captured data (see [page 265](#))
 - To find complex patterns in the captured data (see [page 270](#))
 - To store, recall, or delete favorite find patterns (see [page 271](#))
 - To specify "found" marker placement (see [page 272](#))

To find complex patterns in the captured data

You can expand search criteria to include more than one event describing data patterns.

- 1 From the menu bar, choose **Edit>Find...**, or click the  icon in the standard toolbar (see [page 398](#)).
- 2 In the **Find** dialog, select the number of the occurrences you wish to find, select whether you want to search forward or backward from the start location; then, select the start location.
- 3 Select the pattern event drop down menu to choose **Insert Event After (AND/OR)** or **Insert Event Before (AND/OR)** to insert new find events.



The **Delete Event** option will delete the current event only if there is more than one event present.


- 4 For each event you add, select either **And** or **Or** to specify how the event patterns are combined. AND'ed searches find occurrences of both events, while OR'ed searches find occurrences of either event.
When you have AND'ed and OR'ed events, a button appears above the events for changing the event evaluation order.
- 5 For each event, select the bus or signal name and enter the value you want to locate.
- 6 Click **Find**.

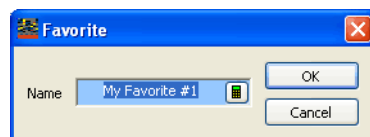
- See Also
- To quickly find bus signal patterns (see [page 264](#))
 - To find bus/signal patterns in the captured data (see [page 265](#))
 - To find packet patterns in the captured data (see [page 267](#))
 - To store, recall, or delete favorite find patterns (see [page 271](#))
 - To specify "found" marker placement (see [page 272](#))

To store, recall, or delete favorite find patterns

- To store a favorite pattern (see [page 271](#))
- To recall a favorite pattern (see [page 271](#))
- To delete a favorite pattern (see [page 272](#))


To store a favorite pattern

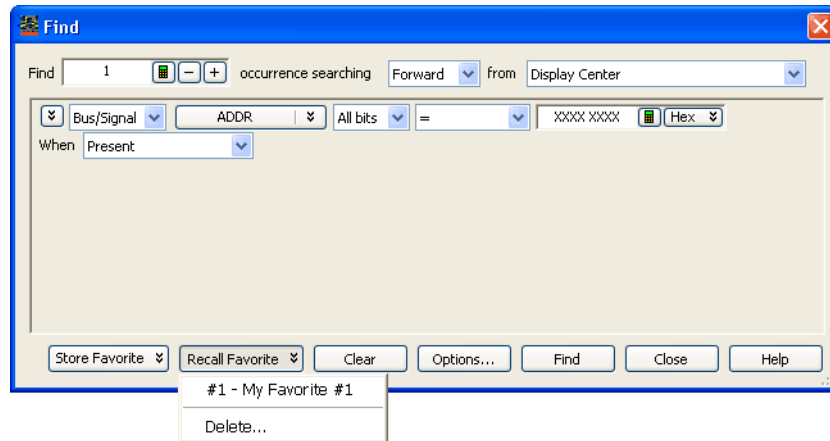
- 1 From the menu bar select, **Edit>Find...**, or click the  icon.
- 2 Set up the pattern you want to find (see [To find bus/signal patterns in the captured data \(see page 265\)](#) or [To find complex bus/signal patterns in the captured data \(see page 270\)](#)).
- 3 Click **Store Favorite**.



- 4 Enter the name of the find pattern.
- 5 Click **OK** to save the find pattern.


To recall a favorite pattern

- 1 From the menu bar select, **Edit>Find...**, or click the  icon.
- 2 Select **Recall Favorite**; then, select the find pattern you want to use from the drop down menu.



- 3 Click **Find**.

To delete a favorite pattern


- 1 From the menu bar select, **Edit>Find...**, or click the  icon.
- 2 Select **Recall Favorite**; then, select **Delete...**
- 3 In the Delete Favorites dialog, select the find patterns you wish to delete; then, click **Delete**.

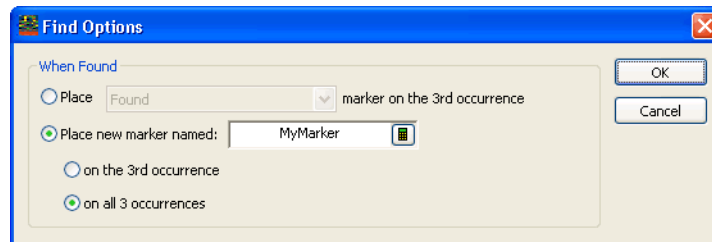
See Also

- Searching the Captured Data (see [page 264](#))
- To specify "found" marker placement (see [page 272](#))

To specify "found" marker placement

When searching for a pattern, you can place an existing marker on the last occurrence, or you can place a new marker on the last occurrence or on all occurrences.

- 1 From the menu bar select, **Edit>Find...**, or click the  icon in the standard toolbar (see [page 398](#)).
- 2 In the Find dialog, set up the pattern you want to find.
- 3 Click **Options...**



To place an existing marker on the last occurrence

- 1 Select the **Place** option.
- 2 Select the marker you want to place from the drop down menu.
- 3 Click **OK**.

To place a new marker

- 1 Select **Place new marker named:**.
- 2 Enter the name of the new marker.
- 3 Select whether you want to place the new marker **on the last occurrence** or **on all occurrences**. The **on all occurrences** option is only available when you are finding more than one occurrence.

- 4 Click **OK**.

- See Also
- Searching the Captured Data (see [page 264](#))
 - To store, recall, or delete favorite find patterns (see [page 271](#))
 - To place markers in data (see [page 247](#))

Comparing Captured Data to Reference Data

By comparing data from different acquisitions, you can look for differences between a known-good device under test and a device under test with a problem or one that is operating under different conditions.

To compare captured to reference data:

- 1 Capture (or load) the data you want to use as the reference data.
- 2 Select **Window>New Compare...** to open a new Compare display window.
- 3 In the Compare display window, click the **Copy...** button to select the current data that should be copied to the reference buffer.
- 4 Capture (or load) the data that you want to compare to the reference.
Differences are highlighted in the Compare window.

For more information on comparing captured data to reference data, see:

- To copy data to the reference buffer (see [page 274](#))
- To find differences in the compared data (see [page 274](#))
- To compare only a range of samples (see [page 275](#))
- To offset the reference data (see [page 275](#))
- To run until a number of compare differences (see [page 275](#))
- To set Compare window properties (see [page 276](#))

- See Also
- Compare Display Window (see [page 411](#))
 - Capturing Data from the Device Under Test (see [page 141](#))
 - Loading Saved Data and Setups (see [page 208](#))

To copy data to the reference buffer

- 1 In the Compare display window, click the **Copy...** button.
- 2 In the Select Buses/Signals dialog:
 - a From the available buses and signals, select the ones to be copied to the reference buffer and click **Add>>**.
To remove buses and signals from the selected list, select them and click **<<Remove**.
 - b Select either *All* data or a range of data using markers.
- 3 When you are ready to begin the copy, click **OK**.

NOTE

Copying generated bus/signal columns, such as those created by an inverse assembler or an analysis tool, takes longer because of the extra processing to re-create the data.

NOTE

If your logic analyzer has deep memory, it takes a while to copy data to the reference buffer.

To find differences in the compared data

In the Compare display window:

- Click the **>>** button to find the next difference (below the center reference).
- Click the **<<** button to find the previous difference (above the center reference).

- Click a blue tick mark in the **Compare Overview** bar (between the vertical scroll bar and the Marker Overview bar on the right side of the window) to go to that difference.

NOTE

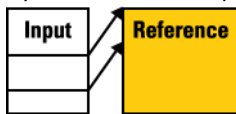
When a difference occurs on a subrow (for example, when the data is inverse-assembled or decoded by an analysis tool), the next and previous buttons go to the sample row instead of the subrow.

To compare only a range of samples

- 1 In the Compare display window, click the **Range & Offset...** button.
- 2 In the Range & Reference dialog, select either *All* data or a range of data using markers.
- 3 Click **OK**.

NOTE

When you specify a range to compare, the range is compared to the *top* of the reference buffer (unless the reference data has been offset (see [page 275](#)) by a number of samples) and not the same range in the reference buffer as you might expect. This behavior allows multiple ranges in the input data to be compared with the reference data.



To offset the reference data

When there are differences in the number of samples captured before the trigger, or when you are comparing a range of samples, you can offset the reference data so that the samples being compared are properly aligned.

- 1 In the Compare display window, click the **Range & Offset...** button.
- 2 In the Range & Reference Offset dialog, enter the number of samples to offset the reference by.
- 3 Click **OK**.

See Also • To compare only a range of samples (see [page 275](#))

To run until a number of compare differences

The Compare display lets you stop comparing, stop a repetitive run, or send e-mail after a run has more than a specified number of differences when compared to the reference data.

- 1 In the Compare display window, click **Compare Until...**
- 2 In the Difference Properties tab of the Compare Properties dialog, enter the number of differences that will stop comparing, stop a repetitive run, or send an e-mail message.
- 3 To stop a compare after the number of differences have been found, select the **Stop comparing** check box.
- 4 To stop a repetitive run after the number of differences have been found, select the **Stop repetitive run** check box.
- 5 To send e-mail after the number of differences have been found, select the **Send e-mail** check box; then, click the **E-mail...** button. In the E-mail dialog (see [page 432](#)), enter the address to which e-mail will be sent, the subject, and the text of the message.
- 6 Click **OK** in the Compare Properties dialog.

- 7 Start the repetitive run measurement.

See Also · [Running/Stopping Measurements \(see page 196\)](#)

To set Compare window properties

- 1 In the Compare display window, right-click on the bus/signal column name; then, select **Properties...** from the popup menu.
- 2 In the Compare Properties dialog:
 - The **Window Properties** tab lets you select the reference data background color, the background color that indicates no reference data, and the difference foreground and background colors.
 - The **Column Properties** tab's Display field lets you display **All** of the reference data, just the reference data where a difference was found (**Difference Pair**), or only the highlighted differences in the data being compared (**Input Only**).
 - The **Difference Properties** tab lets you select the options for running until a number of compare differences are found.

All other Compare property options are the same as in the Listing window.

See Also · [To set Listing window properties \(see page 238\)](#)
· [To run until a number of compare differences \(see page 275\)](#)

Viewing Source Code Associated with Captured Data

- 1 Add and configure the appropriate inverse assembler tool (see "Using Inverse Assembler Tools" (in the online help)).
- 2 Load line number symbols (see To load symbols from a file (see [page 134](#))).
- 3 Select **Window>New Source...** to open a new Source display window.
- 4 In the Add New Window after dialog, select the inverse assembler or filter/colorize tool that the Source window should be added after.
Generally, you want the Source window getting the same data as other display windows (Listing, Waveform, etc.).
- 5 In the source display pane of the Source window, right-click, and choose **Properties...**
- 6 In the Source Viewer Properties dialog's Source Code Directories tab, click **Add...** tab.
- 7 In the Browse for Folder dialog, select the directory that contains the source files, and click **OK**.
- 8 Click **OK** to close the Source Viewer Properties dialog.

For more information on viewing the source code associated with captured data, see:

- To step through captured data by source lines (see [page 277](#))
- To go to captured data associated with a source line (see [page 278](#))
- To browse source files (see [page 278](#))
- To search for text in source files (see [page 279](#))
- To set a Quick Trigger in the Source window (see [page 145](#))
- To go to the source line associated with the listing center rectangle (see [page 279](#))
- To edit the source code directory list (see [page 279](#))
- To select the correlation bus (see [page 280](#))
- Changing Source Window Properties (see [page 281](#))
 - To change the source background color (see [page 282](#))
 - To change the source text color (see [page 282](#))
 - To change the source font size (see [page 282](#))
 - To change the source tab width (see [page 282](#))
 - To show/hide source line numbers (see [page 282](#))
 - To change the "Set Quick Trigger" alignment (see [page 282](#))

- See Also
- Analyzing Listing Data (see [page 235](#))
 - Source Display Window (see [page 412](#))

To step through captured data by source lines

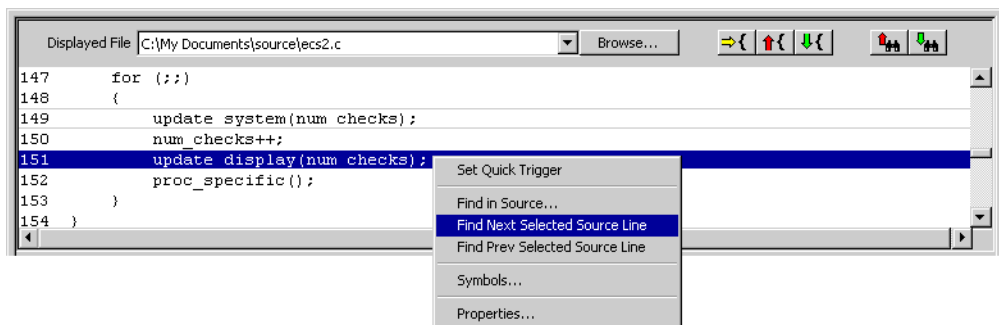


- 1 In the Source window's source pane, click the step to next source line or step to previous source line buttons.

The listing pane is updated to show the captured data associated with the next or previous source line, and the source pane is updated to show the next or previous source line.

See Also · Viewing Source Code Associated with Captured Data (see [page 277](#))

To go to captured data associated with a source line



- 1 In the Source window's source pane, click the source line whose associated captured data you want to view.
- 2 Click the find next selected source line or find previous selected source line buttons.
Or, right-click the selected source line and choose **Find Next Selected Source Line** or **Find Prev Selected Source Line**.

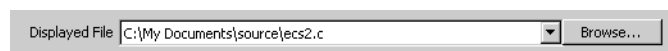
If captured data associated with the source line is found, the listing pane is updated to show the captured data, and the source pane is updated to show the selected source line.

If captured data associated with the source line is not found, an information dialog is displayed.

See Also · Viewing Source Code Associated with Captured Data (see [page 277](#))

To browse source files

- 1 In the Source window's source pane, click **Browse....**

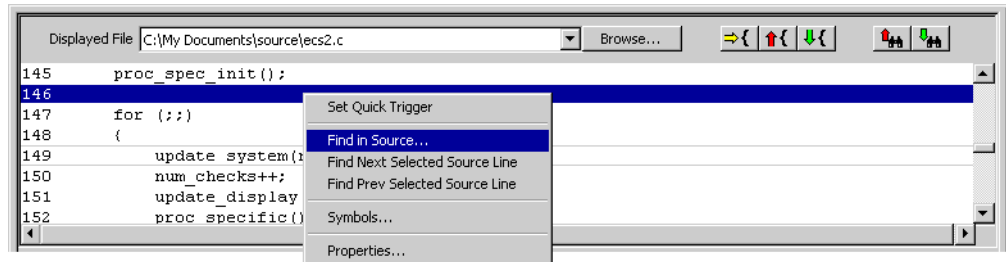


- 2 In the Select Source File to Open dialog, select the source file to browse, and click **Open**.
The selected source file appears in the source pane.

- See Also
- To search for text in source files (see [page 279](#))
 - To set a Quick Trigger in the Source window (see [page 145](#))
 - To go to the source line associated with the listing center rectangle (see [page 279](#))
 - Viewing Source Code Associated with Captured Data (see [page 277](#))

To search for text in source files

- 1 In the Source window's source pane, right-click choose **Find in Source...**




- 2 In the Find dialog, enter the text to search for, select the direction to search, and click **Find Next**.
If the text is found, the source line is highlighted.
If the text is not found, an information dialog is displayed.

- See Also
- To set a Quick Trigger in the Source window (see [page 145](#))
 - To go to the source line associated with the listing center rectangle (see [page 279](#))
 - Viewing Source Code Associated with Captured Data (see [page 277](#))

To go to the source line associated with the listing center rectangle

After browsing or searching for text in source files, you may want to return to displaying the source line associated with the captured data displayed in the listing pane.



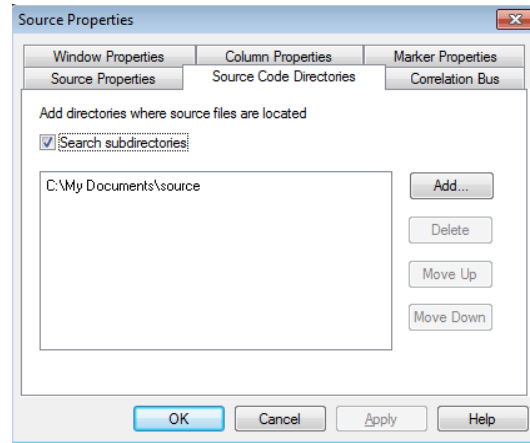
- 1 In the Source window's source pane, click the  show correlated source line button.
The source pane is updated to show either the source line associated with the listing center rectangle or "No matching symbol found."

- See Also
- Viewing Source Code Associated with Captured Data (see [page 277](#))

To edit the source code directory list

Because source file paths specified in the symbol file may not be valid if you compile on one computer and debug on another, you can specify the directories where source code is located.

- 1 In the Source window display areas, right-click, and choose **Properties...**
- 2 Or, choose **Edit>Window Properties...** from the main menu.
- 3 In the Source Viewer Properties dialog's Source Code Directories tab:



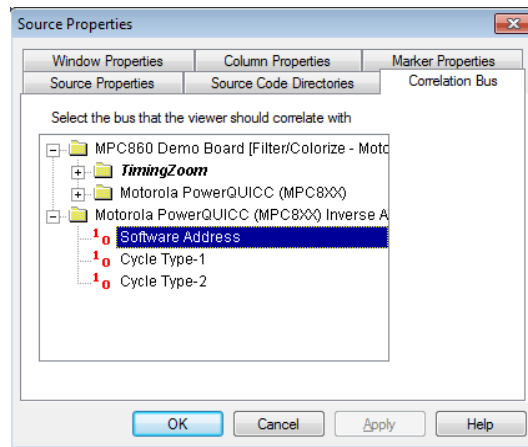
You can:

- Add a directory to the search list by clicking **Add...**
 - Specify whether subdirectories are included in or excluded from the search by checking or unchecking **Search subdirectories**.
 - Change a directory's order in the search list by highlighting a directory and clicking **Move Up** or **Move Down**.
 - Delete directory from the search list by highlighting a directory and clicking **Delete**.
- 4 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

See Also • Viewing Source Code Associated with Captured Data (see [page 277](#))

To select the correlation bus

- 1 In the Source window display areas, right-click, and choose **Properties...**
- 2 Or, choose **Edit>Window Properties...** from the main menu.
- 3 In the Source Viewer Properties dialog's Correlation Bus tab, select the bus on which the Source window should look for line number symbols.



Typically, you will select the "software address" bus generated by an inverse assembler tool or another address bus.

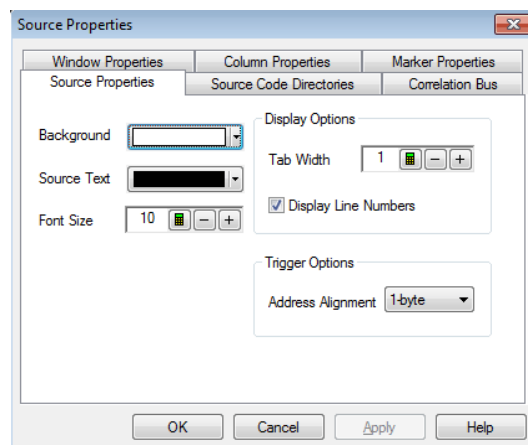
- 4 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

See Also • Viewing Source Code Associated with Captured Data (see [page 277](#))

Changing Source Window Properties

You can change properties that affect the source code pane of the Source display window.

- 1 In the source display pane of the Source window, right-click, and choose **Properties...**
- 2 Or, choose **Edit>Window Properties...** from the main menu.
- 3 In the Source Viewer Properties dialog's Source Properties tab:



You can:

- Change the source background color (see [page 282](#))
- Change the source text color (see [page 282](#))
- Change the source font size (see [page 282](#))
- Change the source tab width (see [page 282](#))
- Show/hide source line numbers (see [page 282](#))
- Change the "Set Quick Trigger" alignment (see [page 282](#))

- 4 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

- See Also
- Changing Listing Window Properties (see [page 238](#))
 - Changing Bus/Signal Column Properties (see [page 240](#))

To change the source background color

- 1 In the Source Viewer Properties dialog's Source Properties tab, click the **Background** color selection button and select the desired background color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

To change the source text color

- 1 In the Source Viewer Properties dialog's Source Properties tab, click the **Source Text** selection button and select the desired color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

To change the source font size

- 1 In the Source Viewer Properties dialog's Source Properties tab, enter the desired **Font Size**.
Fonts can range from size 6 through 72 points.
- 2 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

To change the source tab width

- 1 In the Source Viewer Properties dialog's Source Properties tab, enter the desired **Tab Width**.
Tab widths can range from 1 to 10 spaces.
- 2 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

To show/hide source line numbers

- 1 In the Source Viewer Properties dialog's Source Properties tab, check or uncheck **Display Line Numbers** to specify whether source file line numbers are shown or hidden.
- 2 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

To change the "Set Quick Trigger" alignment

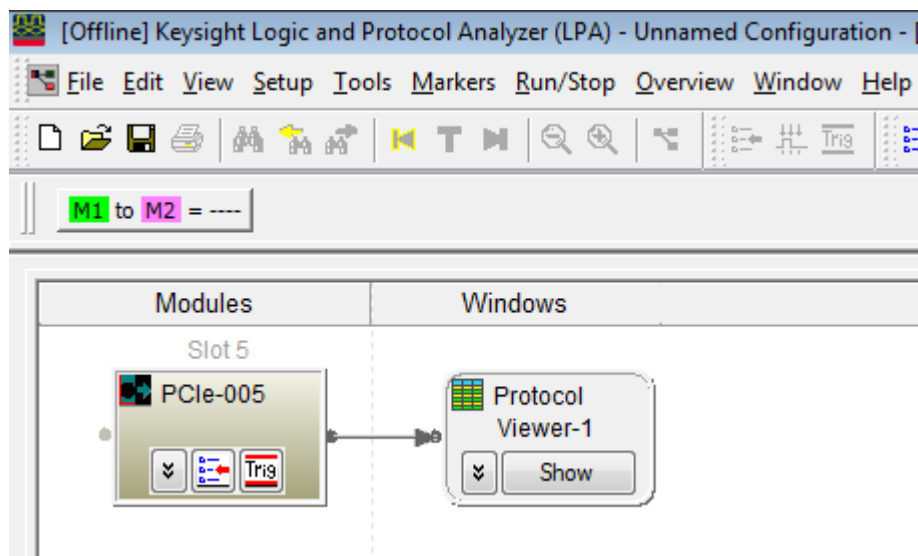
For microprocessors that fetch blocks of instructions at a time (from block boundary addresses only), the address alignment property lets you adjust the source line symbol values to be on block boundary addresses when setting up Quick Triggers on a source line.

- 1 In the Source Viewer Properties dialog's Source Properties tab, select the desired **Address Alignment**.
- 2 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

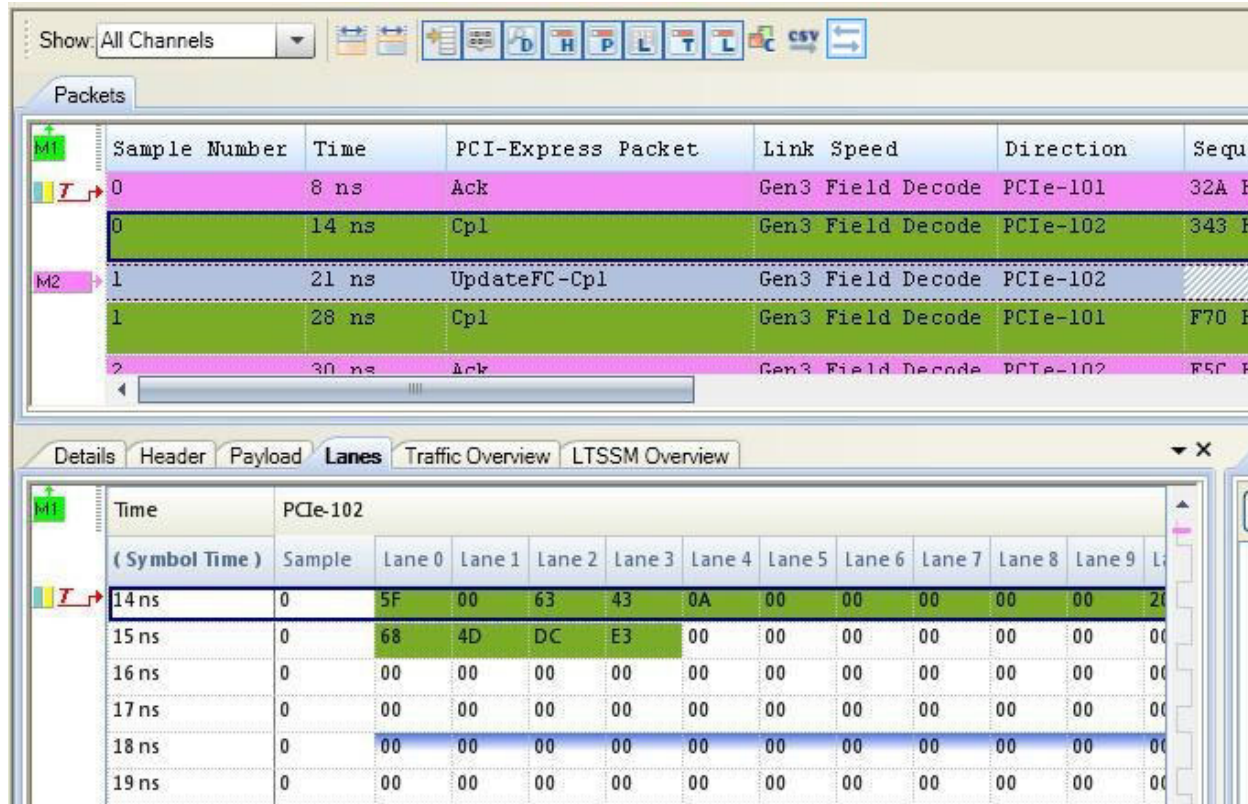
- See Also
- To set a Quick Trigger in the Source window (see [page 145](#))

Analyzing Packet Data

You can use a Protocol Viewer window to display the captured data. The following screen displays an instance of Protocol Viewer added to the U4301A PCIe Gen3 Analyzer module to display the PCIe data captured by this module.



Unlike the Listing window, the Protocol Viewer windows let you view summarized and detailed packet information at the same time within two panes. The following screen displays the captured packets in a Protocol Viewer window.



The upper packet summary pane is similar to a Listing window except that its columns display packets and fields instead of bus/signal values. Like a Listing window, you can insert time or pattern markers.

The lower pane contains tabs for viewing selected packet details, header, payload, and lane information.

The Protocol Viewer window is customized for the protocol family for which the data is displayed.

You can add new Protocol Viewer windows using the Window menu. You can view Protocol Viewer windows that have already been added by choosing from the open window names at the bottom of the menu. If tabbed windows (see [page 313](#)) are turned on, you can also view Protocol Viewer windows by selecting the tab at the bottom of the window.

- Viewing the Packet Summary (see [page 285](#))
 - To go to different locations in the displayed data (see [page 286](#))
 - To re-arrange packet columns (see [page 287](#))
 - To insert or delete packet columns (see [page 288](#))
 - To show/hide parts of the packet summary display (see [page 292](#))
 - To lockstep Protocol Viewer with other display windows (see [page 293](#))
- Viewing a Selected Packet (see [page 293](#))
 - To view and compare packet details (see [page 294](#))
 - To view a packet header (see [page 297](#))
 - To view a packet payload (see [page 298](#))
 - To view a packet's lanes (see [page 299](#))
 - To show/hide Protocol Viewer panes (see [page 301](#))

- Changing Packet Summary Event Colors (see [page 301](#))
- Changing Protocol Viewer Window Properties (see [page 302](#))
 - To change the selected row box color (see [page 303](#))
 - To change the Protocol Viewer font size (see [page 303](#))
 - To lock scrolling with other display windows (see [page 303](#))
- Changing Packet Summary Column Properties (see [page 304](#))
 - To change the directions of a packet column (see [page 305](#))
 - To change the width of a packet column (see [page 305](#))
 - To change the alignment of a packet column (see [page 305](#))
 - To change a packet column's number base (see [page 306](#))
 - To select the marker for marker-relative times (see [page 306](#))
 - To select fixed time units (see [page 306](#))
- Viewing LTSSM States and Transitions (in the *U4301 PCIe Gen3 Analyzer Online help*).
- Viewing Decoded Transactions (in the *U4301 PCIe Gen3 Analyzer Online help*)
- Viewing Offline Performance Summary (in the *U4301 PCIe Gen3 Analyzer Online help*)
- Viewing PCIe Gen3 Packets (in the *U4301 PCIe Gen3 Analyzer Online help*)
- Customizing Protocol Descriptions for Protocol Viewer (in the online help)

- See Also
- To specify packet events (in "Find a packet" trigger function) (see [page 164](#))
 - To find packet patterns in the captured data (see [page 267](#))
 - "To specify packet patterns to filter" (in the online help)
 - Marking, and Measuring Between, Data Points (see [page 245](#))
 - Displaying Names (Symbols) for Packet Summary Column Values (see [page 243](#))
 - Searching the Captured Data (see [page 264](#))

Viewing the Packet Summary

The upper packet summary pane of the Protocol Viewer window is similar to a Listing window except that its columns display packets and fields instead of bus/signal values. Like a Listing window, you can insert time or pattern markers.

Sample Number	Time	PCI-Express Packet	Link Speed	Direction	Sequence Number
0	14 ns	Cp1	Gen3 Field Decode	PCIe-102	343 Hex
1	21 ns	UpdateFC-Cp1	Gen3 Field Decode	PCIe-102	
1	28 ns	Cp1	Gen3 Field Decode	PCIe-101	F70 Hex

Upper pane of the Protocol Viewer

Click a line to select a packet. Notice that a colored box highlights the selected line. You can use the up-arrow or down-arrow keys to select the previous or next packets.

- ["To go to different locations in the displayed data"](#) on page 286

- [“To re-arrange packet columns”](#) on page 287
- [“To insert or delete packet columns”](#) on page 288
- [“To show/hide parts of the packet summary display”](#) on page 292
- To lockstep Protocol Viewer with other display windows ([page 293](#))

See Also

- [“Changing Packet Summary Event Colors”](#) on page 301
- [“Changing Protocol Viewer Window Properties”](#) on page 302
- [“Changing Packet Summary Column Properties”](#) on page 304
- [“Viewing a Selected Packet”](#) on page 293
- [“Marking, and Measuring Between, Data Points”](#) on page 245
- [“Searching the Captured Data”](#) on page 264
- Viewing LTSSM States and Transitions (in the *U4301 PCIe Gen3 Analyzer Online help*).
- Viewing PCIe Gen3 Packets (in the *U4301 PCIe Gen3 Analyzer Online help*)




To go to different locations in the displayed data

In the Protocol Viewer window, you can go to different locations in the captured data by using the vertical scroll bars, by using the Go To buttons on the standard toolbar, or by choosing Go To commands from popup menus.

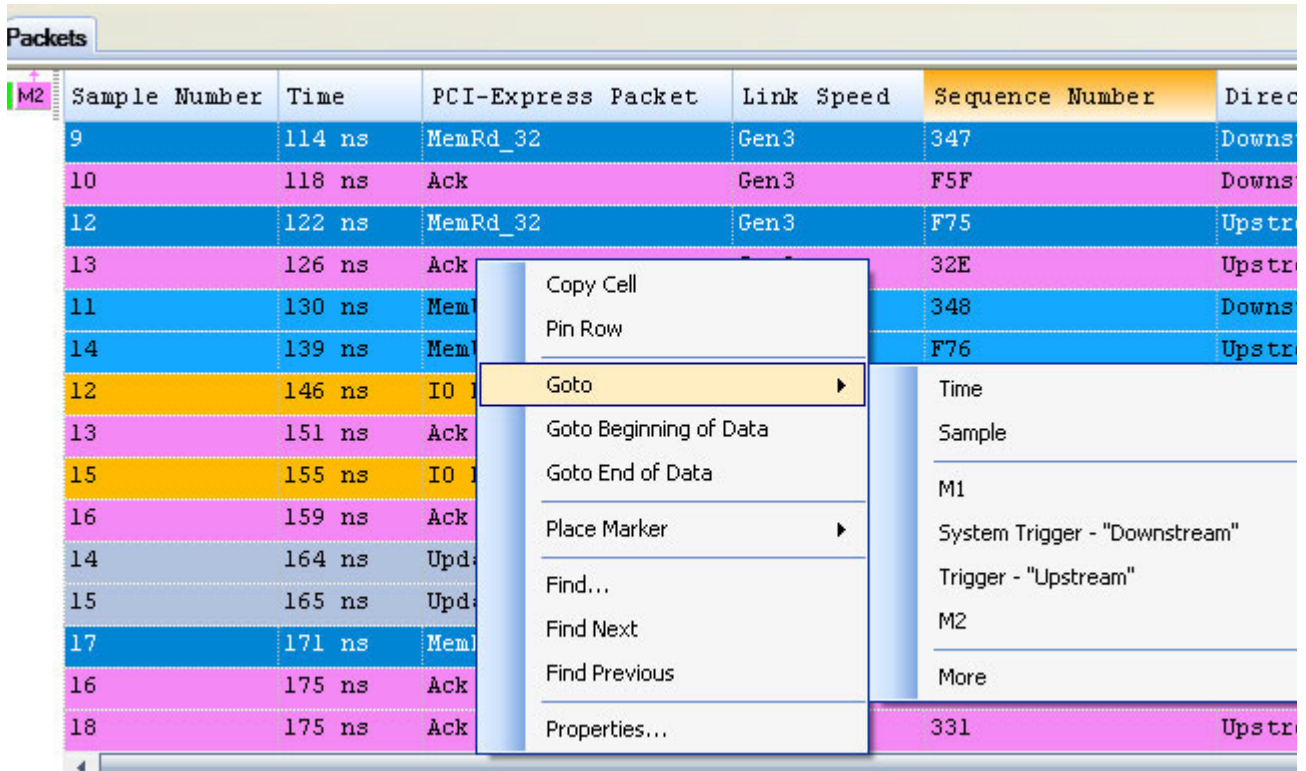
To go to different locations using toolbar buttons

- 1 Click one of the Go To buttons in the standard toolbar.

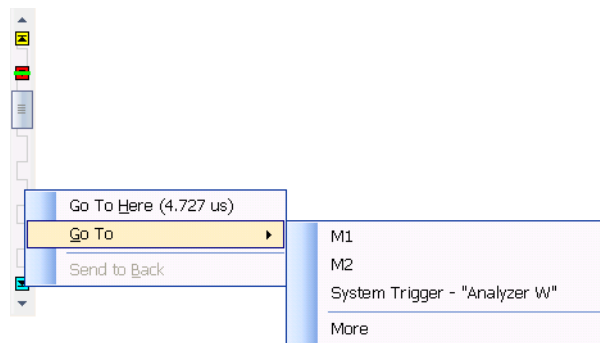


Menu	Description
	Go to Beginning – centers the beginning of the displayed data.
	Go to Trigger – centers the trigger point in the displayed data.
	Go to End – centers the end of the displayed data.

- To go to different locations using popup/context menus
- 1 Right-click in the packet summary display area, and choose one of the **Go To** commands.



You can choose **Beginning Of Data**, **End Of Data**, **Trigger**, a marker, a **Time**, or a **Sample**. Or, click in the marker overview bar, and choose one of the **Go To** commands.

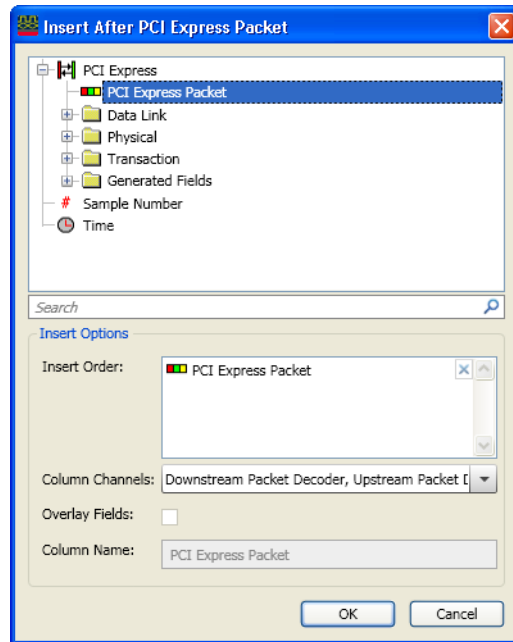


To re-arrange packet columns

- 1 Position the mouse pointer over the packet field column you want to move.
- 2 Click and hold the mouse button.
- 3 Drag-and-drop the packet decode column to its new position.
The column is placed to the left of the red position indicator that appears.

To insert or delete packet columns

- To insert packet columns
- 1 In the Protocol Viewer window, right-click in the column headings; then, choose **Insert Column...**, **Insert Column Before...**, or **Insert Column After...**
 - 2 In the Insert dialog, select the packet fields/column(s) you want to insert.



- 3 In the **Insert Options** area of the Insert dialog, you can:
 - Specify the **Insert Order** by dragging items in the list to the desired order. (Clicking the "X" in the list removes the item.)
 - Specify the **Column Channels** (directions) associated. For a more in-depth discussion, see: ["Understanding Column Channels \(Directions\)"](#) on page 288.
 - Choose to **Overlay Fields**; that is, you can choose to include multiple fields in one column.
 - When fields are overlaid, you can specify the **Column Name**.
- 4 Click **OK**.

- To delete selected packet columns
- 1 Highlight the headings of the columns you want to delete (by clicking, Shift-clicking, or Ctrl-clicking the column names).
 - 2 Right-click in (one of) the selected column heading(s); then, choose **Delete > Column(s)**.

Understanding Column Channels (Directions)

When a column is inserted into a Protocol Viewer, the Insert column dialog lets you optionally select the channels whose data is shown in the column:

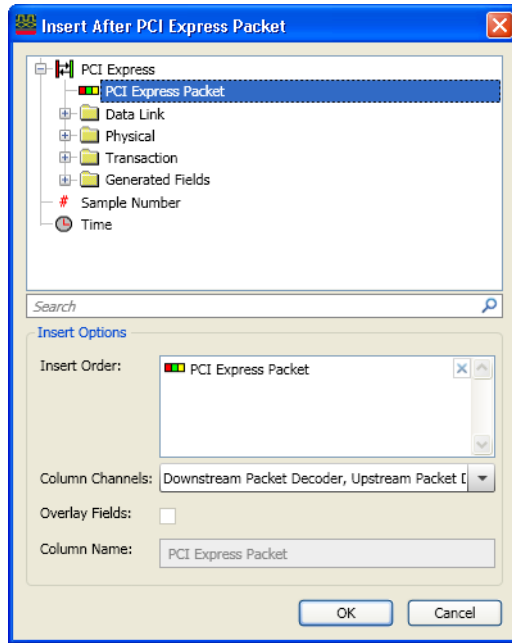


Figure 6 The Insert column dialog from the Protocol Viewer context menu

By default, columns are inserted with all channels selected. This means the new column will display the data for any of the available channels attached to the Protocol Viewer.

Channels are associated with directions which are simply another name for unidirectional links and represent the directional flow of protocol data along a particular network within the target platform's topology.

For example, say we have a bidirectional communication path between two PCI Express components: a root complex and a device as seen below:

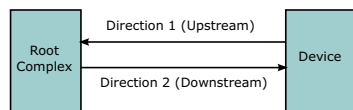


Figure 7 Two directions of PCI Express traffic

Assume that we are probing both directions of PCIe traffic by using a single PCIe Gen3 Analyzer module. In this case, we have two directions of PCI Express packet traffic being generated and then displayed using Protocol Viewer.

When inserting a new column within the Protocol Viewer window, you can specify for which of the channels the data in the inserted column should appear:

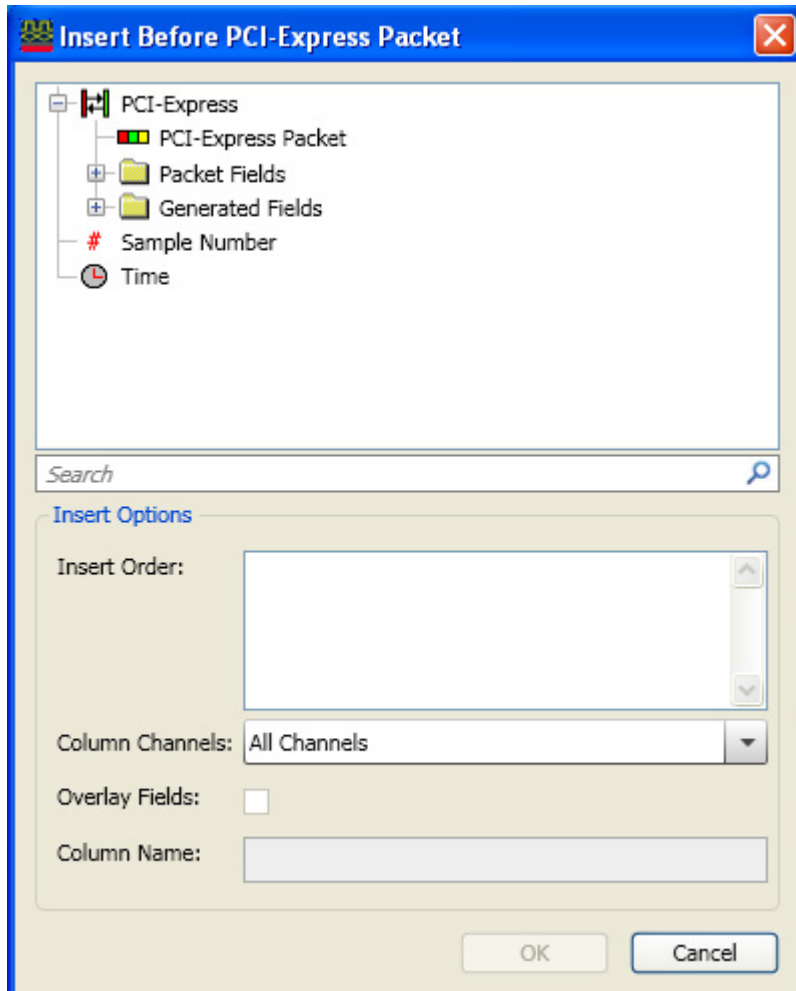


Figure 8 Insert a PCI Express Packet column for both directions

For example, if you specify that a new PCI Express Packet column should show data for All Channels (directions), it looks like:

Sample Number	Time	PCI-Express Packet	Direction
10	112 ns	UpdateFC-NP (Gen3)	Upstream
11	113 ns	UpdateFC-Cpl (Gen3)	Upstream
9	114 ns	Memory Read 32b (Gen3)	Downstream
10	118 ns	Ack (Gen3)	Downstream
12	122 ns	Memory Read 32b (Gen3)	Upstream
13	126 ns	Ack (Gen3)	Upstream
11	130 ns	Memory Write 32b (Gen3)	Downstream
14	139 ns	Memory Write 32b (Gen3)	Upstream
12	146 ns	I/O Read (Gen3)	Downstream
13	151 ns	Ack (Gen3)	Downstream
15	155 ns	I/O Read (Gen3)	Upstream
16	159 ns	Ack (Gen3)	Upstream

Figure 9 A PCI Express Packet column that displays data for both the Upstream and Downstream directions

Alternatively, you can specify that a column be dedicated to certain channels (directions). Dedicating a column to certain channels (directions) can be useful when trying to visualize the flow of packets across a topology. For example, viewing the packets flow across both directions can be accomplished by creating two PCI Express Packet columns, each dedicated to a unique direction like:

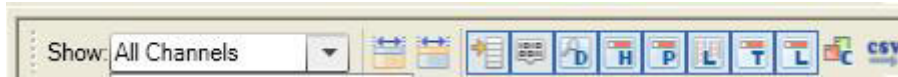
Sampl	Time	PCI-Express Packet	PCI-Express Packet	Direction
10	112 ns		UpdateFC-NP	Upstream
11	113 ns		UpdateFC-Cpl	Upstream
9	114 ns	MemRd_32		Downstream
10	118 ns	Ack		Downstream
12	122 ns		MemRd_32	Upstream
13	126 ns		Ack	Upstream
11	130 ns	MemWr_32		Downstream
14	139 ns		MemWr_32	Upstream
12	146 ns	I/O Rd		Downstream
13	151 ns	Ack		Downstream
15	155 ns		I/O Rd	Upstream
16	159 ns		Ack	Upstream

Figure 10 Two PCI Express Packet columns that are each dedicated to a unique direction

Note that you can also use the **Show:Channels** button drop-down to specify the channels (directions) that should be displayed in the Packet summary.

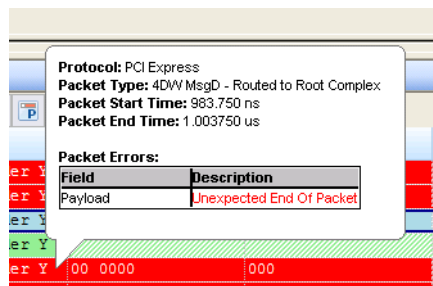
To show/hide parts of the packet summary display

- 1 Click one of the **Show:** buttons in the Packet Summary toolbar.









The Packet Summary toolbar contains the following Show buttons:

Menu	Description
	Displays the column channels (directions) and allows you to select the required directions for which data will be displayed in Protocol Viewer. The channels you select here are displayed in the Direction column of Summary pane. For more information, see "Understanding Column Channels (Directions)" on page 288.
	Resize columns based on column content.
	Resize columns based on column headers.
	Toggles marker visibility. Markers appear in the far left column.
	Toggles tool tip visibility. Tool tips are the information pop-ups that appear when the mouse cursor is held still over a packet row for a couple seconds:



The Packet Summary toolbar also contains the following Show buttons to show/hide Protocol Viewer panes (see [page 301](#)).

Menu	Description
	Toggles Header pane visibility from the Details pane.
	Toggles Payload pane visibility from the Details pane.
	Toggles Details pane visibility from the Details pane.
	Toggles Lanes pane visibility from the Details pane.
	Toggles Traffic Overview pane visibility from the Details pane.
	Toggles LTSSM Overview pane visibility from the Details pane.


You can also make these selections in the **Pane Display Options** area of the Protocol Viewer Properties dialog box.

See Also • [“Changing Protocol Viewer Window Properties”](#) on page 302

To lockstep Protocol Viewer with other display windows

You can use the *Lockstep windows* feature to ensure that when a Protocol Viewer window is scrolled, other lockstepped windows are scrolled as well, such that the same time is centered in each lockstepped display window. This allows you to easily map and view correlated data in multiple display windows.

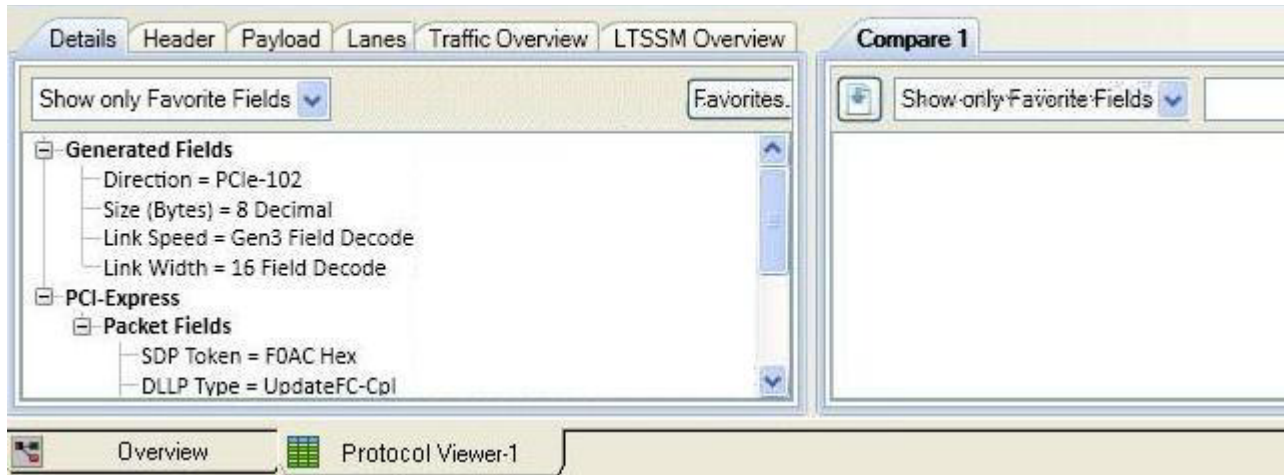
You can lockstep a Protocol Viewer window with display windows such as another Protocol Viewer, Waveform, Listing, or Compare.

- 1 From the Protocol Viewer toolbar, click the  **Lockstep windows** toolbar button.
- 2 In the **Lockstep Windows** dialog, select the display window(s) whose scrolling should be locked with this window and specify any offset from this window.
- 3 Click **OK** to close the Lockstep Windows dialog.

NOTE: In Waveform Viewer and Listing, you can access the Lockstep Windows dialog by clicking **Lockstep Windows...** from the **Window Properties** tab of the display window's **Properties** dialog,

Viewing a Selected Packet

When a packet is selected in the upper packet summary portion of the Protocol Viewer window (by clicking on a line or by using the up arrow or down arrow keys to highlight the previous or next line), information about the selected packet appears in the lower part of the window.



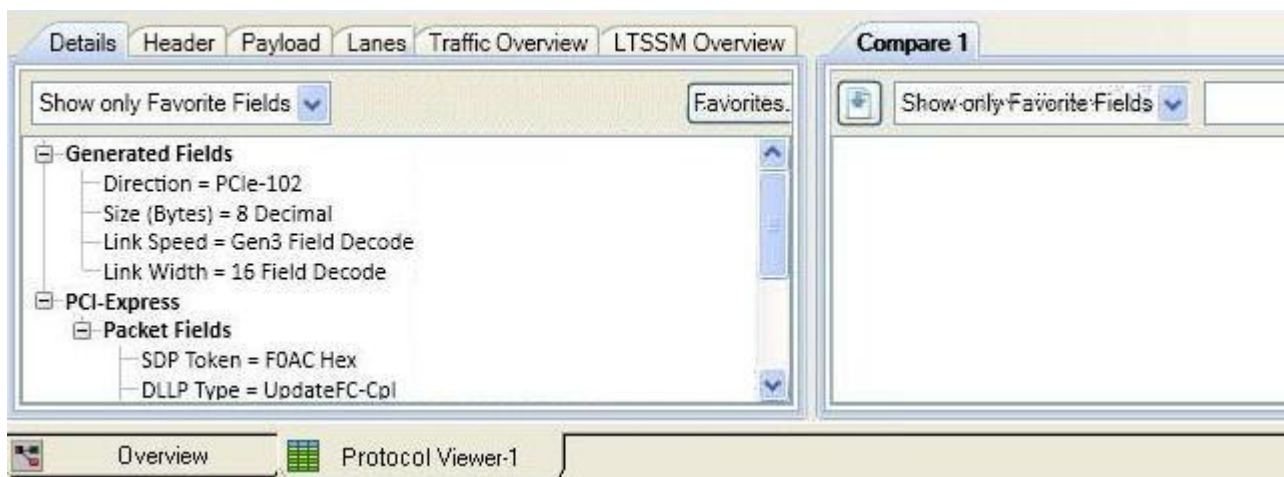
Lower pane of the Protocol Viewer

- ["To view and compare packet details"](#) on page 294
- ["To view a packet header"](#) on page 297
- ["To view a packet payload"](#) on page 298
- ["To view a packet's lanes"](#) on page 299
- ["To view LTSSM States and Transitions"](#) on page 300
- ["To show/hide Protocol Viewer panes"](#) on page 301

See Also • ["Viewing the Packet Summary"](#) on page 285

To view and compare packet details

- To view packet details
- 1 Select a packet in the upper packet summary portion of the Protocol Viewer window (by clicking on a line or by using the up arrow or down arrow keys to highlight the previous or next line).
 - 2 Select the **Details** tab in the lower portion of the window.



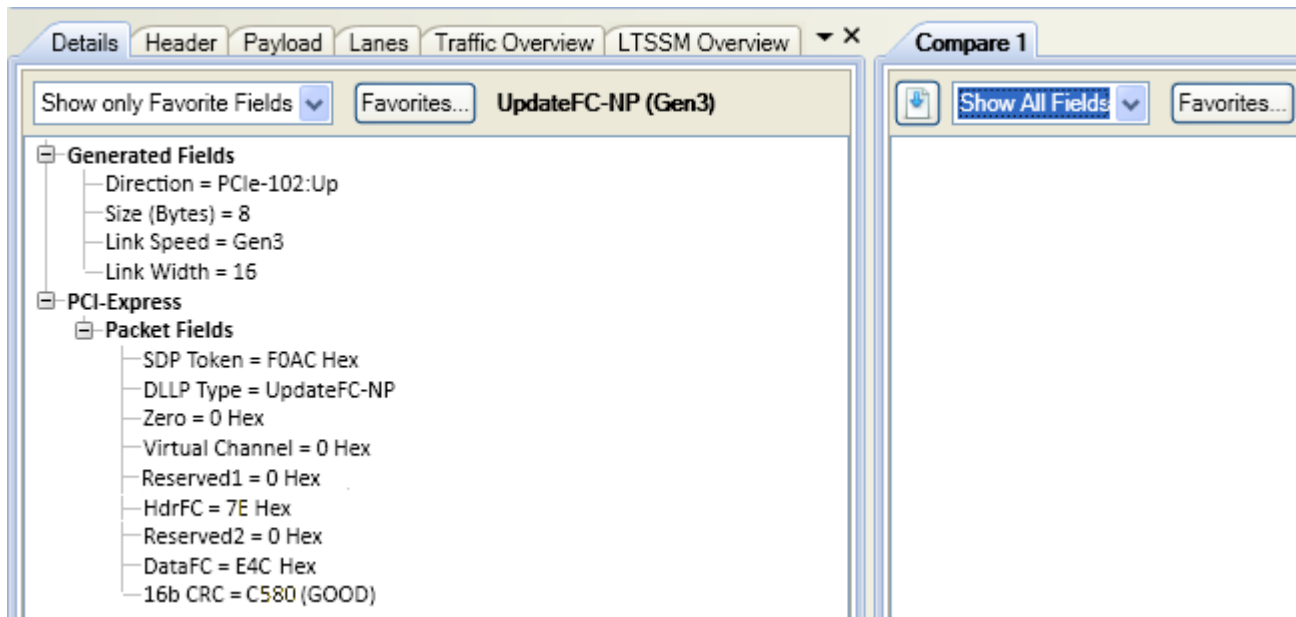
Packet details are displayed in the left pane of the lower portion of the window.


If you hold the mouse pointer motionless for a second over one of the packets, a *tool tip* (that is, a small box with text) appears with more information.

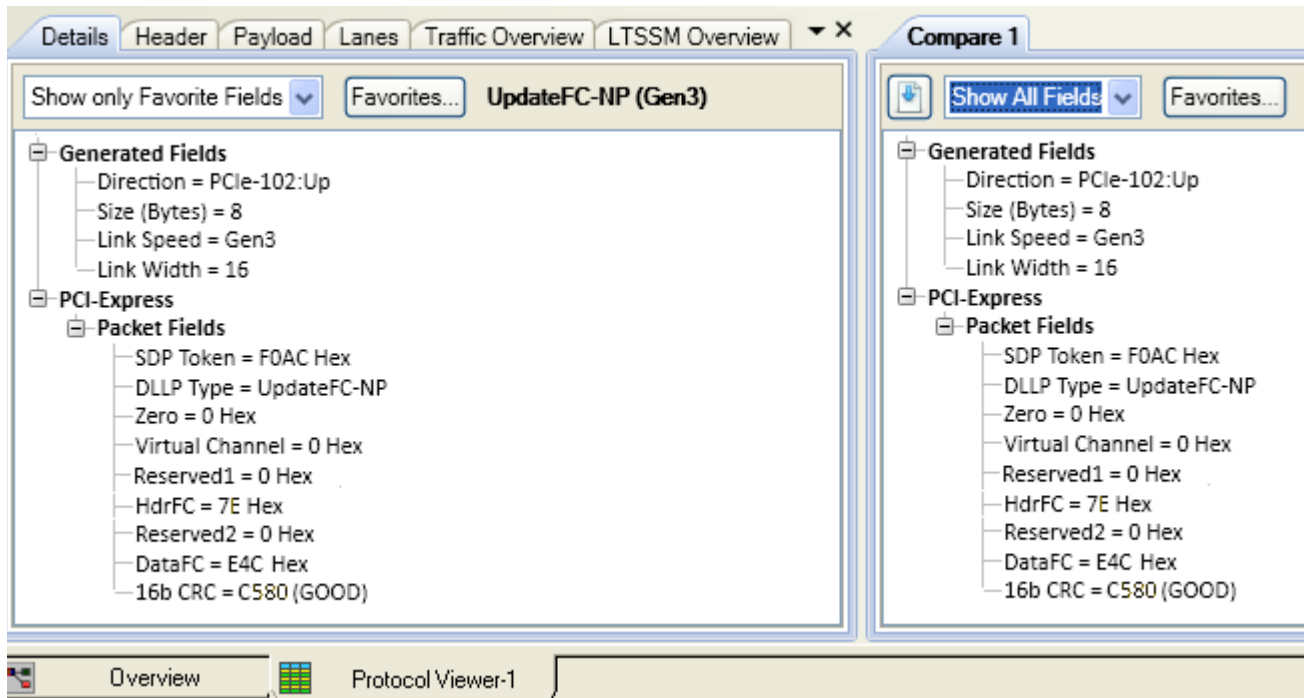
- 3 You can expand or collapse the displayed information by clicking "+" or "-" in the packet hierarchy tree.

To compare packet details

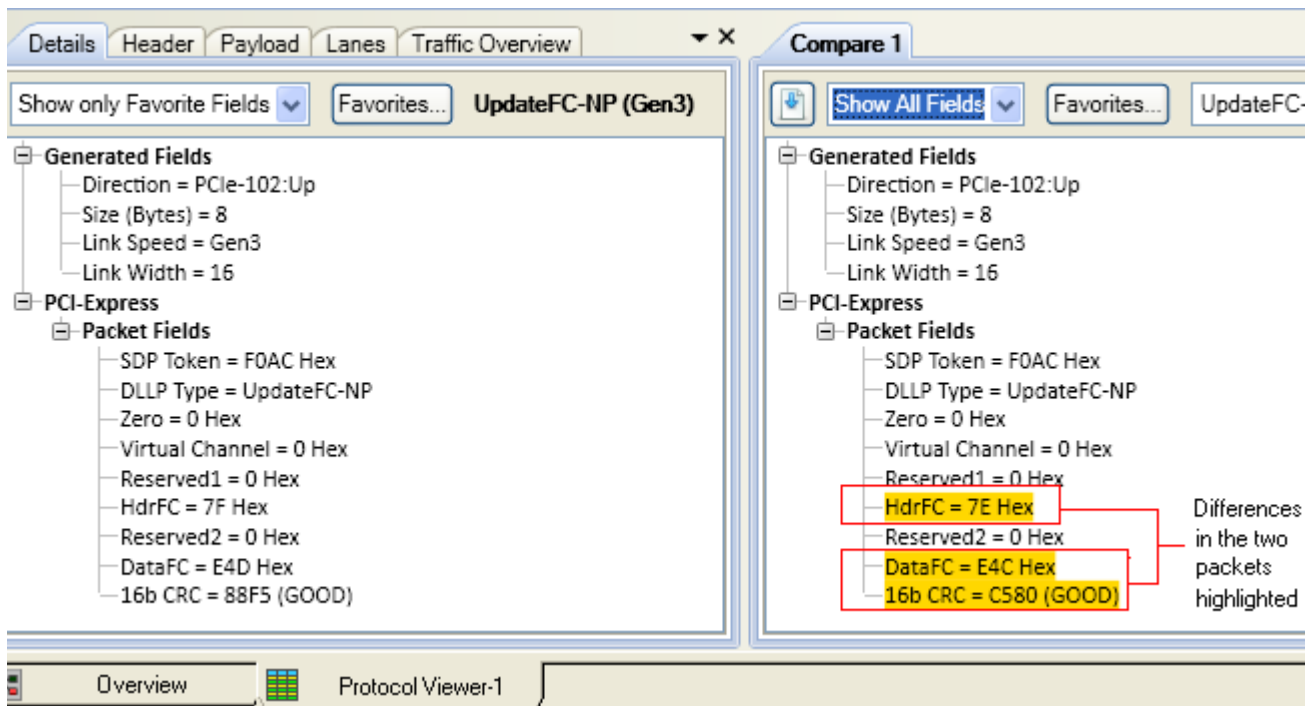
- 1 In the lower pane of the window, display the details of the packet that you want to compare. To do this, select the required packet from the list of packets in the upper pane. The details of the packet are now displayed in the lower-left pane. The Compare buffer in the lower-right pane is currently empty.



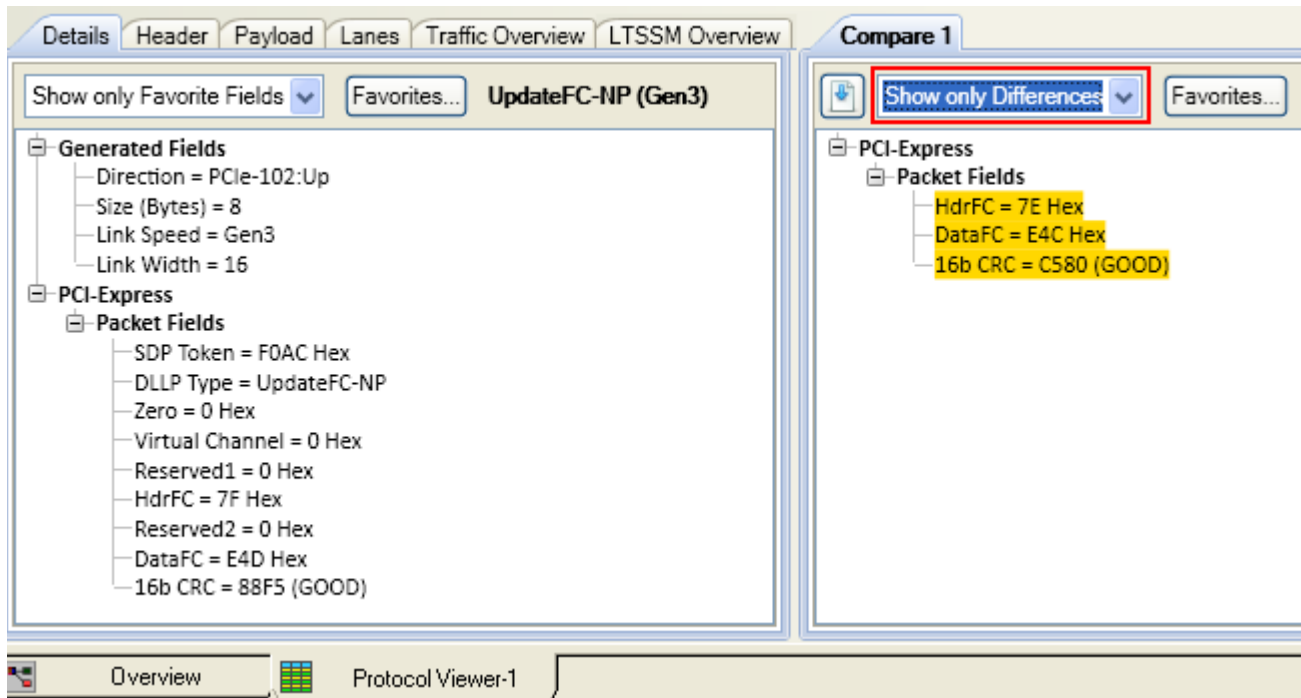
- 2 Click the  button displayed in the Compare buffer in the lower-right pane. Clicking this button copies the details of the packet currently displayed in the lower-left pane to the compare buffer at the right.



- From the list of the packets displayed in the upper pane, select the other packet with which you want to compare the packet already displayed in the compare buffer. The details of the selected packet are now displayed in the lower-left pane and the differences between the two packets are highlighted in the compare buffer in the lower-right pane.



- 4 You can customize the display of packet information in the compare buffer as per the following options available in the compare buffer.
 - If you want to display only the differences between the two packets, then select the **Show only Differences** option from the listbox in the compare buffer.



- If you want to display only a few selected fields in the comparison, then click the **Favorites** button in the compare buffer, uncheck the fields that you do not want to display, and click **OK**. Finally, select the **Show only Favorite Fields** option from the listbox in the compare buffer. Doing so, displays only those fields in the lower pane that you selected for display. If you want to display the differences in only the favorite fields of the two packets, then select the **Show only Favorite Differences** option from the listbox in the compare buffer.
- If desired, you can change the name for the packet information displayed in the compare buffer. A text field next to the Favorites button in the compare buffer displays the packet type. You can alter this text while comparing packets.

- See Also
- ["To view a packet header"](#) on page 297
 - ["To view a packet payload"](#) on page 298
 - ["To view a packet's lanes"](#) on page 299
 - ["To view LTSSM States and Transitions"](#) on page 300

To view a packet header

- 1 Select a packet in the upper packet summary portion of the Protocol Viewer window (by clicking on a line or by using the up arrow or down arrow keys to highlight the previous or next line).
- 2 Select the **Header** tab in the lower portion of the window.

	+0				+1				+2				+3											
	7	6	5	4	3	2	1	0	7	6	5	4	3	2	1	0	7	6	5	4	3	2	1	0
Byte 0	SDP Token								DLLP Type		VC		R		HdrFC[7:2]									
	0xF0AC								0xA		0		0x0		0x0		0x19							
Byte 4	H[1:0]		R		DataFC								16b CRC											
	0x1		0x0		0x400								0xEB24											

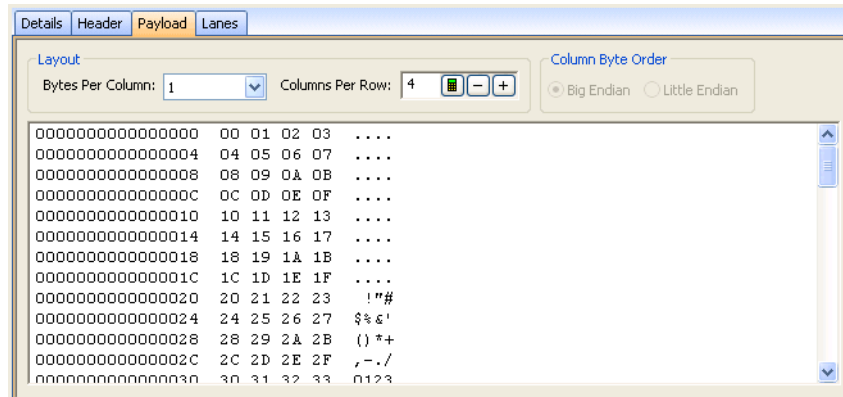
If you hold the mouse pointer motionless for a second over one of the header fields, a *tool tip* (that is, a small box with text) appears with more information about the field.

- 3 If desired, you can select a different number **Base**.

- See Also
- ["To view and compare packet details"](#) on page 294
 - ["To view a packet payload"](#) on page 298
 - ["To view a packet's lanes"](#) on page 299
 - ["To view LTSSM States and Transitions"](#) on page 300

To view a packet payload

- 1 Select a packet in the upper packet summary portion of the Protocol Viewer window (by clicking on a line or by using the up arrow or down arrow keys to highlight the previous or next line).
- 2 Select the **Payload** tab in the lower portion of the window.



Payload data bytes are displayed in the selected number of bytes per column and columns per row. The right-most column displays the row data in ASCII format.

- 3 You can format the display of payload bytes by selecting the **Bytes Per Column**, **Columns Per Row**, and **Column Byte Order** (when there is more than one byte per column).

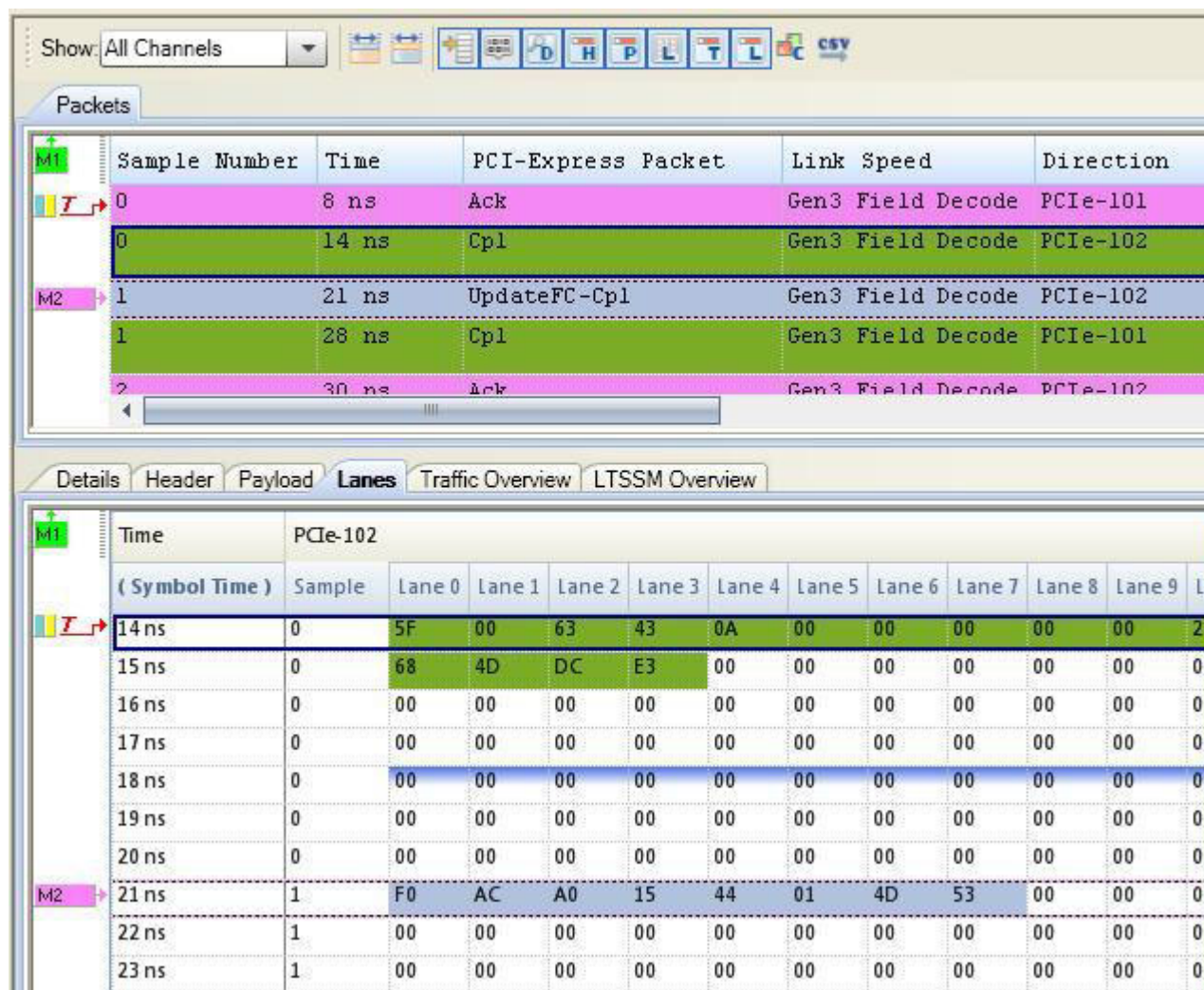
- See Also
- To view and compare packet details (see [page 294](#))
 - To view a packet header (see [page 297](#))
 - To view a packet's lanes (see [page 299](#))
 - ["To view LTSSM States and Transitions"](#) on page 300

To view a packet's lanes

- 1 Select a packet in the upper packet summary portion of the Protocol Viewer window (by clicking on a line or by using the up arrow or down arrow keys to highlight the previous or next line).
- 2 Select the **Lanes** tab in the lower portion of the window.

Lanes view in Protocol Viewer

The Lanes view in Protocol Exerciser shows vertical listing of packet data with respect to the logical lanes. In this view, you can see not just the packet data for the packet selected in the Summary pane but also the post packet data represented by different color codes. This helps you identify the start and end of data. The packet data in the lanes view is shown by the same color as used for the packet in the summary pane.



Context-menu of the Lanes view

If you right-click anywhere in the Lanes view, a context-menu is displayed.

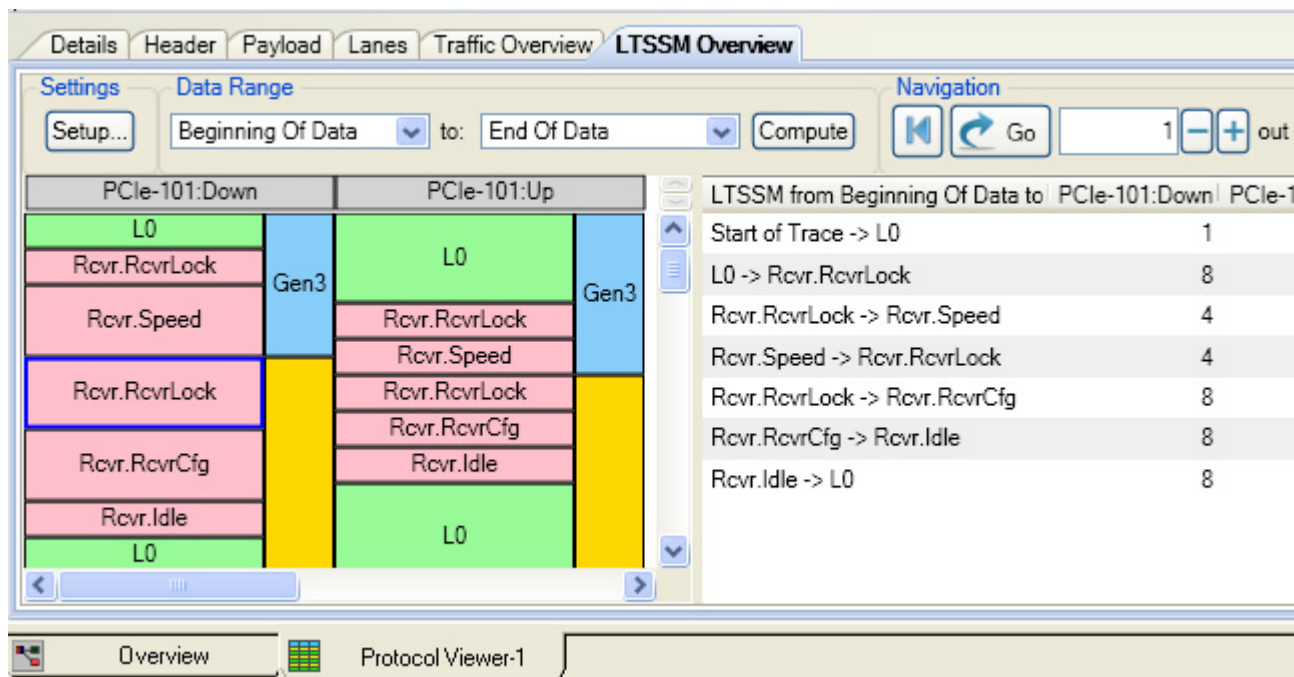
Time	PCIe-101												
(Symbol Time)	Sample Number	Lane 0	Lane 1	Lane 2	Lane 3	Lane 4	Lane 5	Lane 6	Lane 7	Lane 8	Lane 9	Lane 10	Lane 11
21 ns	0	00	00	00	00	00	00	00	00	00	00	00	00
22 ns	0	00	00	00	00	00	00	00	00	00	00	00	00
23 ns	0	00	00	00	00	00	00	00	00	00	00	00	00
24 ns	0	00	00										00
25 ns	0	00	00										00
26 ns	0	00	00										00
27 ns	0	00	00										00
28 ns	1	5F	00										00
29 ns	1	26	3F	AC	41	00	00	00	00	00	00	00	00
30 ns	1	00	00	00	00	00	00	00	00	00	00	00	00

You can use this menu to customize the display of data in the Lanes view. For instance, if you disable the Lockstep option, then the upper summary pane and the lower Lanes view of Protocol viewer are not synchronized. Clicking on a packet in the summary view does not show its details in the Lane view and vice versa. You can also choose to color the packet data in the Lanes view with the color of the packet in the Summary pane or not to color the packets data.

- See Also
- To view and compare packet details (see [page 294](#))
 - To view a packet header (see [page 297](#))
 - To view a packet payload (see [page 298](#))

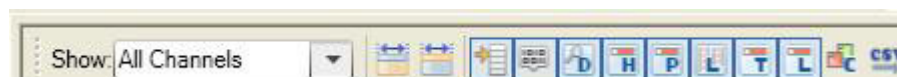
To view LTSSM States and Transitions

You can use the *LTSSM Overview* tab to display the LTSSM states and their transitions as detected from the PCIe data captured in a trace. To know more about how to view these states, refer to the topic **Viewing LTSSM States and Transitions** in the U4301 PCIe Gen3 Analyzer online help.



To show/hide Protocol Viewer panes

- 1 Click the toolbar buttons in the Packet Summary toolbar to toggle the visibility of various panes.



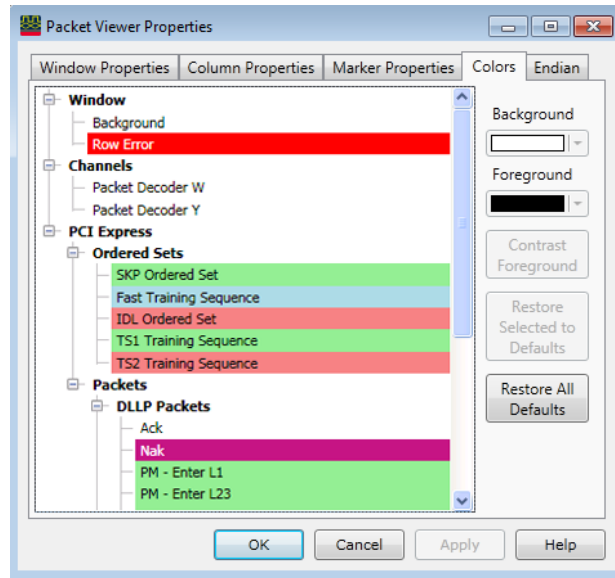
Alternatively, make these selections in the Pane Display Options area of the Window Properties tab in the Packet Viewer Properties dialog box.

See Also • [Changing Protocol Viewer Window Properties \(see page 302\)](#)

Changing Packet Summary Event Colors

To change the colors associated with events in the Protocol Viewer window:

- 1 Right-click in the packet summary portion of the window, and choose **Properties...**
- 2 In the Protocol Viewer Properties dialog, select the **Colors** tab.



- 3 In the Colors tab:
 - a Select the packet event type whose color you want to change.
 - b Select the **Background** color.
 - c Select the **Foreground** color or click **Contrast Foreground** to automatically get a color with good contrast to the selected background color.
- 4 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

To restore event color defaults

- 1 In the Protocol Viewer Properties dialog's Colors tab, click **Restore All Defaults** or **Restore Selected Defaults**.
- 2 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

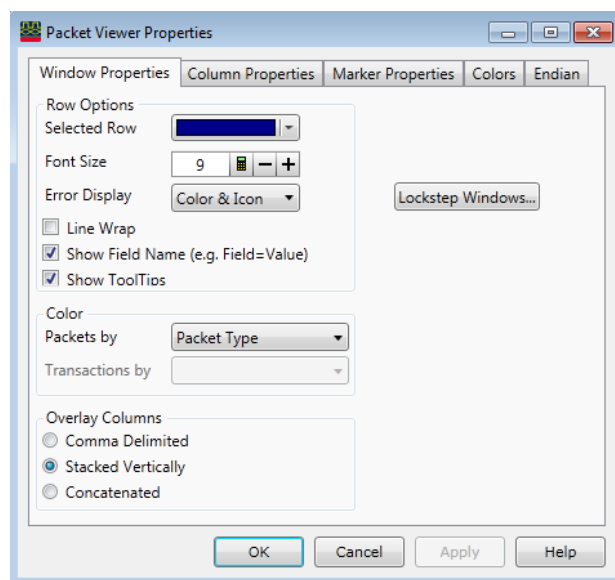
See Also

- [“Changing Protocol Viewer Window Properties”](#) on page 302
- [“Changing Packet Summary Column Properties”](#) on page 304

Changing Protocol Viewer Window Properties

You can change properties that affect the entire Protocol Viewer display window.

- 1 Right-click in the packet summary portion of the window, and choose **Properties...**
- 2 In the Protocol Viewer Properties dialog, select the **Window Properties** tab.



- 3 In the Window Properties tab, you can:
 - Change the selected row box color (see [page 303](#))
 - Change the Protocol Viewer font size (see [page 303](#))
 - Lock scrolling with other display windows (see [page 303](#))
 - Show/hide parts of the packet summary display (see [page 292](#))
 - Show/hide Protocol Viewer panes (see [page 301](#))
- 4 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

See Also

- Changing Packet Summary Event Colors (see [page 301](#))
- Changing Packet Summary Column Properties (see [page 304](#))

To change the selected row box color

To highlight the selected line in the upper packet summary area of the Protocol Viewer window, a box is drawn around it.

- 1 In the Window Properties tab of the Protocol Viewer Properties dialog, click the **Selected Row** color selection button and select the desired highlight box color from the palette.
If you want to use a color that is not on the palette, click **Other...** to access the custom color dialog.
- 2 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

To change the Protocol Viewer font size

The font size property adjusts the data display and packet decode column heading text size.

- 1 In the Window Properties tab of the Protocol Viewer Properties dialog, enter the desired **Font Size**.
Fonts can range from size 6 through 72 points.
- 2 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

As the font size is changed, the column width may be automatically increased to fit the new text size.

To lock scrolling with other display windows

You can lock display windows (for example, Waveform, Listing, Compare, etc.) so that when one window is scrolled, others are scrolled as well, such that the same time is centered in each display.

- 1 In the Window Properties tab of the Protocol Viewer Properties dialog, click **Lockstep Windows...**
- 2 In the Lockstep Windows dialog, select the display windows whose scrolling should be locked with this window and specify any offset from this window.



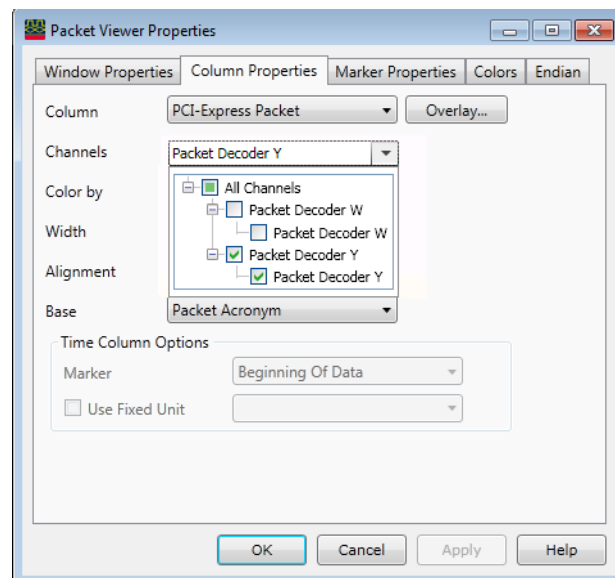
- 3 Click **OK** to close the Lockstep Windows dialog.
- 4 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

Changing Packet Summary Column Properties

In the Protocol Viewer display window, you can change the color, width, alignment, or number base of bus/signal data columns.

To change the properties of a bus/signal data column in the Protocol Viewer window:

- 1 Right-click on a packet decode column, and choose **Properties...**
Or, highlight the packet decode columns whose properties you want to change (by clicking, Shift-clicking, or Ctrl-clicking the column headings), and choose **Edit>Window Properties...** from the main menu.
- 2 In the Protocol Viewer Properties dialog's Column Properties tab:



You can:

- Select the **Column** to which the property changes apply. You can select:
 - Any packet decode column that is being displayed.
 - **<all>** packet decode columns.

- **Overlay...** to include multiple fields in a column.
 - [“To change the channels \(directions\) of a packet column”](#) on page 305
 - [“To change the width of a packet column”](#) on page 305
 - [“To change the alignment of a packet column”](#) on page 305
 - [“To change a packet column's number base”](#) on page 306
 - [“To select the marker for marker-relative times”](#) on page 306
 - [“To select fixed time units”](#) on page 306
- 3 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

- See Also
- [“Changing Packet Summary Event Colors”](#) on page 301
 - [“Changing Protocol Viewer Window Properties”](#) on page 302

To change the channels (directions) of a packet column

- 1 In the Protocol Viewer Properties dialog's Column Properties tab, check or uncheck the Channels **All Channels** checkbox.
If you want the column to display data for specified channels (directions) only, uncheck **All** and check the desired directions in the drop-down list.
If you want the column to display data for all channels (directions), check **All Channels**.
For more information, see [“Understanding Column Channels \(Directions\)”](#) on page 288.
- 2 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

To change the width of a packet column

TIP

You can autosize individual columns by placing the mouse pointer over the right border of the column header box; then, when the pointer icon changes to a resizing pointer, double-click.

TIP

If your keyboard has a numeric keypad, you can autosize all columns by selecting any column header box (to highlight it) and by pressing **Ctrl** and **+** on the numeric keypad.

- 1 In the packet headings row of the Protocol Viewer window, position the mouse pointer over a column separator line; when the cursor changes to a resizing cursor, drag the column border.

Or:

- 1 In the Protocol Viewer Properties dialog's Column Properties tab, enter the **Width** value in pixels.
The minimum column width is 1 pixel, while the maximum width is 1000 pixels.
- 2 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

To change the alignment of a packet column

The Alignment property sets the display of data to be left-justified, right-justified, or centered within the column.

- 1 In the Protocol Viewer Properties dialog's Column Properties tab, select the **Alignment** from:
 - **Left**
 - **Center**
 - **Right**
- 2 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

To change a packet column's number base

The base property specifies the number base to use when displaying the decoded packet values.

- 1 In the Column Properties tab of the Protocol Viewer Properties dialog, select the desired number **Base** from:

- **Binary**
- **Hex**
- **Octal**
- **Decimal**
- **Signed Decimal** (two's complement, the only choice for the "Sample Number" column)
- **Ascii**
- **Symbol** (see Displaying Names (Symbols) for Bus/Signal Values (see [page 243](#)))
- **Hardware Address**
- **Dot Notation**
- **Field Decode**

For the main packet decode information column, you can select from:

- **Packet Summary**
- **Packet Bytes**

For other generated packet columns, the only choice may be:

- **String**

If the "Time" column has been selected instead of a data column, your choices change from a numeric format to:

- **Absolute**
- **Relative Previous**
- **Relative Marker**

- 2 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

To select the marker for marker-relative times

In the Protocol Viewer window, you can display times relative to a marker.

- 1 In the Column Properties tab of the Protocol Viewer Properties dialog, use the **Bus/Signal** selection to select the **Time** column.
- 2 For the **Base** property, select **Relative Marker**.
- 3 For the **Marker** property, select the marker to which relative times should be displayed.
- 4 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

To select fixed time units

In the Protocol Viewer window, you can display time column values with a fixed unit.

- 1 In the Column Properties tab of the Protocol Viewer Properties dialog, use the **Bus/Signal** selection to select the **Time** column.
- 2 In the Time Column Properties box, check **Use Fixed Unit**; then, select the desired time unit from the drop-down list.
- 3 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

Analyzing the Same Data in Different Ways (Using the Overview Window)

The Overview window lets you specify how the data is sent from the logic analyzer data acquisition module to post-processing tools and display windows. For example, you can display the same data filtered in one Listing window and unfiltered in another Listing window.

To analyze the same data in different ways:

- 1 Open or display the Overview window.
- 2 Add new windows.
If the Add New Window After dialog appears, select the module or tool that the new window should be placed after.
- 3 Add new tools.
If the New Tool dialog appears, select where the new tool should be placed.

For more information on using the Overview window, see:

- To open or display the Overview window (see [page 307](#))
- To add, duplicate, or delete windows and tools (see [page 307](#))
- To edit window or tool properties (see [page 310](#))
- To rename windows, tools, and modules (see [page 311](#))
- To redraw the Overview window (see [page 311](#))
- To delete the Overview window (see [page 311](#))

- See Also
- Overview Window (see [page 418](#))
 - Waveform Display Window (see [page 404](#))
 - Listing Display Window (see [page 409](#))
 - Compare Display Window (see [page 411](#))
 - Source Display Window (see [page 412](#))
 - "Filter/Colorize Tool" (in the online help)
 - "Inverse Assembly Tools" (in the online help)
 - "Bus Analysis Tools" (in the online help)
 - "Tools" (in the online help)

To open or display the Overview window

- Select **Tools>Overview**.
- Select **Window>Overview**.
- If the Overview window is already open and you have Tabbed Windows (see [page 313](#)) turned on, you can display the Overview window by selecting the Overview tab at the bottom of the window.

- See Also
- Analyzing the Same Data in Different Ways (Using the Overview Window) (see [page 307](#))

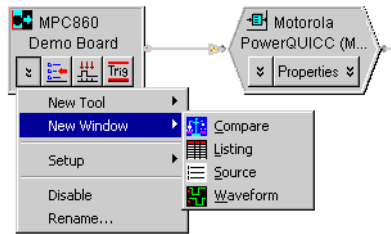
To add, duplicate, or delete windows and tools

You can add new listing and waveform display windows to the interface. As new windows are added, they appear in the list under **Window** in the menu bar. The active window will have a check mark. All available windows can be accessed either through the menu bar or through the use of tabs.

When you add a new tool to the logic analyzer's measurement configuration, its name appears at the bottom of the Tools menu. The tools interact with each other, so that you can progressively filter data or color parts of an inverse-assembled listing.

To add new windows

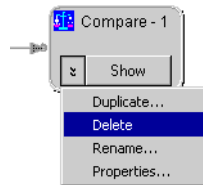
- From the menu bar, select **Window>New type...**
- If the windows are tabbed, you can also right-click on the tab and select **Window>New type...**
- In the Overview window, right-click in the background, and select **New Window** from the popup menu.
- In the Overview window, select **New Window** from a module or tool menu.



The new window is placed after the module or tool.

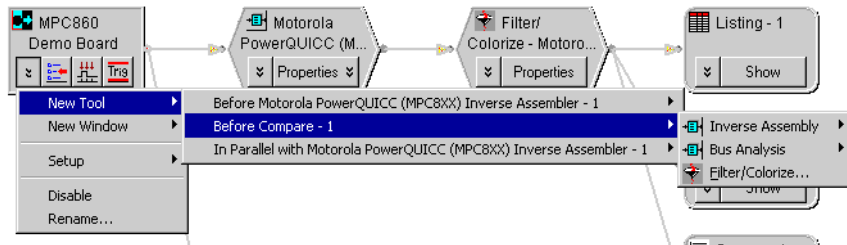
To delete windows

- From the window's menu in the menu bar, select **Delete**.
- If the windows are tabbed, you can right-click on the tab and select **Delete**.
- In the Overview window, select **Delete** from the window's menu.



To add new tools

- From the menu bar, select **Tools>New type...**
- In the Overview window, right-click in the background, and select **New Tool** from the popup menu.
- In the Overview window, select **New Tool** from a module or tool menu.



The new tool is placed after the module or tool.

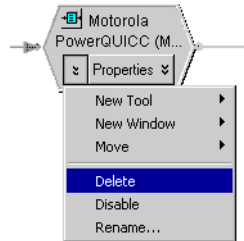
If the tool you want is not listed, make sure that you have "installed" (in the online help) and licensed (see [page 323](#)) the tool.

TIP

Many tools come with a configuration file. Loading the configuration file will add the tool, as well as set up the bus names, symbols, or filters used by the tool.

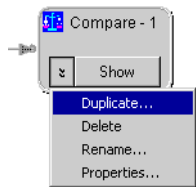
To delete tools

- In the Overview window, select **Delete** from the tool's menu.



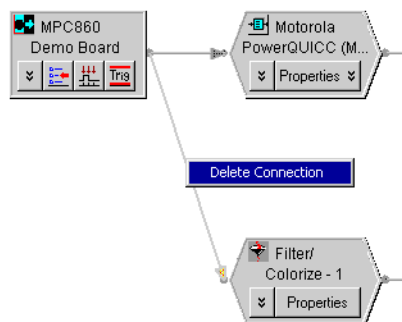
To duplicate windows

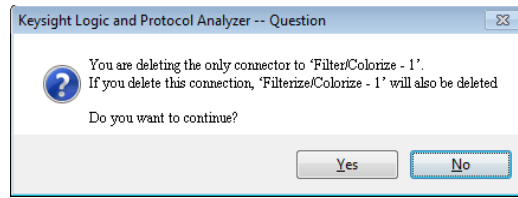
In the Overview window, you can duplicate windows from window menus. Duplicating a window is the same as adding a new window except that the new window has the same properties of the duplicated window.



To delete connections

- In the Overview window, select the connection you wish to delete.
- Select **Delete Connection**.





Deleting a connection has the effect of deleting the window or tool at the end of the connection.

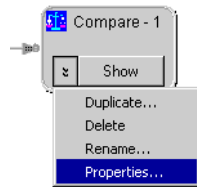
To add connections There is no way to draw connections between modules, tools, and windows other than by adding new windows or tools. See: Connection Rules (see [page 418](#)).

- See Also
- Analyzing the Same Data in Different Ways (Using the Overview Window) (see [page 307](#))
 - To turn window tabs on/off (see [page 313](#))

To edit window or tool properties

To edit window properties

- In the Overview window, select the **Properties...** command from the window menu.



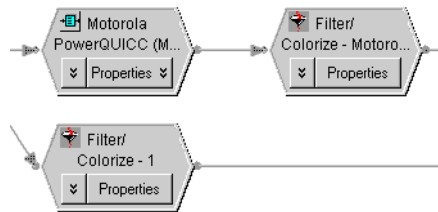
Or, from the menu bar, select the **Properties...** command from the window's menu.

Or, right-click in the display window and select the **Properties...** command from the popup menu.

- In the window's properties dialog, make the desired changes.
- Select **OK** to apply the changes and close the dialog.

To edit tool properties After adding a new tool such as a filter or inverse assembler, you can modify its properties as you refine your analysis of the data.

- 1 In the Overview window, click **Properties** on the tool.



Or, from the menu bar, select **Tools>tool name**.

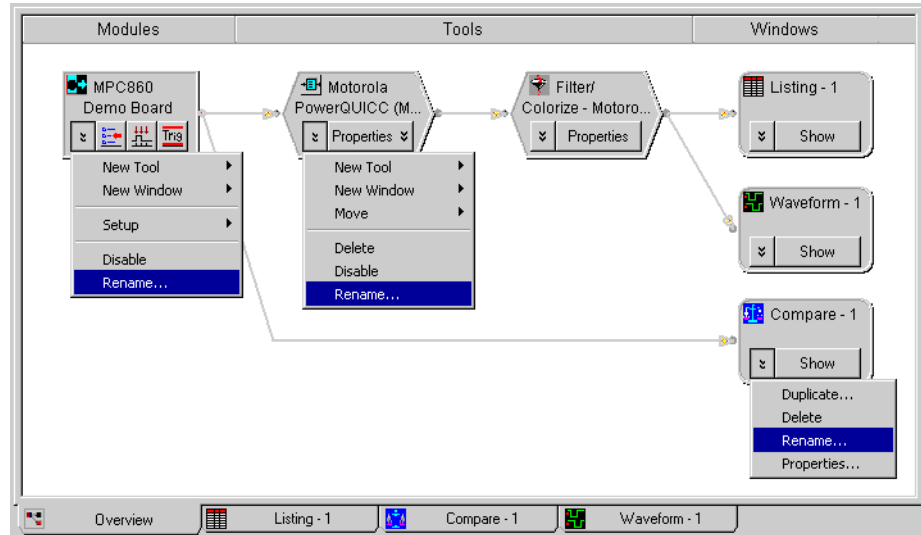
- 2 In the tool dialog box, change properties.
- 3 Select **OK** to apply the changes and close the box.

- See Also
- Analyzing the Same Data in Different Ways (Using the Overview Window) (see [page 307](#))
 - To set waveform window properties (see [page 227](#))

- To set listing window properties (see [page 238](#))
- To set Compare window properties (see [page 276](#))

To rename windows, tools, and modules

- 1 Display the Overview window.
- 2 Select the **Rename...** command from the window, tool, or module menu.



See Also • Analyzing the Same Data in Different Ways (Using the Overview Window) (see [page 307](#))

To redraw the Overview window

- 1 Display the Overview window.
- 2 Select the **Overview>Redraw** command, or:
Right-click in the Overview window and select **Redraw**.

See Also • Analyzing the Same Data in Different Ways (Using the Overview Window) (see [page 307](#))

To delete the Overview window

- 1 Display the Overview window.
- 2 Select the **Overview>Delete** command, or:
If Tabbed Windows (see [page 313](#)) are turned on, right-click the Overview tab and select **Delete**.

See Also • Analyzing the Same Data in Different Ways (Using the Overview Window) (see [page 307](#))

Setting the System Trigger and Skew Between Modules

When there are multiple *module* (see [page 616](#))s in a logic analyzer or logic analysis system, there is a single *Time=0* point for all modules. If one module arms another, the second module has a trigger that is not at *Time=0* with respect to the first module. Because there is a single *Time=0* point, when you see one module captures an event at *Time=-435 ns* and another module captures an event at *-835 ns*, you know the two events occurred 400 ns apart.

This means one module's trigger reference point must be designated the *system trigger* (which is *Time=0*).

You can specify the skew between the *system trigger* and the trigger reference points of other modules. When two modules are looking at the same data, you may want to specify skew so that the waveforms from the two modules line up.

In all display windows, there are global, immovable trigger markers for each module. The marker for the *system trigger* has a special icon.

Each display window has its own *Beginning Of Data* and *End Of Data* markers based upon the buses and signals displayed in that window. For example, if Bus1 is acquired on Logic Analyzer-1 and Bus2 is acquired on Logic Analyzer-2 and both buses are included in Viewer1, then the *Beginning Of Data* will be the earliest sample in either Logic Analyzer-1 or Logic Analyzer-2, and the *End Of Data* will be the latest sample in either Logic Analyzer-1 or Logic Analyzer-2. If Viewer1 only contains buses from Logic Analyzer-1, then its beginning and end of data are only based upon Logic Analyzer-1.

To set the system trigger and skew between modules:

- 1 From the main menu, choose **Setup>Skew & System Trigger....**
- 2 In the Module Skew and System Trigger dialog (see [page 446](#)), select the module whose trigger reference point is to be *Time=0* as the **System Trigger**.
- 3 To specify skew for other modules, enter the appropriate values in their **Skew** fields.
- 4 If you want the *system trigger* to be changed after the next run to the first module that triggers, check **First module to trigger designates the System Trigger**.
- 5 Click **OK**.

If a module is not the *system trigger*, the module icon in the Overview window is the standard logic analyzer icon:



When a module is designated the *system trigger*, an additional red "T" icon appears:



Using Display Windows

- To add or delete display windows (see [page 313](#))
- To turn window tabs on/off (see [page 313](#))

See Also • To change the "Go to Trigger on Run" option (see [page 196](#))

To add or delete display windows

You can add new listing and waveform display windows to the interface. As new windows are added, they appear in the list under **Window** in the menu bar. The active window will have a check mark. All available windows can be accessed either through the menu bar or through the use of tabs.

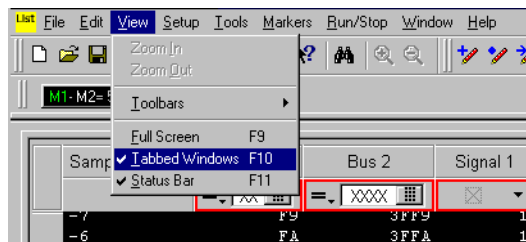
- | | |
|------------------------------|--|
| To add a new display windows | 1 In the menu bar, click Window>New Listing or New Waveform . If the windows are tabbed, you can also right-click on the tab, then select New Listing or New Waveform. |
| To delete display windows | 1 From the menu bar, click Window>Close . If windows are tabbed, you can also right-click on the tab, then select Close. |

See Also • To turn window tabs on/off (see [page 313](#))

To turn window tabs on/off

By default, the Listing and Waveform display windows are tabbed for ease of switching between displays.

To turn on or off window tabs, select **View>Tabbed Windows**.



To switch display windows when tabs are turned off, you must select **Window>"display window name"**.

See Also • To add or delete display windows (see [page 313](#))

Printing Captured Data

There are three ways to create printed documentation of your measurement:

- To print captured data (see [page 314](#))
- To copy text to the clip board (see [page 314](#))
- To copy a screen to the clip board (see [page 315](#))

- See Also
- To install a printer (see [page 315](#))
 - To connect a LAN (see [page 315](#))

To print captured data

- 1 From the menu bar, select **File>Print...**

NOTE

The first time you access the print dialog, you are asked to install a printer (see [page 315](#)). Follow the directions in the printer install dialogs that appear.

- 2 In the **Print What** section, select the desired display window.
- 3 To change the headers, footers, or margins, click **Options...** and specify the changes in the resulting dialog box. When you are done, click **OK**.
- 4 In the **Print range** section, select either:
 - **All**
 - **Time** range
 - **Sample** range
 - **Marker** range

If you selected **Time**, **Sample**, or **Marker**, set the desired range by entering or selecting the **from** and **to** values.
- 5 Click **OK** to print the specified data.
Data is printed from the smallest time/sample to the largest.

To print captured data to ASCII text files

Set up a generic/text only printer that prints to the "FILE:" port. In the Windows **Add Printer Wizard**:

- 1 Select a **Local** printer. (Do not automatically detect and install a plug and play printer.)
- 2 Select the **FILE:** port.
- 3 Select the **Generic** manufacturer and the **Generic / Text Only** printer model.

After you have set up the generic/text only printer, you can print captured data to it just like any other printer.

Note that you can also export captured data to CSV format ASCII text files (see [page 199](#)).

- See Also
- To export data to CSV format files (see [page 199](#))
 - To install a printer (see [page 315](#))

To copy text to the clip board

- 1 From the listing display area, position the mouse cursor over the upper-left corner of the desired display region.
- 2 Click and hold the left mouse button, then drag the mouse cursor to the lower-right corner. Release the mouse button. A rectangle is drawn around the defined region (snaps to state lines and bus/signal columns).

- 3 From the shortcut list that appears, click **Copy Text**.
- 4 Open a word processor or spreadsheet program, paste the text into the program, and print the pasted data text.

To copy a screen to the clip board

- 1 Click **Edit>Copy Screen**. The currently displayed window is copied into the windows clip board buffer.
- 2 Paste the contents of the clip board buffer into a graphics editing program of your choice.
- 3 Print the screen from the graphics program.

To install a printer

Local and network printers are installed outside of the logic analyzer environment using the Windows printer install wizard.




- 1 Click **Start>Settings>Printers**.
- 2 Click on an **existing printer**, or click **Add Printer**.
- 3 Follow the Windows printer install wizard instructions.

See Also · To connect a LAN (see [page 315](#))

To connect a LAN

Local area networks (LAN) are installed outside of the logic analyzer environment using the Windows network configuration wizard.

- 1 Click **Start>Settings>Network and Dial-up Connections**.
- 2 Click an **existing connection**, or click **Make New Connection**.
- 3 Follow the Windows network install wizard instructions.

See Also ·  "AXIe based Logic and Protocol Analysis – Quick Start Guide"
 ·  "AXIe based Logic and Protocol Analysis – Installation Guide"
 ·  "16850 Series Portable Logic Analyzers Installation/Quick Start Guide"
 · "Changing the Windows XP Firewall Settings" (in the online help)
 · Network Troubleshooting Guide (see [page 348](#))

8 B4661A Memory Analysis Software Tools

The B4661A software package includes a variety of tools that can help you perform DDR/LPDDR post-process as well as real-time protocol compliance violation testing, monitor and improve performance of your SDRAM, decode and view memory transactions, and set up your logic analyzer for DDR/LPDDR data capture and analysis.

You can download and install this package from www.keysight.com/find/B4661A page.

Tools installed with the B4661A Memory Analysis Software

When you install the B4661A Memory Analysis software package, the following tools are installed. Some of these tools require that you purchase and install the required software license to use them. Others are standard tools and do not require any license for use.

Software Tool	Description and Usage	License Requirement	Where to find detailed information on tool?
DDR Bus Decoder (With Address Conversion and Trigger Tool)	Allows you to decode and view transactions, commands, and data from a DDR2, DDR3, or DDR4 memory bus in your target system. Includes default configurations for DDR probing solutions for Keysight logic analyzers	Licensed tool Requires B4661A-1 software license for the Address Conversion and Trigger functionality	Refer to the tool's online help that gets installed with this tool. DDR Bus Decoder Online Help
LPDDR Bus Decoder (With Address Conversion and Trigger Tool)	Allows you to decode and view transactions, commands, and data from a LPDDR1, LPDDR2, LPDDR3 or LPDDR4 SDRAM memory bus in your device under test. Includes default configurations for LPDDR probing solutions for Keysight logic analyzers	Licensed tool Requires B4661A-2 software license for the Address Conversion and Trigger functionality	Refer to the tool's online help that gets installed with this tool. LPDDR Bus Decoder Online Help
DDR/LPDDR Post Process Compliance	Evaluates a captured DDR/LPDDR trace against a set of user-defined limits to help you validate that a memory system is operating properly.	Licensed tool Requires B4661A-3 software license	Refer to the tool's online help that gets installed with this tool and is available in the tool's GUI.
Real-Time Compliance	Evaluates DDR/LPDDR data captured by a Keysight Logic Analyzer in real-time to detect and report violations to compliance limits specified in DDR/LPDDR specifications.	Licensed tool Requires B4661A-3 software license	Refer to the tool's online help that gets installed with this tool and is available in the tool's GUI.
DDR3/4 and LPDDR2/3/4 Performance Analysis (Available as Memory Analysis window in Logic and Protocol Analyzer GUI)	Helps you to decode and process the captured memory data to analyze the performance of your SDRAM.	Licensed tool Requires B4661A-4 software license to get the full feature set and capabilities of the tool	Refer to the help book Analyzing Memory Data using the Memory Analysis Window in the Logic and Protocol Analyzer help.

Software Tool	Description and Usage	License Requirement	Where to find detailed information on tool?
ONFi Analysis (Available as ONFi Analysis window in Logic and Protocol Analyzer GUI)	Helps you to decode and process the captured ONFi data to analyze the performance of your NAND Flash memory device.	Licensed tool Requires B4661A-4 software license to get the full feature set and capabilities of the tool	Refer to the help book Analyzing ONFi Data using the ONFi Analysis Window in the Logic and Protocol Analyzer help.
DDR Setup Assistant	A wizard-like application that helps you set up your logic analyzer properly for DDR/LPDDR data capture and analysis.	Standard tool Does not require a license	Refer to the online help book DDR Setup Assistant Online Help that gets installed when you install this tool.
DDR/LPDDR Custom Configuration Creator	Allows you to define the footprints layout as per your custom probing solution used in the DDR/LPDDR setup and then create an XML configuration file based on this footprint information.	Standard tool Does not require a license	Refer to the tool's online help that gets installed with this tool and is available in the tool's GUI.

For all the above-mentioned tools, you can download the user guides from www.keysight.com/find/B4661A page. These guides are also available at the following location when you install the Logic and Protocol Analyzer software and B4661A software:

<logic Analyzer Install location>\help\pdfs

9 Managing Software Licenses

To view active software license information / 322

To activate software licenses / 323

To access floating license servers / 324

To borrow floating licenses and return them early / 326

Starting with the 3.20 release of the *Keysight Logic Analyzer* application, you are able to order *floating* (also known as *counted*) licenses for tools and other add-in software. (Previously, all licenses were *node-locked*.)

NOTE

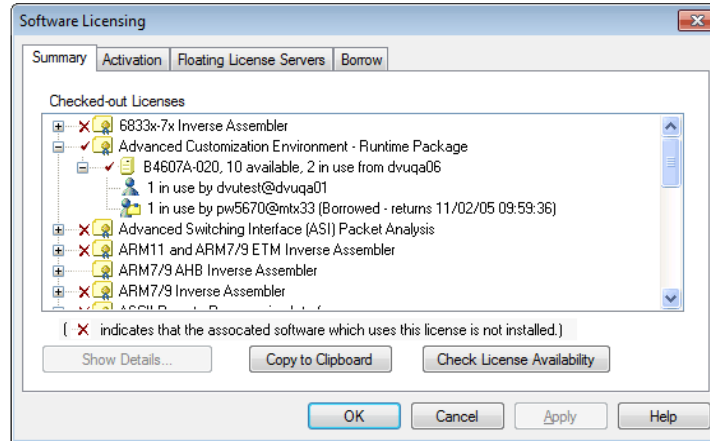
With the 3.20 release of the *Keysight Logic Analyzer* application, you had to set up the `LM_LICENSE_FILE` environment variable to access floating license servers. Starting with the 3.30 release, license servers are accessed from within the *Keysight Logic Analyzer* application, and you *must not* use the `LM_LICENSE_FILE` environment variable any more.

Before you can use floating licenses, you need to set up a license server.

- See Also
- ["License Server Administration Guide"](#) for more information on setting up license server.
 - Software Licensing Dialog (see [page 461](#))

To view active software license information

- 1 From the main menu, choose **Help > Software Licensing...**
- 2 In the Software Licensing dialog's Summary tab (see [page 461](#)):



- You see all the software licenses that can be used.
- Red check marks show floating licenses that are already in use.
- Red "X"s show that software is not installed.
- You can select a license and click **Show Details...** to see detailed information about the license.
- You can copy all licensing summary information to the clip board.

NOTE

When an "Advanced Customization Environment - Development Package" floating license is used, it is taken for the whole session. You must open a new configuration to return the license.

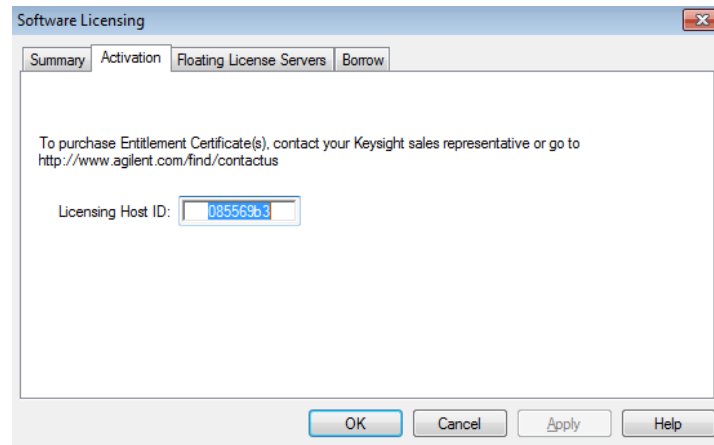
NOTE

When an "ASCII Remote Programming Interface Package" floating license is used, it is taken for as long as the *Keysight Logic Analyzer* application runs. You must close the application to return the license.

- See Also
- To activate software licenses (see [page 323](#))
 - To access floating license servers (see [page 324](#))
 - To borrow floating licenses and return them early (see [page 326](#))

To activate software licenses

- 1 Follow the instructions on the Entitlement Certificate you received with your software purchase.
- 2 From the *Keysight Logic Analyzer* application's main menu, choose **Help>Software Licensing...**
- 3 In the Software Licensing dialog's Activation tab (see [page 463](#)), copy the **Licensing Host ID**. You will need this when activating licenses.



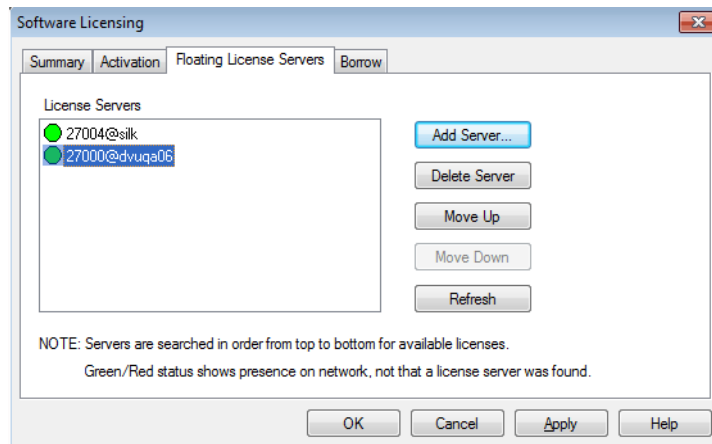
- 4 Visit the Keysight license redemption web site. The URL should be printed on the Entitlement Certificate.
The license redemption web site will use the order number or other license activation code which is printed on the certificate, along with the Licensing Host ID, to generate a license file. The license file will be e-mailed to you.
- 5 To install the license file and enable the software, follow the instructions in the e-mail that contains the license file.
Those instructions will tell you to install the license file in the proper directory on the logic analysis system or floating license server and restart the *Keysight Logic Analyzer* application or license server. On a logic analysis system, the license directory is usually "C:\Program Files (x86)\Agilent Technologies\Logic Analyzer\License". For the proper directory on a license server, see the ["License Server Administration Guide"](#).
The license file must have a .lic extension.

- See Also
- To view active software license information (see [page 322](#))
 - To access floating license servers (see [page 324](#))
 - To borrow floating licenses and return them early (see [page 326](#))

To access floating license servers

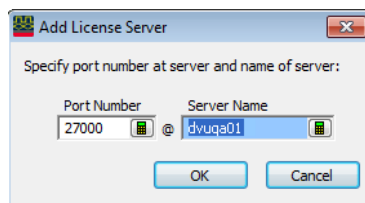
Before you can use floating licenses, you need to set up a license server (see ["License Server Administration Guide"](#)).

- 1 Open the *Keysight Logic Analyzer* application (with the default configuration) so that no floating licenses are in use.
- 2 From the main menu, choose **Help>Software Licensing...**
- 3 Select the Software Licensing dialog's Floating License Servers tab (see [page 464](#)).



To add a floating license server

- 1 Click **Add Server...**
- 2 In the Add License Server dialog, enter the port number and name of the floating license server.



The port number is typically 27000, but it can be different depending on how the floating license server was set up.

CAUTION

Only enter names of computers (or logic analyzers) that are floating license servers. Otherwise, the license manager interface hangs up for many minutes trying to determine if the computer is really a floating license server.

- 3 Click **OK** to close the Add License Server dialog.
- 4 In the Software Licensing dialog, click **Apply**.

To move a server up or down in the search order

- 1 In the License Servers list, select the license server you want to move.
- 2 Click **Move Up** or **Move Down**.
- 3 Click **Apply**.

To refresh floating license server status


- 1 Click **Refresh**.

The green or red server availability indicators are only a check of whether the computer is on the network, not of whether the license server software is running on that computer.

To delete a floating license server

- 1 In the License Servers list, select the license server you want to delete.
- 2 Click **Delete Server**.
- 3 Click **Apply**.

See Also

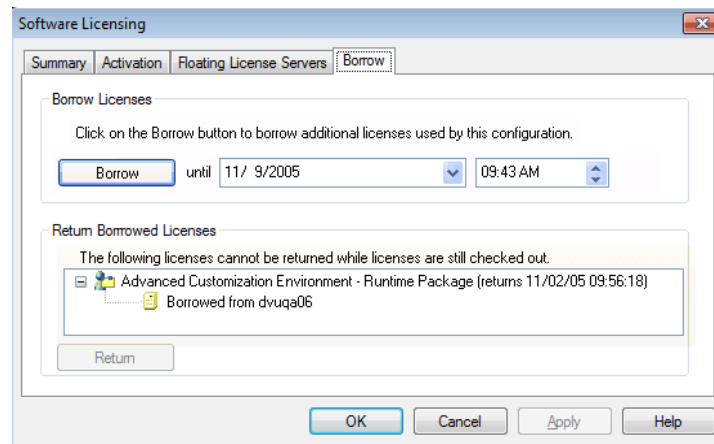
-  ["License Server Administration Guide"](#)
- To view active software license information (see [page 322](#))
- To activate software licenses (see [page 323](#))
- To borrow floating licenses and return them early (see [page 326](#))

To borrow floating licenses and return them early

You can borrow floating licenses from a server for a period of time, for example, if you're taking a logic analyzer (or a computer running the *Keysight Logic Analyzer* application) out of the office (or just off the network). When a borrowed license's time expires, the license is automatically returned to the server. However, you can also return licenses early.

To access the Software Licensing dialog's Borrow tab

- 1 From the main menu, choose **Help>Software Licensing...**
- 2 Select the Software Licensing dialog's Borrow tab (see [page 465](#)).



To borrow floating licenses

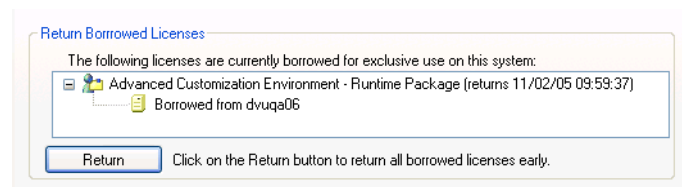
- 1 Set up the configuration (or open a configuration file) that uses the software you need to borrow licenses for.
- 2 Access the Software Licensing dialog's Borrow tab.
- 3 In the Borrow Licenses area, enter the date and time when the borrowed license will be returned. The default time is seven days. The minimum time is ten minutes.
- 4 Click **Borrow**.

Repeat these steps to borrow additional licenses.

To return floating licenses early

When returning borrowed floating licenses early, all borrowed licenses must be returned. You are not able to return borrowed licenses while any licenses are checked out.

- 1 Open the *Keysight Logic Analyzer* application (with the default configuration).
- 2 Access the Software Licensing dialog's Borrow tab.
- 3 In the Return Borrowed Licenses area, click **Return**.



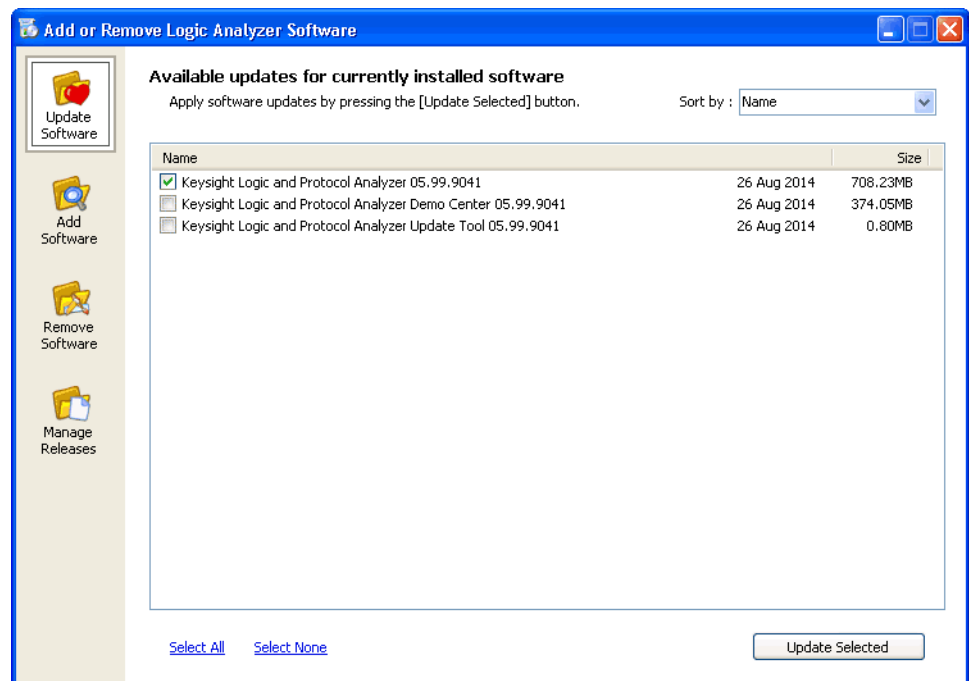
- See Also
- To view active software license information (see [page 322](#))
 - To activate software licenses (see [page 323](#))
 - To access floating license servers (see [page 324](#))

10 Updating Software

You may be able to install logic analyzer software from the logic analysis system's hard disk (depending on when it shipped from the factory or the application install CD that was last used).

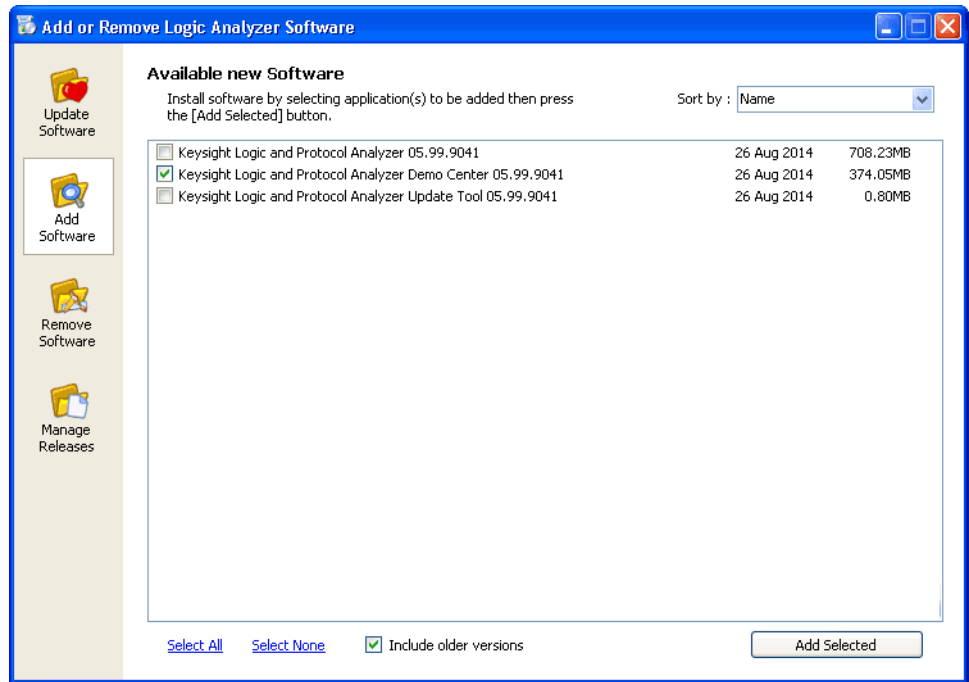
To update, add, or remove logic analyzer software, or to remove the install packages from previous releases (and free disk space):

- 1 In the *Keysight Logic Analyzer* application, choose **Help>Software Update...**
- 2 In the Add or Remove Keysight Logic Analyzer Software tool:
 - To update software: click **Update Software** and select the software you want to update; then, click **Update Selected**.

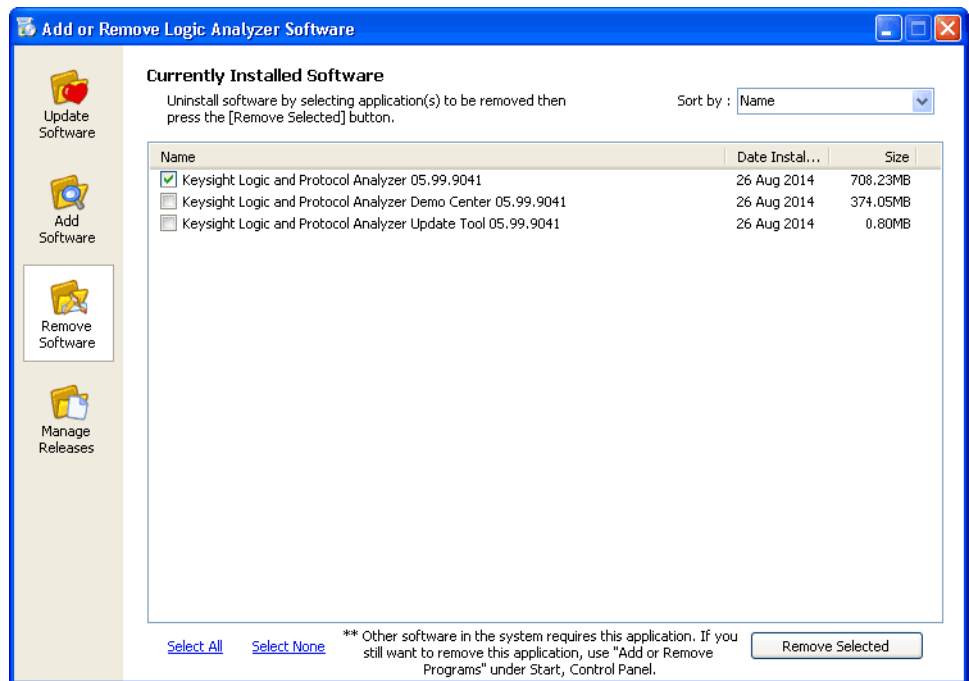


- To add software: click **Add New Software** and select the software you want to add; then, click **Add Selected**.

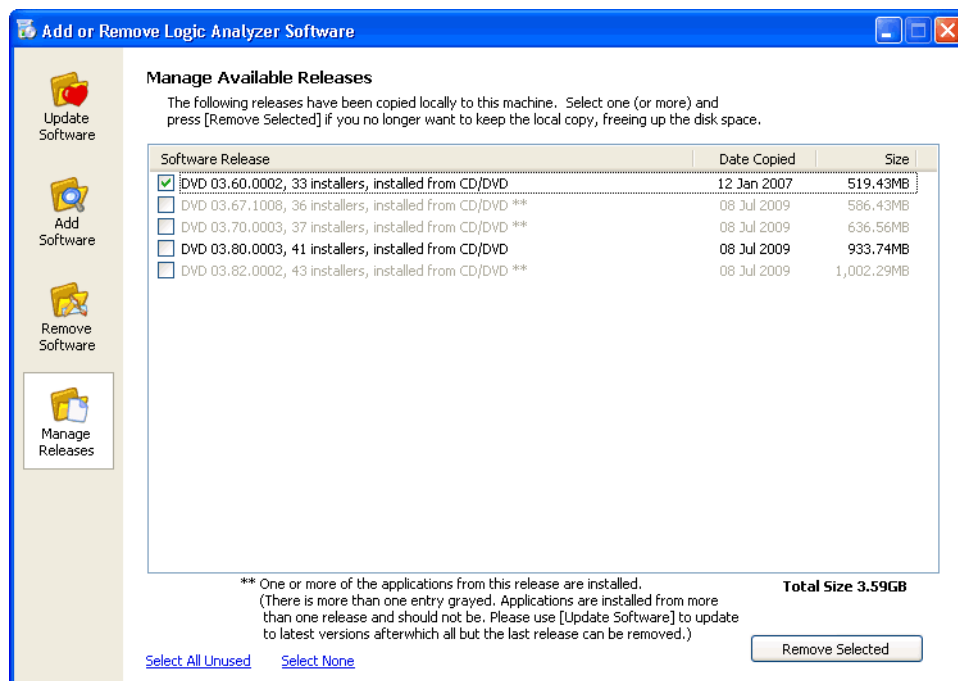
When adding new software, you can show all versions or just the latest version.



- To remove software: click **Remove Software** and select the software you want to remove; then, click **Remove Selected**.



- To remove old releases and free disk space: click **Manage Releases** and select the release whose files you want to remove; then, click **Remove Selected**.



Note that, when adding updating, or removing software, you can select all software or none, and you can sort the software list by name, version, size, or date.

See Also You can also download and install the latest versions of logic analyzer software from the Keysight web site:

- ["http://www.keysight.com/find/la-sw-download"](http://www.keysight.com/find/la-sw-download)

11 Solving Problems

When troubleshooting problems or looking for more information, see:

- If starting in offline mode is unexpected (see [page 332](#))
- If an ALA format configuration file won't open (see [page 333](#))
- Interpreting Error Messages (see [page 334](#))
- License Problems (see [page 342](#))
- Running Self Tests (see [page 343](#))
- Network Troubleshooting Guide (see [page 348](#))
- Remote Desktop Set Up (see [page 350](#))

- See Also
- For More Information (see [page 351](#))
 - Keysight Logic Analyzer Readme

If starting in offline mode is unexpected

When starting the *Keysight Logic Analyzer* application on a logic analysis system or logic analyzer, you expect to connect to local hardware. If you have set up to auto-connect to a remote logic analysis system or logic analyzer, you expect to connect to the remote hardware. If the *Keysight Logic Analyzer* application starts in offline mode instead:

- It could be that the *Keysight Logic Analyzer* application is already running a local session. You can run multiple instances of the application, but if there's a local session is already running, additional instances start in offline mode.
- In the case where you have set up to auto-connect to a remote logic analysis system or logic analyzer, it could be:
 - The remote system is powered-down or off the network. In this case, you are given an information dialog about the system being offline before starting in offline mode.
 - The remote system is in the process of having its software updated.
 - The remote system software has been updated, resulting in an "incompatible remote service". In this case, make sure the same version of software is installed on the local computer or logic analysis system.

If an ALA format configuration file won't open

If an ALA format configuration file won't open because modules are incompatible, you can still load the setup information from the ALA format configuration file:

- 1 Load the incompatible ALA format configuration file in offline mode.
(If you're only interested in looking at the data, you can ignore the following steps.)
- 2 Save the configuration's setup information as an XML format configuration file (see [To save a configuration file \(see page 198\)](#)).
- 3 Go back online (see [Returning to Online Analysis \(see page 59\)](#)).
- 4 Open the XML format configuration file (see [To open a configuration file \(see page 208\)](#) and possibly [To transfer module setups to/from multi-module systems \(see page 210\)](#)).
- 5 Save the loaded setup information to an ALA format configuration file.

This procedure converts an incompatible ALA format configuration file into one that is compatible.

Interpreting Error Messages

To locate the error you received, use the help window's Search tab to search for key words in the error message.

- Error Messages (see [page 334](#))
- Warning Messages (see [page 339](#))
- Informational Messages (see [page 340](#))

See Also • Solving Problems (see [page 331](#))

Error Messages

- Acquisition Errors (see [page 334](#))
- Bus/Signal Errors (see [page 334](#))
- File Errors (see [page 336](#))
- Hardware Errors (see [page 337](#))
- Help File Errors (see [page 337](#))
- Trigger Errors (see [page 337](#))

See Also • Interpreting Error Messages (see [page 334](#))

Acquisition Errors

An acquisition error has occurred due to state clock edges occurring too close together.

This could be the result of:

- Poor state clock quality (signal integrity).
- Inadequate probe grounding (try multiple grounds around clock signals).
- State clock edges spaced closer than specifications allow.
- Multiple clocks selected and spaced closer than specifications allow.

When in the state acquisition mode, the logic analyzer requires a clear clock signal no faster than the maximum state clock speed (see Specifications and Characteristics (see [page 587](#))). Poor state clock quality may be caused by loading in the device under test. It may also be caused by a clock setup (see [page 124](#)) in the Sampling Setup dialog that is a combination of several signals which combined together violate the clock specification. When your clock setup uses multiple edges, the logic analyzer's setup/hold time typically increases (see Specifications and Characteristics (see [page 587](#))). When you are using a clock speed near the specification, grounding every second or third probe connection is recommended.

Bus/Signal Errors

Maximum of 128 channels per Bus.

The logic analyzer cannot handle buses that contain more than 128 channels (signals). If you require wider buses, try breaking the bus into two or more buses, for example Data_HI and Data_LO.

Cannot group into Bus. Maximum of 128 channels per Bus.

The logic analyzer cannot handle buses that contain more than 128 channels (signals). If you require wider buses, try breaking the bus into two or more buses, for example Data_HI and Data_LO.

The following Bus/Signals are required to have a specific number of assigned channels because they are locked. Please correct the following Bus/Signals: *name* (has *num1* channels, requires *num2* channels)

Some tools may "lock" buses and signals that are necessary to produce their output. The locked buses and signals may have their specific channel assignments changed, but the total number of channels on each bus or signal must stay the same. Please change the channel assignment for each indicated bus or signal so that the width is *num2*. This message sometimes appears in combination with the next one. In these cases, you may have changed a configuration to use half the pods for sampling. Check the sampling tab.

Every Bus/Signal requires at least one assigned channel. Please assign channels to the following Bus/Signals:

Every bus or signal requires at least one channel. If you do not see the Bus/Signal named in the error dialog, try scrolling the Bus/Signal listing. Certain tools may also have created buses or signals within folders. If you are trying to avoid showing extra information on the viewer, delete the row (see [page 227](#)) or column (see [page 238](#)) the bus or signal is in. This removes the information from the viewer without losing the bus/signal setup information. This message sometimes appears in combination with the previous one. In these cases, you may have changed a configuration to use half the pods for sampling. Check the sampling tab.

Minimum of one Bus/Signal with assigned channels required. Please add a Bus/Signal.

The bus/signal setup cannot be closed because you have deleted all buses and signals. Folders only contain buses and signals, but do not represent data mappings of themselves. In order to close the dialog, select **Add Bus/Signal**. Assign at least one channel to the new bus or signal. Alternatively, you can select **Cancel** and revert to the previous bus and signal assignments.

name is locked and cannot be deleted because it is required by another tool in the application. In order to unlock it, the following tools must be deleted:
tool

Some tools may "lock" buses and signals that are necessary to produce their output. Until the tool is deleted via **Tools>Overview**, you cannot delete or rename the bus or signal.

name is locked and cannot be renamed because it is required by another tool in the application. In order to unlock it, the following tools must be deleted:
tool

Some tools may "lock" buses and signals that are necessary to produce their output. Until the tool is deleted via **Tools>Overview**, you cannot delete or rename the bus or signal.

Cannot change pod selection while there are locked Bus/Signals.

Some tools may "lock" buses and signals that are necessary to produce their output. Until the tool is deleted via **Tools>Overview**, you cannot modify the bus or signal by changing the pod in use.

Cannot delete *folder* because it contains one or more locked children.

Some tools may "lock" buses and signals that are necessary to produce their output. The folder you have tried to delete contains a unique copy of at least one locked bus or signal. You can move the locked buses or signals outside the folder, and then delete the folder. Alternatively, you can delete the tool locking the buses or signals via **Tools>Overview**, and then delete the folder.

Unable to set setup and hold times for this Bus/Signal since no channels have been assigned. Please assign channels before using setup and hold.

The possible valid range of setup and hold values depends on the clock setup used by the pods that the channels are attached to. Without knowing which pods' channels are part of the bus or signal, it is impossible for the logic analyzer to set appropriate ranges. Please assign channels to the bus or signal, and then set setup and hold.

Please enter a user threshold value.

All pods will be set to the same threshold value. If you select OK without setting a value, the current threshold values (at least one of which is different from the rest) are retained. Please check the dialog and be sure all fields are filled in.

Please enter a threshold value.

All pods will be set to the same threshold value. If you select OK without setting a value, the current threshold values (at least one of which is different from the rest) are retained. Please check the dialog and be sure all fields are filled in.

File Errors

Error trying to remove file: "*directory/hardware_log.txt*".

When the logic analyzer is started up, it replaces the old hardware_log.txt file. For some reason, this time the old hardware log was not able to be deleted. This could indicate a problem with the disk that the file is stored on.

File "*filename*" could not be opened.

When the logic analyzer is started up, it creates a new hardware_log.txt file. For some reason, this time the log was not able to be opened after creation. This could indicate a file system or disk problem.

Hardware Errors

Analyzer Calibration Failed [<i>time</i>] - Instrument may need service.	The logic analyzer's pre-measurement calibration failed. Any data collected after receiving this error message is possibly incorrect. If the failure is transient, cycling power may fix the problem. If the failure is persistent, run Help>Self Test... or call your Keysight Sales Office to arrange for service.
High speed system clock failure - Instrument may need service. Sleep Duration and Count: <i>duration</i> , <i>counted</i> .	The internal 100 MHz clock did not pass initialization tests. Any measurements are likely to be faulty. Please contact Keysight Technologies sales or support at " http://www.keysight.com/find/contactus " for information on getting the instrument repaired.

Help File Errors

Help file information was not found in the registry. You may need to reinstall the tool.	The logic analyzer could not find a registry entry for the help file associated with the tool. If you have done a custom installation of the tool, you must also install the help file to access help. If the problem persists after re-installing, please contact Keysight Technologies sales or support at " http://www.keysight.com/find/contactus " for assistance.
Help file information not found in registry. Cannot display help.	The logic analyzer could not find a registry entry for the help file associated with the tool. If you have done a custom installation of the tool, you must also install the help file to access help. If the problem persists after re-installing, please contact Keysight Technologies sales or support at " http://www.keysight.com/find/contactus " for assistance.
Help file not found. Cannot display help.	The help file was not found where specified by the registry. It may have been deleted or moved. You can search the drive where the logic analyzer software is installed for .chm files, or re-install the tool.
The HTML Help file " <i>filename</i> " was not found. You may need to re-install the product.	The help file was not found where specified by the registry. It may have been deleted or moved. You can search the drive where the logic analyzer software is installed for the file, or re-install. To re-install, close the logic analyzer application and run the setup program on the logic analyzer CD.
The HTML Help file " <i>filename</i> " was not found. You may need to re-install the tool.	The help file for the tool was not found where specified by the registry. It may have been deleted or moved. You can search the drive where the logic analyzer software is installed for .chm files, or re-install the tool.

Trigger Errors

Only one action per timer per branch is allowed.	A branch is the collection of actions after a "Then" in an Advanced Trigger step. Some steps, such as Advanced 2-Way Branch (see page 500), may have multiple branches. Within a branch, only one of Start from reset, Stop and reset, Pause, or Resume is allowed per timer. For more on timers, see To configure a timer (see page 153).
--	--

Only one action per counter per branch is allowed.	A branch is the collection of actions after a "Then" in an Advanced Trigger step. Some steps, such as Advanced 2-Way Branch (see page 500), may have multiple branches. Within a branch, you can not both Increment and Reset the same counter. You can increment one and reset the other. For more on counters, see To configure a counter (see page 154).
Only one store action per branch is allowed.	A branch is the collection of actions after a "Then" in an Advanced Trigger step. Some steps, such as Advanced 2-Way Branch (see page 500), may have multiple branches. Within a branch, you can set Store sample or Don't store sample but not both in the same branch. If you do not specify any store actions, default storage (see page 167) is used.
Only one reset occurrence counter action per branch is allowed.	A branch is the collection of actions after a "Then" in an Advanced Trigger step. Some steps, such as Advanced 2-Way Branch (see page 500), may have multiple branches. Within a branch, you can only specify Reset occurrence counter once.
No more edge resources available for this pod pair.	The logic analyzer hardware can only handle two edge statements per pod pair in <i>Full Channel Timing Mode</i> or <i>Half Channel Timing Mode</i> , or one edge statement per pod pair in <i>Transitional / Store Qualified Timing Mode</i> (see page 536). If the edges are on different signals, try probing one of the signals with another channel on another pod pair. If all the edges are being used on the same signal, replace the "either edge" terms with "rising edge OR falling edge". See To insert events (see page 179) for how to replace "either edge".
No more pattern resources available for this pod pair	The logic analyzer hardware has a limited number of pattern (bus value) variables per pod pair. If the values you are checking for are on different buses, try probing one of the buses with another pod pair.
Branch expression is too complex	<p>The expression in one of the branches of the trigger specification is too complicated for the logic analyzer. The logic analyzer first combines all AND terms and then ORs the expressions together. AND terms that have more than 4 events use twice the resources. Try rewriting the branch expression to use more OR terms, or delete some events.</p> <p>One situation that leads to this error is using the In Range and Not In Range operators with buses that span more than 2 pod pairs. These operators are limited to buses that span 2 or fewer pod pairs (up to 64 bits wide).</p>
Trigger Specification is too complex	Although no single branch expression is too complex, the total number of ANDs and ORs has exceeded the logic analyzer's resources. Try simplifying some expressions in some steps, or removing steps altogether.
Replacement failed. Maximum number of sequence steps exceeded.	The logic analyzer translates the trigger you specified into internal sequence steps. Different trigger functions use different numbers of internal sequence steps. Also, the "trigger and fill memory" action requires an additional internal sequence step each time it is used in state acquisition mode. One possible way to simplify the trigger specification is to replace all other "trigger and fill memory" actions with a "goto N" action that points to a "Find anything then trigger and fill memory" step.
Unable to insert step. The maximum number of sequence steps are already allocated.	The logic analyzer translates the trigger you specified into internal sequence steps. Different trigger functions use different numbers of internal sequence steps. Also, the "trigger and fill memory" action requires an additional internal sequence step each time it is used in state acquisition mode. One possible way to simplify the trigger specification is to replace all other "trigger and fill memory" actions with a "goto N" action that points to a "Find anything then trigger and fill memory" step.

Too many sequence steps.	The logic analyzer translates the trigger you specified into internal sequence steps. Different trigger functions use different numbers of internal sequence steps. Also, the "trigger and fill memory" action requires an additional internal sequence step each time it is used in state acquisition mode. One possible way to simplify the trigger specification is to replace all other "trigger and fill memory" actions with a "goto N" action that points to a "Find anything then trigger and fill memory" step.
Goto action specifies an undefined step.	The last step in the trigger sequence includes the action "Goto next". Because there is no next step, the logic analyzer cannot run and look for a trigger. Select Setup>(Logic Analyzer Module)>Advanced Trigger... , and change the action for the last trigger step.
Counter event specified both true and false in the same product term	In the trigger specification, at least one branch ANDs together "bus equals X" and "bus not equal X". Because this condition can never be true, the logic analyzer will not trigger and does not start the acquisition. If you intend to have it run until you press stop, use the trigger function Run Until User Stop, found under the "Other" tab in advanced trigger.
Cannot use <, <=, >, >= for a bus with clock bits that spans pod pairs	You have defined a bus that both spans pod pairs and includes a clock bit. The clock bits are numbered the same as the pod they are located on, and it is possible for them to be the channel that is not on the same pod pair as the others. Check the channel assignment in the Buses/Signals (see page 426) tab of the Setup dialog. The logic analyzer will not run until this problem is corrected.
Cannot specify a range on a bus with clocks bits that spans pod pairs	You have defined a bus that both spans pod pairs and includes a clock bit. The clock bits are numbered the same as the pod they are located on, and it is possible for them to be the channel that is not on the same pod pair as the others. Check the channel assignment in the Buses/Signals (see page 426) tab of the Setup dialog. The logic analyzer will not run until this problem is corrected.

Warning Messages

You are currently running "Offline," so running the analyzer is not possible. If you wish to create "fake" data while offline, go to "Edit -> Options" and select "Create Data When Offline". Note: This setting is persistent from session to session.	The logic analyzer is running in offline mode. Offline mode means that the logic analyzer software does not have access to logic analyzer or logic analysis system hardware. If you have a logic analyzer attached, please check the connection. For more on running with fake data, see Options Dialog (see page 447). Fake data is useful when learning how to use the logic analyzer software.
This module is already being used by another instance of the application. You are now working Offline.	The logic analyzer hardware is attached to an open instance of the logic analyzer software. If you need to acquire data, locate that instance from the Windows taskbar. The Local/Remote/Offline indicator is at the bottom of the application window. In offline mode, the software can still work with saved data.

Event specified both true and false in the same product term	In the Advanced Trigger dialog, one of the branches for one of the steps checks that an event is both true and not true. An event may be a bus or signal equal to a value, a timer expiring, or a count exceeding some value. Because of the AND combination, the branch cannot be true. You may want to modify the trigger to use either an OR combination of the events, or separate them into different branches or steps. For more on constructing complex triggers, see To replace or insert trigger functions into trigger sequence steps (see page 158). For more on how to interpret the trigger sequence, see Reading Event and Action Statements (see page 153).
Timer <i>n</i> value checked as an event, but no start action specified.	In the Advanced Trigger dialog, the trigger sequence checks the value of a timer that was never started. Timers need to be explicitly started in a previous trigger step. See To configure a timer (see page 153) for more information.
Counter <i>n</i> value checked as an event, but no increment action specified.	In the Advanced Trigger dialog, the trigger sequence checks the value of a counter that is never incremented. Counters need to be incremented in the action statements of a trigger step. See To configure a counter (see page 154) for more information.
The Bus/Signals listed below could not be loaded from the configuration file. Please recheck your Trigger since it may have changed.	The configuration file you just loaded was created on a logic analyzer with more pods than this model. Because of this, some buses and signals which rely on the additional pods could not be loaded. If these buses or signals were used in the trigger sequence, the trigger sequence will have changed. You may be able to work around this by assigning different channels to the affected buses and signals, and re-creating the trigger sequence.
Slow or missing clock in Trigger Step <i>n</i> ...	The logic analyzer is not able to detect the state clock, and is therefore unable to take samples and evaluate the trigger sequence. If your device under test's clock is bursty, this may be expected behavior. If it is not, please check all probing connections. To verify the clock signal is being received, you can assign the clock channels to a bus in timing acquisition mode and acquire data.
Informational Messages	
Filling memory after trigger...	The logic analyzer has triggered and is filling memory. Due to either a slow clock or storage qualification in state acquisition mode, or infrequent transitions in transitions-only timing acquisition mode, the logic analyzer is taking enough time to fill memory that this message is showing.
Trigger inhibited during prestore...	The logic analyzer is in timing acquisition mode. In timing acquisition mode, the logic analyzer fills the designated amount of memory before searching for the trigger. If this message is showing, the logic analyzer is filling memory and has not yet begun to compare data to the trigger sequence. To capture triggers that happen during the beginning of a device under test's boot sequence, be sure to set the trigger position in the Sampling tab to 100% poststore.
Waiting in Trigger Step <i>n</i>	The logic analyzer is waiting for a sample that matches the events defined in step <i>n</i> of the trigger sequence. Sometimes the event is rare, causing long waits. If you feel that the logic analyzer should have triggered already, check the trigger sequence in Advanced Trigger. For more on triggering, see Specifying Advanced Triggers (see page 152).
"No signal activity. Check connection, threshold, and stimulus."	This channel appears to be completely quiet. <ul style="list-style-type: none"> • Check the probe connection between the analyzer and the device under test. • Check the threshold voltage setting (see Setting the Logic Analyzer Threshold Voltage (see page 77)).

- Check that the device under test is turned on and is running the appropriate diagnostic or other stimulus program.

If all these things are set up correctly, activity will be shown in the Analyzer Setup dialog's Buses/Signals tab (see [page 426](#)).

"No stable regions.
Check clock and
thresholds."

A common possibilities exists:

- 1 The signal on this channel is asynchronous to the clock defined for the logic analyzer. If this is the case, there is no stable relationship between the times when the signal switches and when the clock arrives.

"No voltage scan:
Channel in pod
with assigned
clock."

"One or more
stable regions
found."

"Only a few
transitions
detected. Change
stimulus or
increase
measurement
duration."

The signal on this channel was observed to toggle fewer than 500 times. The characterization may be accepted as it stands or you may wish to change the stimulus program or diagnostic in the device under test to increase the toggle rate.

"Stable region at N
ns is an estimate."

This message only appears for certain bus probes (not general purpose probes).

"The stable region
extends beyond
the limits of the
display."

This channel is active, but the signal does not switch within 5 ns before or after the clock. For example, this could occur if the propagation delay in the device under test from clock to data is greater than 5 ns and the clock period is greater than 10 ns (slower than 100 MHz).

License Problems

Some problems that can occur with licenses are:

- License Not Available (see [page 342](#))
- Floating License Server Communication Timeout (see [page 342](#))

License Not Available

If you attempt to use a software feature that requires a license, and a license is not available, the License Not Available dialog appears.

Depending on the situation, there are several ways to solve this problem:

- If all licenses are in use, you can wait until one of them becomes available again. If there is only one license, the License Not Available dialog shows you who is using the license. If there are multiple licenses, you can view the active software license information (see [page 322](#)) to see the license users.
- If all licenses are in use or there are no licenses, you may be able to get a license from another license server (see [To access floating license servers \(see page 324\)](#)).
- If the license management software detects that one of your license servers is unavailable, make sure the computer or logic analyzer hosting the license server is running, is accessible over the network, and is running the license service. For more information, see the ["License Server Administration Guide"](#).
- If an expected node-locked license is not found, make sure the license file is located in the "License" subdirectory under the installation directory (typically C:\Program Files (x86)\Agilent Technologies\Logic Analyzer), and make sure the license file has the .lic extension.
- If you decide to purchase additional licenses, contact Keysight Technologies (see ["http://www.keysight.com/find/contactus"](http://www.keysight.com/find/contactus)). When you get your Entitlement Certificate, activate your licenses by using the License Activation Wizard (see [To activate software licenses \(see page 323\)](#)).

See Also • [Managing Software Licenses \(see page 321\)](#)

Floating License Server Communication Timeout

When floating licenses are used, the license subsystem checks for communication with the license server every two minutes.

After 10 minutes of communication loss (6 checks), licenses are considered lost, and you are given a message about the server that is no longer communicating and the features that are disabled.

Depending on the feature, you may be able to continue working in the *Keysight Logic Analyzer* application, or you may be forced to exit the application. In either case, you are able to save your setup and data to a configuration file.

If you are able to continue using the *Keysight Logic Analyzer* application with disabled features, the communication checks continue every two minutes. If communication with the license server is re-established, an information dialog tells you about the server and features that have been re-enabled.

When communication with a floating license server is lost, make sure the computer or logic analyzer hosting the license server is running, is accessible over the network, and is running the license service. For more information, see the ["License Server Administration Guide"](#).

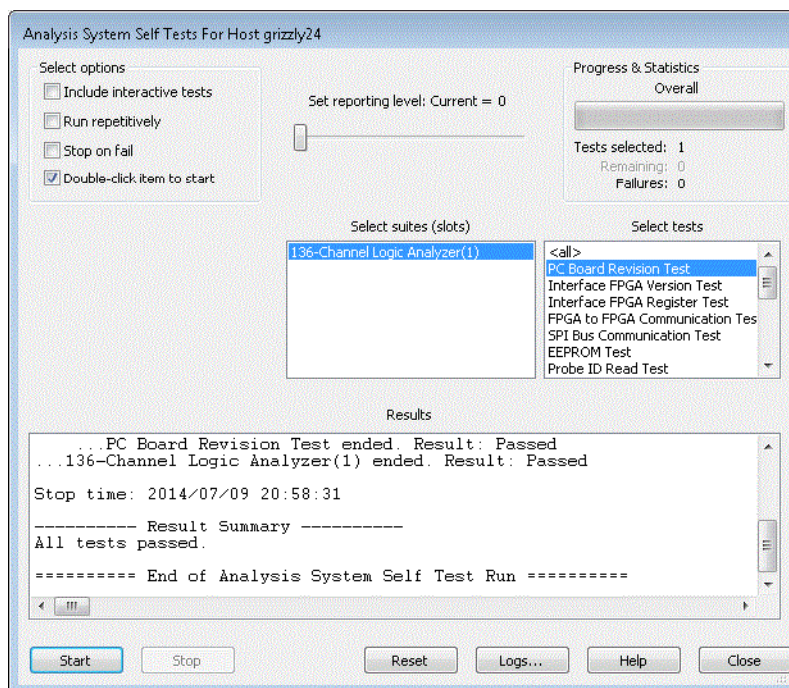
Running Self Tests

The Self Test menu checks the major hardware functions of the logic analysis system to verify that it is working correctly.

CAUTION

Because the most recently acquired data will be lost, be sure to save important data before running self tests.

- 1 From the menu bar select **Help>Self Test...**
If you have acquired data, a warning message appears, "Running self-tests will invalidate acquired data"; click **OK** to continue.



- 2 In the Analysis System Self Tests dialog, select the self test options:
 - **Include interactive tests** – causes interactive tests to appear in the selection lists.
 - **Run repetitively** – runs the selected tests repetitively until you click **Stop**.
 - **Stop on fail** – if you are running multiple tests or running tests repetitively, this causes the tests to stop if there is a failure.
 - **Double-click item to start** – lets you double-click a test to start it.
- 3 Set the reporting level.
Higher levels produce increasingly verbose output.
- 4 If you have a multiframe configuration, select the instruments you want to test.
- 5 If you have a slotted instrument, select the suites you want to run.
- 6 Select the tests you want to run.
- 7 Click **Start**.

As the tests are running, the results are reported in the lower part of the dialog and saved to a log file.

To stop running test, click **Stop**.

To reset the self-test options, click **Reset**.

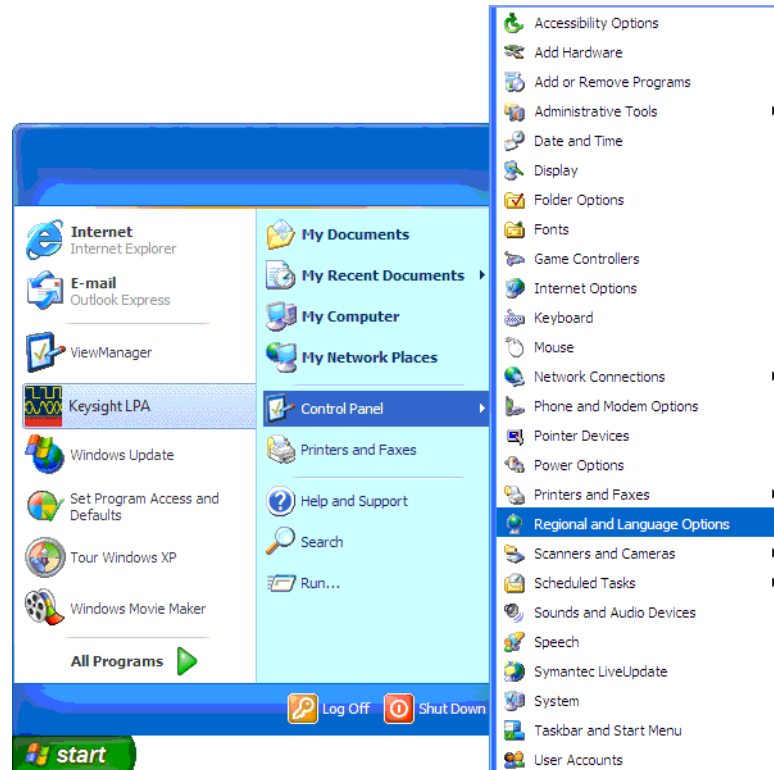
To view the log file, click **Logs...**, select the log file you want to view, and click **Open**.

If, after completing the self tests, you have failures or you have questions about the performance of the logic analysis system, contact Keysight Technologies sales or support at "<http://www.keysight.com/find/contactus>".

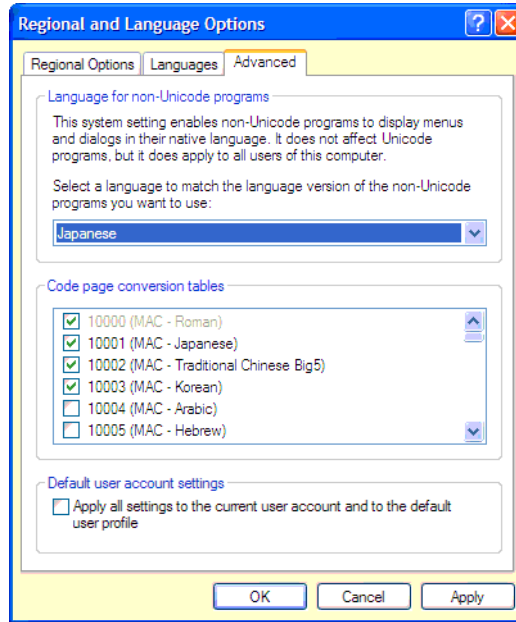
See Also • For More Information (see [page 351](#))

Accessing Japanese Online Help

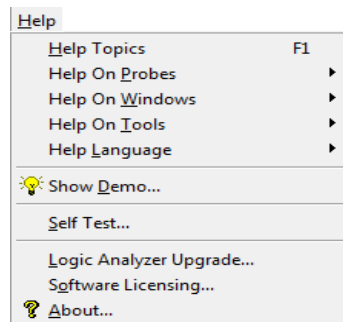
- 1 From the Windows Start menu, choose **Start>Control Panel>Regional and Language Options**.



- 2 In the Advanced tab of the Regional and Language Options dialog:
 - a In the "Language for non-Unicode programs" box, select **Japanese**.



- b Click **OK** to close the Regional and Language Options dialog.
- 3 In the dialog that appears, select the files to copy:
 - On a logic analyzer or logic analysis system, the dialog asks if you would like to use existing files or recopy files from the Windows CD-ROM; click **Yes** to use the existing files.
- 4 A dialog appears asking if you would like to restart your computer; click **Yes** to restart your computer.
- 5 After your computer restarts, start the *Keysight Logic Analyzer* application, and choose **Help>Help Language>Japanese**.



Now, when you access the online help, you get the Japanese version.

On Windows 7:

- 1 From the Windows Start menu, choose **Start>Control Panel>Region and Language**.
- 2 In the **Administrative** tab of the **Region and Language** dialog, click the **Change system locale** button.
- 3 In the **Region and Language settings** dialog box, select **Japanese** from the **Current system locale** listbox.

- 4 Click **OK** and then click **Apply**.
- 5 After your computer restarts, start the Keysight Logic Analyzer application, and choose Help>Help Language>Japanese.

Network Troubleshooting Guide

- Network Setup (see [page 348](#))
 - Network Access Issues (see [page 348](#))
 - Login Issues (see [page 348](#))
 - Using with Multiframe (see [page 348](#))
 - Network Hardware and Configuration (see [page 348](#))
 - Network Topology (see [page 349](#))
 - Known OS Issues (see [page 349](#))
 - Logic Analyzer Specific Issues (see [page 349](#))
 - Keep System Protected, Up-To-Date (see [page 349](#))
- Network Setup** For information on setting up logic analysis systems on the network (and in multiframe configurations), refer to the [AXIe based Logic and Protocol Analysis – Installation Guide](#)
- Is the logic analysis system registered with DNS?
 - Is the logic analysis system registered with DHCP?
- Network Access Issues**
- The link activity light must be on. If these LEDs are not on, the LAN segment may be dead.
 - Can system be accessed on the network from another computer?
 - Note: if on a network without a DHCP server, it can take up to 5 minutes before auto-negotiation configures an IP address. Use **ipconfig** to verify that the IP address is not 0.0.0.0 before trying to do any network activity.
 - What IP address does **ping hostname** yield?
 - What IP address does **nslookup hostname** yield?
 - Can system be accessed via UNC in a Windows Explorer (\\hostname)?
 - Are the systems all in the same subnet? If not, do they have normal IP address (that is, not one of the following 'unroutable' IP addresses):
 - 10.x.x.x
 - 172.16.x.x
 - 192.168.x.x
 - 224.0.0.0 (multicast-reserved)
 - Is the Windows Network Firewall enabled? If so, is it configured correctly? (See "Changing the Windows Firewall Settings" (in the online help))
- Login Issues**
- How are you logged-in? (Workgroup vs. Domain).
 - Are there different behaviors between workgroup and domain logins?
- Using with Multiframe**
- Is the agLogicSvc.exe service running on each system?
 - If you changed any network cables, you need to re-initialize the agLogicSvc.exe service by stopping it and restarting it or by rebooting. Has this been done?
 - All Gbit LAN cards on a Gbit LAN must use the same "Jumbo Frames" setting (see the LAN cards' advanced properties dialog). Also, if a switch/hub is used, you must make sure it supports the same size "Jumbo Frames" setting.
- Network Hardware and Configuration**
- Which OS is running? (Windows 7, Windows 8, or Windows Vista)
 - How many network cards? (Motherboard built-in 100Base-T, gigabit LAN)
 - From command prompt, run **ipconfig /all** for detailed configuration info.
 - From command prompt, run **ipconfig /release** to release current IP addresses.
 - From command prompt, run **ipconfig /renew** to re-obtain IP addresses.

- Network Topology
- On public LAN? On private LAN? Cross-over cable? Multiple networks?
 - Configurations:
 - Stand-alone
 - 1 network adapter - LAN.
 - 1 network adapter - cross-over cable.
 - 1 network adapter - private (switch or hub).
 - 2 network adapters - LAN + cross-over cable.
 - 2 network adapters - LAN + private (switch or hub).
- Known OS Issues
- Are the **SSDP** and **UPnP** services running? (SSDP = Manual, UPnP = Automatic).
 On some systems, the SSDP Discovery Service suddenly wakes up and starts using a lot of CPU time. This is a 'manual' service, which listens for a Universal Plug-n-Play request, and is thus started when a particular packet is received.
 When this happens, the SSDPSRW task starts, stops, and starts again in a loop.
 To disable the service, choose **My Computer>Manage>Services>SSDP Discovery Service>Properties** and set to **Disable**.
 Then, set the SSDP service back to manual.
 Failure to do this can cause the system logger to fill up and other nasty side effects.
 - Is the **Computer Browser** service running? If there is a DHCP server on the network, it may not be necessary for Computer Browser to be running. Furthermore, if the domain name server is NT 4.0, running computer browser on the network can lock out the 'real' domain server making it impossible for people on the network to log in reliably.
- Logic Analyzer Specific Issues
- Are there any *Keysight Logic Analyzer* application specific problems (application or service) being seen? (Connection error dialogs, crashes, other).
- Keep System Protected, Up-To-Date
- Use Windows Update to keep up-to-date on critical updates and service packs.
 - Keep the virus definitions up-to-date in your anti-virus software.
 - Install any *Keysight Logic Analyzer* application updates using their InstallShield packages.
 - The local Administrator password should not be empty.

Remote Desktop Set Up

If your logic analysis system has the Windows 7/8 operating system, it supports Remote Desktop Protocol (RDP) connections.

To enable Remote Desktop connections to the logic analysis system, see the Remote Desktop topics in the Windows 7/8 online help:

- 1 From the Windows Start menu, choose **Start>Help and Support**.
- 2 In the Help and Support Center window, enter "Remote Desktop" in the Search field; then, click the green arrow button to start the search.
- 3 Go to the topic on setting up the computer to use Remote Desktop, and follow its instructions.




When you set up a computer to use Remote Desktop, it is enabled as an exception in the operating system's firewall.

See Also · " Changing Firewall Settings, Windows 7 " (in the online help)





For More Information

Documentation

Quick Start/Installation Guide The *Quick Start/Installation Guide* gives you information on how to connect system peripherals and probing. Also included is an overview of the interface and information on installing software upgrades. Use this guide to quickly get familiar with the analyzer and also as a future reference for keeping your analyzer up-to-date and running properly.

-  ["AXIe based Logic and Protocol Analysis – Quick Start Guide"](#)
-  ["AXIe based Logic and Protocol Analysis – Installation Guide"](#)
-  ["16850 Series Portable Logic Analyzers Installation/Quick Start Guide"](#)

Probing Documentation For more information on general-purpose probing, QFP package probing, target connector and connectorless probing, and other probing options, see:

-  ["Probing Selection Quick Reference Card"](#)
-  ["Probing Solutions for Logic Analyzers"](#) ( ["latest version on web"](#))
-  ["Logic Analyzer Probing Solutions"](#)

For more information on analysis probes and other processor and bus solutions, see:

-  ["Processor and Bus Support for Logic Analyzers"](#) ( ["latest version on web"](#))
-  ["Processor, Bus, and FPGA Support for Logic Analyzers"](#)

Online Help System The online help gives you product reference and feature information. Also included is a tutorial (see [page 40](#)) showing you how to make a basic measurement and containing links to time-saving features and concepts.

Keysight Technologies Web Sites

Corporation/Contact

- Corporation - ["http://www.keysight.com"](http://www.keysight.com)
- Contact Us - ["http://www.keysight.com/find/contactus"](http://www.keysight.com/find/contactus)
- Email Updates - ["http://www.keysight.com/find/emailupdates"](http://www.keysight.com/find/emailupdates)

Product Information

- Logic Analysis - ["http://www.keysight.com/find/logic"](http://www.keysight.com/find/logic)
- Logic Analysis Software Download - ["http://www.keysight.com/find/la-sw-download"](http://www.keysight.com/find/la-sw-download)
- Software on CD - ["http://software.cos.keysight.com/LogicAnalyzerSW"](http://software.cos.keysight.com/LogicAnalyzerSW)

See Also

- Tutorial - Getting to know your logic analyzer (see [page 40](#))

12 Concepts

- Logic Analysis Basics
- When should you use an oscilloscope? (see [page 354](#))
 - When should you use a logic analyzer? (see [page 355](#))
 - What is a logic analyzer? (see [page 356](#))

Timing analyzer:

- Sampling clock (see [page 356](#))
- Sampling (see [page 356](#))
- Triggering (see [page 357](#))

State analyzer:

- Sampling clock (see [page 358](#))
- Sampling (see [page 359](#))
- Triggering (see [page 359](#))

- Other Logic Analysis Concepts
- Pod and Channel Naming Conventions (see [page 361](#))
 - Why Are Pods Missing? (see [page 362](#))
 - Memory Depth and Channel Count Trade-offs (see [page 363](#))
 - Transitional Timing (see [page 365](#))
 - Understanding State Mode Sampling Positions (see [page 367](#))
 - Understanding Logic Analyzer Triggering (see [page 369](#))
 - ALA vs. XML, When to Use Each Format (see [page 380](#))
 - Multiframe Logic Analysis Systems (see [page 381](#))

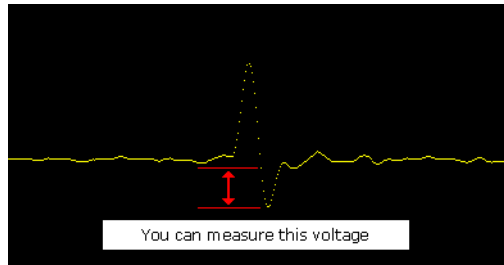
When should you use an oscilloscope?

[[Tutorial Home](#) (see [page 40](#))] [[Next Topic](#) (see [page 355](#))] [[Previous Topic](#) (see [page 40](#))]

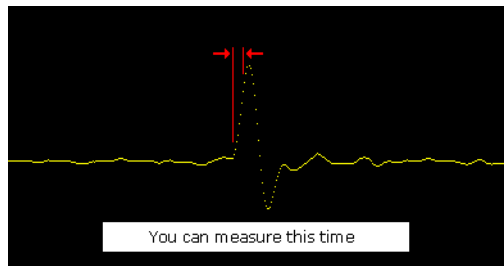
Generally, an oscilloscope is used when you need precise parametric information such as time intervals and voltage readings.

More specifically:

- When you need to measure small voltage excursions on your signals such as undershoot or overshoot.



- When you need high time-interval accuracy. Oscilloscopes can capture precise parametric information such as the time between two points on a rising edge of a pulse with very high accuracy.



When should you use a logic analyzer?

[[Tutorial Home](#) (see [page 40](#))] [[Next Topic](#) (see [page 356](#))] [[Previous Topic](#) (see [page 354](#))]

Generally, a logic analyzer is used to view timing relationships among many signals, or if you need to trigger on patterns of logic highs and lows. A logic analyzer reacts the same way as the logic circuits do when a voltage threshold is crossed by a signal in the device under test. It will recognize the signal to be either low or high.

More specifically:

- When you need to see many signals at once.
Logic analyzers are very good at organizing and displaying multiple signals. A common task is to group multiple signals into a bus and assign a custom name. Good examples are address, data, and control buses.
- When you need to look at signals in your system the same way your hardware does.
Signals are displayed on a time axis so you can see when transitions occur relative to other bus signals or clock signals.
- When you need to trigger on a unique bus pattern or signal edge.
Logic analyzers can be configured to store data when the high or low values of a group (bus) of signals match a predefined pattern.
Logic analyzers can be configured to store data when a specific edge or level is detected on a single signal.

What is a logic analyzer?

[[Tutorial Home](#) (see [page 40](#))] [[Next Topic](#) (see [page 356](#))] [[Previous Topic](#) (see [page 355](#))]

Now that we've talked a little about when to use a logic analyzer, let's look in more detail at what a logic analyzer is. Up to now, we've used the term "logic analyzer" rather loosely. In fact, most logic analyzers are really two analyzers in one.

What is a timing analyzer?

A timing analyzer is the part of a logic analyzer that is analogous to an oscilloscope. As a matter of fact, they can be thought of as close cousins.

The timing analyzer displays information in the same general form as an oscilloscope, with the horizontal axis representing time and the vertical axis as voltage amplitude. Because the waveforms on both instruments are time-dependent, the displays are said to be in the "time domain".

The basic areas of functionality in a timing analyzer are as follows:

- Sampling clock in the timing analyzer (see [page 356](#))
- Sampling in the timing acquisition mode (see [page 356](#))
- Triggering the timing analyzer (see [page 357](#))

What is a state analyzer?

A state analyzer is very good at tracking down bugs in software or defective components in hardware. It can help eliminate the question whether a problem is in the software code or some hardware device.

Most often, state analyzers are used to find out what logic levels are present on a bus when a particular clock signal occurs. In other words, you want to know what "state of activity" is present when the clock occurs and data is suppose to be valid. Data captured in memory is displayed in a listing format with a time tag attached to every state.

The basic areas of functionality in a state analyzer are as follows:

- Sampling clock in the state analyzer (see [page 358](#))
- Sampling in the state acquisition mode (see [page 359](#))
- Triggering the state analyzer (see [page 359](#))

Sampling clock in the timing analyzer

[[Tutorial Home](#) (see [page 40](#))] [[Next Topic](#) (see [page 356](#))] [[Previous Topic](#) (see [page 356](#))]

The timing analyzer uses its own internal clock to control the sampling of data. This type of clocking makes the sampling of data in the logic analyzer *asynchronous* to the clocking in the device under test.

More specifically:

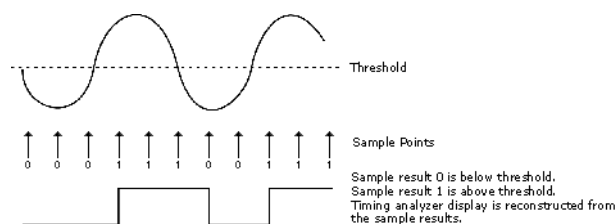
- A timing analyzer is good at showing you "When" signal activity occurs "Relative to other signals".
- A timing analyzer is more interested in viewing the timing relationships between individual signals, than the timing relationships to the signals that are controlling execution in the device under test.
- This is why a timing analyzer can sample data "out of sync", or asynchronous to the device under test clock signals.

Sampling in the timing acquisition mode

[[Tutorial Home](#) (see [page 40](#))] [[Next Topic](#) (see [page 357](#))] [[Previous Topic](#) (see [page 356](#))]

In the timing acquisition mode, the logic analyzer works by sampling the input waveforms to determine whether they are high or low. It determines a high or low by comparing the voltage level of the incoming signal to a user-defined voltage threshold. If the signal is above that threshold when it samples, it will be displayed as a 1 or high by the analyzer. By the same criterion, any signal sampled that is below threshold is displayed as a 0 or low.

The figure below illustrates how a logic analyzer samples a sine wave as it crosses the threshold level.

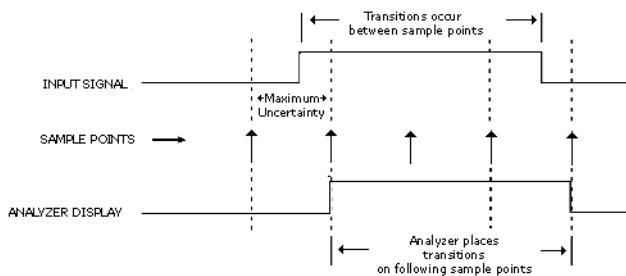


The sample points are then stored in memory and used to reconstruct a more squared-off digital waveform.

This tendency to square everything up would seem to limit the usefulness of a timing analyzer. However, a timing analyzer is not intended as a parametric instrument. If you want to check rise time of a signal, use an oscilloscope. If you need to verify timing relationships among several or hundreds of signals by seeing them all together, a timing analyzer is the right choice.

Sampling accuracy

When the timing analyzer samples an input channel, it is either high or low. If the channel is at one state (high or low) on one sample, and the opposite state on the next sample, the analyzer "knows" that the input signal has transitioned sometime between the two samples. It doesn't know when, so it places the transition point at the next sample, as shown in the figure below.



This presents some ambiguity as to when the transition actually occurred and when it is displayed by the analyzer.

Worst case for this ambiguity is one sample period, assuming that the transition occurred immediately after the previous sample point.

With this technique however, there is a trade-off between resolution and total acquisition time. Remember that every sampling point uses one memory location. Thus, the higher the resolution (faster sampling rate), the shorter the acquisition window.

Triggering the timing analyzer

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 358](#))] [Previous Topic (see [page 356](#))]

At some point in a measurement, the logic analyzer has to know when to capture (store) the data that is flowing through its memory. This is known as the trigger point.

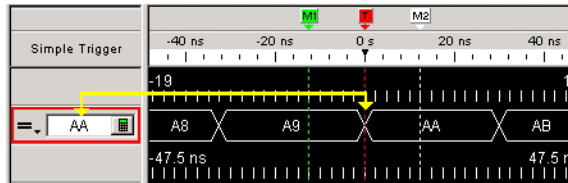
One way to get the analyzer to trigger is to configure the analyzer to look for either a pattern of highs and lows from a group of signals (bus), or a rising or falling edge from a single signal. When the analyzer sees the specified patterns or edges in data, it triggers.

Pattern Trigger

Pattern triggers are used to find specific patterns of highs and lows across a bus. You can specify different kinds of criteria such as equal, not equal, in or out of a range, or greater than/less than.

NOTE

Example: You have a bus containing 8 signal lines. You configure the *Simple Trigger* to specify that the analyzer triggers when the incoming data is equal to a pattern of "AA".



To make things easier for some users, the trigger point on most analyzers can be set not only in Hex, but in binary (1's and 0's), octal, ASCII, or decimal. For instance, the Hex trigger value of AA could also be set to an equivalent binary trigger value of 1010 1010. However, using hex for the trigger point is particularly helpful when looking at buses that are 16, 24, 32, or 64 bits wide.

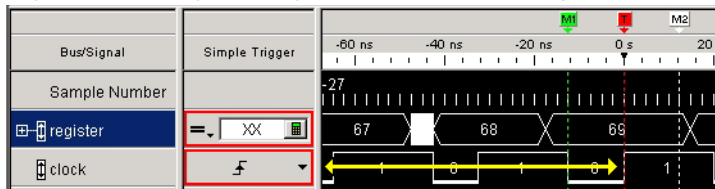
Edge Trigger

Edge triggering is a familiar concept to those accustomed to using an oscilloscope. When adjusting the "trigger level" knob on an oscilloscope, you could think of it as setting the level of a voltage comparator that tells the oscilloscope to trigger when the input voltage crosses that level. A timing analyzer works essentially the same on edge triggering except that the trigger level is preset to a logic threshold.

While many logic devices are level dependent, clock and control signals of these devices are often edge-sensitive. Edge triggering allows you to start capturing data as the device is clocked.

NOTE

Example: Take the case of an edge-triggered shift register that is not shifting data correctly. Is the problem with the data or the clock edge? In order to check the device, we need to verify the data when it is clocked – on the clock edge. The analyzer can be told to capture data when the clock edge occurs (rising or falling) and catch all of the outputs of the shift register.



Sampling clock in the state analyzer

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 359](#))] [Previous Topic (see [page 357](#))]

The state analyzer requires a sampling clock signal from the device under test. This type of clocking makes the sampling of data in the logic analyzer *synchronous* to the clocked events on the device under test.

More specifically:

- A state analyzer is good at showing you "What" the signal activity is during a "Valid clock or control signal".

- A state analyzer is more interested in viewing signal activity during specified times of execution, than signal activity unrelated to the timing.
- This is why a state analyzer wants to sample data that is "synchronized" or synchronous to the device under test clock signals.

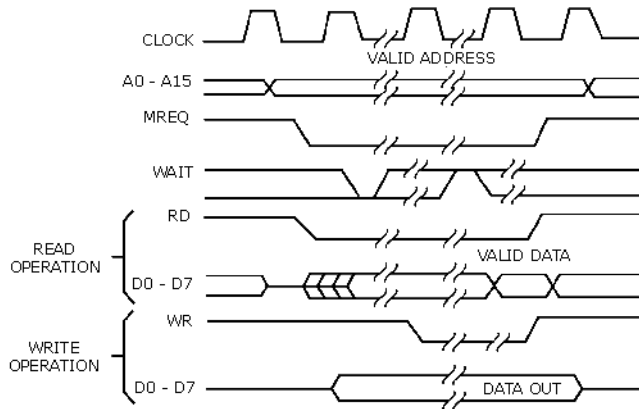
Sampling in the state acquisition mode

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 359](#))] [Previous Topic (see [page 358](#))]

In the world of microprocessors, you can have both data and address appearing on the same signal lines. To capture the correct data, the logic analyzer has to restrict the sampling of data to times when only the desired data is valid and appears on the signal lines. It does this by sampling data from the same signal lines but with different sampling clocks from the device under test.

NOTE

Example: The following timing diagram shows that to capture addresses, we want the analyzer to sample when MREQ line goes low. To capture data, we want the analyzer to sample when the WR line goes low (write cycle) or when RD goes low (read cycle).



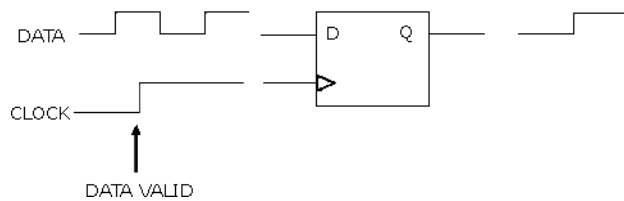
Triggering the state analyzer

[Tutorial Home (see [page 40](#))] [Next Topic (see [page 40](#))] [Previous Topic (see [page 359](#))]

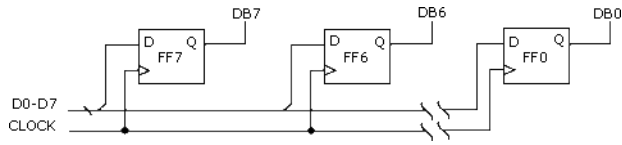
Similar to a timing analyzer, a state analyzer has the capability to qualify the data we want to store. If we are looking for a specific pattern of highs and lows on the address bus, we can tell the analyzer to start storing when it finds that pattern and to continue storing until the analyzer's memory is full.

Simple Trigger Example

Looking at the "D" flip-flop shown below, data on the "D" input is not valid until after a positive-going clock edge occurs. Thus, a valid state for the flip-flop is when the clock input is high.



Now imagine that we have eight of these flip-flops in parallel. All eight are connected to the same clock signal as shown below.



When a high level occurs on the clock line, all eight capture data at their "D" inputs. Again, a valid state occurs each time there is a positive level on the clock line.

The following simple trigger tells the analyzer to collect data on lines D0 - D7 when a high level is on the clock line.

Sample Number	D0-D7	Clock
	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>
-10	00DB	1
-9	00DF	1
-8	0062	0
-7	0065	1

Advanced Trigger Example

You want to see what data is stored in memory at the address value 406F6. You configure the advanced trigger to look for the pattern 406F6 (hexadecimal) on the address bus and a high level on the RD (memory read) clock line.

Step 1 Edge and Pattern

Find

And

Then

As you configure the Edge And Pattern trigger dialog, try to think of it as constructing a sentence that reads left-to-right.

"Find the first occurrence of a **Bus** named **ADDR**, and on **All bits** a pattern that **Equals 406F6 Hex**, And a **Signal** named **RD** with a **High** level. Then **Trigger and fill memory** with **Anything**.

Pod and Channel Naming Conventions

In 16850-series logic analysis systems:

- Slots are named "A" through "F" starting with the top slot.
- There is a cable marked "Pod 2" connected to every logic analyzer card. It is important to know which slot a pod is connected to because if you have logic analyzer cards in slots A and B, there will be two pod cables labeled "Pod 2", but the *Keysight Logic Analyzer* application will refer to one as "Slot A Pod 2" and the other as "Slot B Pod 2". It's important not to mix up the two cables.
- Slot A Pod 2 is the same as "Pod A2". "A2" is used interchangeably with "Slot A Pod 2"; likewise, "D1" is used interchangeably with "Slot D Pod 1".

See Also • Pod and Channel Naming Conventions in U4154A/B Logic Analyzer (see [page 561](#))

Why Are Pods Missing?

There are a number of reasons all pods are not available to a logic analyzer module:

- In the state sampling mode, with the *General State Mode* sampling option selected, choosing the maximum acquisition memory depth requires one pod pair to be reserved for time tag storage. In this case, setting the memory depth to half of the maximum (or less) will return the pods.
- In the state sampling mode, with the *Turbo State Mode* (see [page 536](#)) sampling option selected, one pod pair is reserved for time tag storage.
- In the timing sampling mode, with the *Transitional / Store Qualified Timing Mode* (see [page 536](#)) sampling option selected:
 - When the smallest sampling period is selected, one pod pair is reserved for time tag storage.
 - When sampling periods other than the smallest are selected, choosing the maximum acquisition memory depth requires one pod pair to be reserved for time tag storage. In this case, setting the memory depth to half of the maximum (or less) will return the pods.

- See Also
- Memory Depth and Channel Count Trade-offs (see [page 363](#))
 - Configuring Logic Analyzer Modules (see [page 72](#))

Memory Depth and Channel Count Trade-offs

This topic describes the interaction between channel count, memory depth, and triggering in the:

- State Sampling Mode (see [page 363](#))
- Transitional Timing Sampling Mode (see [page 363](#))

State Sampling Mode

Time Tag Storage Requires 1 Pod Pair or 1/2 Acquisition Memory

- In the *Keysight Logic Analyzer* application, all modules are time-correlated; you cannot turn off *time tag storage* (as you could with previous Keysight logic analysis systems).
- To use more than 1/2 of a module's acquisition memory, one *pod pair* must be *reserved for time tag storage*. To use all pod pairs, you must use 1/2 (or less) of a module's acquisition memory.
- In general, the number of timers available = the number of pod pairs not reserved for time tag storage (refer to your logic analyzer characteristics (see [page 587](#)) for the actual number of timers available).

Default Settings

- Time tag storage is always on (and cannot be turned off).
- Memory depth is set at 1/2 of the total acquisition memory.
- All pod pairs are available for capturing data.
- If full memory is selected, the default pod pair to be used for time tag storage is the leftmost, but any pod pair without buses or signals assigned can be used.

Selecting Full Memory Depth when No Channels Assigned to a Pod Pair

- The pod pair is automatically reserved for time tag storage.

Selecting Full Memory Depth when Channels Assigned to All Pod Pairs

A dialog appears to caution you that:

- Bus/signals will lose assigned channels.
- Trigger specifications that use timer resources may be affected.

Going from Full Memory Depth to Half Memory Depth

- The pod pair reserved for time tag storage is automatically freed (assigned to the logic analyzer) so it can be used to capture data.

Transitional Timing Sampling Mode

Time Tag Storage Requires 1 Pod Pair or 1/2 Acquisition Memory

- The transitional timing sampling mode also requires *time tag storage*.
- When the smallest sampling period is chosen, one *pod pair* must be *reserved for time tag storage*. In this case, you cannot use 1/2 (or less) of a module's acquisition memory to gain back the *pod pair*.
- With other sampling periods, the memory depth and channel count trade-offs are the same as in the state sampling mode. That is, to use more than 1/2 of a module's acquisition memory, one *pod pair* must be *reserved for time tag storage*. To use all pod pairs, you must use 1/2 (or less) of a module's acquisition memory.

- In general, the number of timers available = the number of pod pairs not reserved for time tag storage (refer to your logic analyzer characteristics (see [page 587](#)) for the actual number of timers available).

Default Settings

- Time tag storage is required.
- If full memory is selected, the default pod pair to be used for time tag storage is the leftmost, but any pod pair without buses or signals assigned can be used.

See Also

- [Configuring Logic Analyzer Modules \(see page 72\)](#)
- [To set acquisition memory depth \(see page 129\)](#)
- [Choosing the Sampling Mode \(see page 96\)](#)
- [Logic Analyzer Notes, Channels and Memory Depth \(see page 536\)](#)

Transitional Timing

In the **Transitional / Store qualified** timing mode, the timing analyzer samples data at regular intervals, but only stores data when there is a signal transition across the threshold voltage level. Each time a transition occurs on any of the bits in the defined buses/signals (that haven't been excluded), data on all channels is stored. A *time tag* is stored with each stored data sample so the measurement can be reconstructed and displayed later.

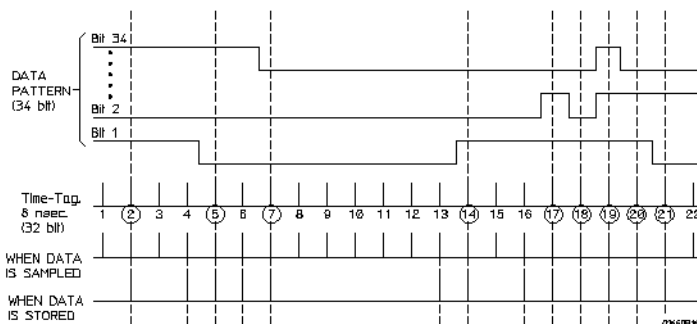
More on Storing Transitions

Minimum Transitions Stored

Normally, transitions do not occur at each sample point. This is illustrated below with time tags 2, 5, 7, and 14. When transitions do occur, two samples are stored for every transition. Therefore, with 2K samples of memory, 1K transitions are stored. You must subtract one, which is necessary for a starting point, for a minimum of 1023 stored transitions.

Maximum Transitions Stored

If transitions occur at a fast rate, such that there is a transition at each sample point, only one sample is stored for each transition as shown by time tags 17 through 21 below. If this continues for the entire trace, the number of transitions stored is 2K samples. Again, you must subtract the starting point sample, which then yields a maximum of 2047 stored transitions.



In most cases a transitional timing trace is stored by a mixture of the minimum and maximum cases. Therefore, in this example the actual number of transitions stored will be between 1023 and 2047.

Transitional Timing Considerations

Data Storage

When an edge is detected, two samples are stored across all channels assigned to the timing analyzer. Two samples are needed to avoid data loss if a second edge occurs (after the first edge) before the edge detectors can reset.

Sequence Step Branching

In transitional timing, only 2 branches are available per sequence step.

Global Counters

In transitional timing, only one global counter is available.

Storing Time Tags

Transitional timing requires time tags to recreate the data. Time tags are stored by interleaving them with measurement data in memory.

Increasing Duration of Storage (Amount of Time Measured)	By default, the analyzer looks for transitions on all buses/signals defined for the logic analyzer module. However, to increase usable memory depth and acquisition time, you can, in the Advanced Trigger dialog (see page 424), exclude certain bus/signal transitions from being stored (like clock or strobe signal transitions that add little useful information to the measurement).
Data on Unassigned Channels	<p>When you run a measurement, data is captured on all logic analyzer channels, whether buses/signals are defined and assigned to those channels or not. In the transitional timing mode, captured samples are saved if there are transitions on the <i>defined</i> buses/signals (that haven't been excluded).</p> <p>After a transitional timing measurement has been run, if you define new buses/signals for previously unassigned logic analyzer channels, the data captured on those channels appears, but it is unlikely that all transitions on those buses/signals have been stored; the data that appears is as if the new buses/signals had been excluded before the measurement was run.</p>
Trigger Position	In transitional timing, no data prestore (samples acquired before trigger) is required. Therefore, much like state mode, the trigger position (start/center/end) indicates the percentage of memory filled with samples after the trigger. The number of samples acquired/displayed before the trigger will vary between measurements.

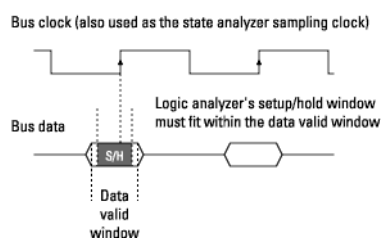
Understanding State Mode Sampling Positions

Synchronous sampling (state mode) logic analyzers are like edge-triggered flip-flops in that they require input logic signals to be stable for a period of time before the clock event (setup time) and after the clock event (hold time) in order to properly interpret the logic level. The combined setup and hold time is known as the setup/hold window.

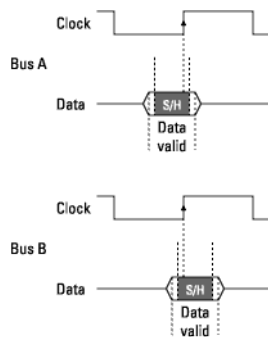
A device under test (because of its own setup/hold requirements) specifies that data be valid on a bus for a certain length of time. This is known as the data valid window. The data valid window on most buses is generally less than half of the bus clock period.

To accurately capture data on a bus:

- The logic analyzer's setup/hold time must fit within the data valid window.



- Because the location of the data valid window relative to the bus clock is different for different types of buses, the position of the logic analyzer's setup/hold window must be adjustable (relative to the sampling clock, and with fine resolution) within the data valid window. For example:



To position the setup/hold window (sampling position) within the data valid window, a logic analyzer has an adjustable delay on each sampling input (to position the setup/hold window for each channel).

Sample Position Adjustments on Individual Channels

When you can make sampling position adjustments on individual channels, you can make the logic analyzer's setup/hold window smaller because you can correct for the skew effects caused by the probe cables and the logic analyzer's internal circuit board traces, and you are left with the setup/hold requirements of the logic analyzer's internal sampling circuitry.

However, the process of manually positioning the setup/hold window for each channel is time consuming. For each signal in the device under test and each logic analyzer channel, you must measure the data valid window in relation to the bus clock (with an oscilloscope), repeatedly position the setup/hold window and run measurements to see if the logic analyzer captures data correctly, and finally position the setup/hold window in between the positions where data was captured incorrectly.

With Keysight Technologies logic analyzers that have the eyescan feature, in a small fraction of the time that it takes to make the adjustments manually (and without the extra test equipment), you can automatically:

- Position the setup/hold window on each channel.
- Adjust the threshold voltage setting for the widest possible data valid window.

Eyescan is an easy way to get the smallest possible logic analyzer setup/hold window.

NOTE

You use the Eyescan feature with the U4154A/B Logic Analyzer to automatically adjust sample positions on individual channels. Refer to the topic [“Setting up and Running Eyescans in Logic Analyzers”](#) on page 567 to know more about the eyescan feature.

- See Also
- To automatically adjust state sampling positions and threshold voltages (see [page 128](#))
 - To manually adjust state sampling positions (see [page 128](#))
 - Selecting the State Mode (Synchronous Sampling) (see [page 98](#))
 - [“Setting up and Running Eyescans in Logic Analyzers”](#) on page 567

Understanding Logic Analyzer Triggering

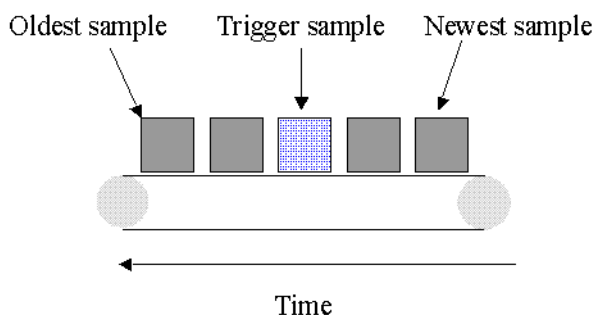
Setting up logic analyzer triggers can be difficult and time-consuming. You could assume that if you know how to program, you should be able to set up a logic analyzer trigger with no difficulty. However, this is not true because there are many concepts that are unique to logic analysis. The purpose of this section is to describe these key concepts and how to use them effectively.

- The Conveyor Belt Analogy (see [page 369](#))
- Summary of Triggering Capabilities (see [page 370](#))
- Sequence Steps (see [page 371](#))
- Boolean Expressions (see [page 372](#))
- Branches (see [page 373](#))
- Edges (see [page 373](#))
- Ranges (see [page 373](#))
- Flags (see [page 373](#))
- Counters (see [page 373](#))
- Timers (see [page 374](#))
- Storage Qualification (see [page 375](#))
- Strategies for Setting Up Triggers (see [page 376](#))
- Conclusions (see [page 379](#))

See Also • [Capturing Data from the Device Under Test \(see page 141\)](#)

The Conveyor Belt Analogy

The memory of a logic analyzer can be compared to a very long conveyor belt, and the samples acquired from the device under test (DUT) as boxes on the conveyor belt. At one end, new boxes are placed on the conveyor belt, and at the other end the boxes fall off. In other words, because logic analyzer memory is limited in depth (number of samples), whenever a new sample is acquired the oldest sample currently in memory is thrown away if the memory is full. This is shown in the following figure.



The conveyor belt analogy

A logic analyzer trigger is similar to someone standing at the beginning of the conveyor belt placing more boxes on it. They are told to "look for a special box and to stop the conveyor belt when that box reaches a particular position on the belt". Using this analogy, the special box is the trigger. Once a logic analyzer detects a sample that matches the trigger condition, this is the indication that it should stop acquiring more samples when the trigger is located appropriately in memory.

The location of the trigger in memory is known as the *trigger position*. Normally, the trigger position is set to the middle so that the maximum number of samples that occurred before and after the trigger are in memory. However, you can set the trigger position to any point in memory.

The concepts in this analogy are summed up in the following table.

Mapping of concepts in the Conveyor Belt Analogy to a Logic Analyzer

Conveyor Belt Analogy	Logic Analyzer
Boxes on the belt	Samples acquired from the device under test
Number of boxes that will fit on the belt	Memory depth
Special box	Trigger point

Next: Summary of Triggering Capabilities (see [page 370](#))

Summary of Triggering Capabilities

Because logic analyzer triggering provides a great deal of functionality, the following table provides a brief summary of the capabilities covered in this article. Each of these capabilities will be described.

Summary of Logic Analyzer Triggering Capabilities

Capability	Examples
Edges	<pre>If there is rising edge on SIG1 then Trigger If there is falling edge on SIG1 then Trigger</pre>
Boolean expressions	<pre>If ADDR = 1000 and DATA = 2000</pre>
Ranges	<pre>If ADDR in range 1000 to 2000</pre>
Storage qualification	<pre>1. If.. Else If ADDR in range 1000 to 2000 then Store Sample Go to 1 Else If ADDR not in range 1000 to 2000 then Don't Store Sample Go to 1</pre>
Counters	<pre>1. If DATA = 1000 Then Increment Counter 1 Go to 2 2. If Counter 1 > 2 Then Trigger</pre>
Timers	<pre>1. If DATA = 1000 Then Start Timer 1 Go to 2 2. If Timer 1 > 500 ns Then Trigger</pre>

Next: Sequence Steps (see [page 371](#))

Sequence Steps

While logic analyzer triggers are often simple, they can require complex programming. For example, you may want to trigger on the rising edge of one signal that is followed by the rising edge of another signal. This means that the logic analyzer must first find the first rising edge before it begins looking for the next rising edge. Because there is a sequence of steps to find the trigger, this is known as a *trigger sequence*. Each step of the sequence is called a *sequence step*.

Each sequence step consists of two parts; the conditions and the actions. The conditions are Boolean expressions such as "If ADDR = 1000" or "If there is a rising edge on SIG1". The actions are what the logic analyzer should do if the condition is met. Examples of actions include triggering the logic analyzer, going to another sequence step, or starting a timer. This is similar to an If/Then statement in programming.

Each step in the trigger sequence is assigned a number. The first sequence step to be executed is always Sequence Step 1, but because of the Go To actions, the rest of the sequence steps can be executed in any order.

When a sequence step is executed and none of the Boolean expressions are true, the logic analyzer acquires the next sample and executes the same sequence step again. As a simple example, consider the following trigger sequence:

1. If DATA = 7000 then Trigger

If the following samples were acquired, the logic analyzer would trigger on sample #6.

Sample #	ADDR	DATA	
1	1000	2000	
2	1010	3000	
3	1020	4000	
4	1030	5000	
5	1040	6000	
6	1050	7000	<- This is where the logic analyzer triggers
7	1060	2000	

In essence, Sequence Step 1 is equivalent to "Keep acquiring more samples until DATA=7000, then trigger".

If a Boolean expression in a sequence step is met, another sample is always acquired before the next sequence step is executed. In other words, if a sample meets the condition in Sequence Step 1, another sample will be acquired before executing Sequence Step 2. This means that it is not possible for a single sample to be used to meet the conditions of more than one sequence step. Each sequence step can be thought of as representing events that occur at different points in time. Two sequence steps can never be used to specify two events that happen simultaneously.

For example, consider the following trigger sequence:

1. If ADDR = 1000 then Go to 2
2. If DATA = 2000 then Trigger

If the following samples were acquired, the logic analyzer would trigger on sample #7.

Sample #	ADDR	DATA	
1	1000	2000	<- This meets the condition in Sequence step #1
2	1010	3000	
3	1020	4000	
4	1030	5000	
5	1040	6000	
6	1050	7000	
7	1060	2000	<- This is where the logic analyzer triggers

Note that the logic analyzer will not trigger on Sample #1 because a new sample is acquired between the time that the condition in Sequence Step 1 is met and when the condition in Sequence Step #2 is tested. A good way to think of this trigger sequence is "Find ADDR = 1000 followed by DATA = 2000 and then trigger". Multiple sequence steps in a trigger sequence imply a "followed by".

Once a logic analyzer triggers, it does not trigger again. In other words, even if more than one sample meets the trigger condition, the logic analyzer still only triggers once. For example, using "ADDR=1000" as our trigger, if the logic analyzer acquires the following samples, it will trigger on Sample #2 and only on Sample #2.

Sample #	ADDR	
1	0000	
2	1000	<- The logic analyzer triggers here
3	2000	
4	1000	<- The logic analyzer does NOT trigger again here
5	1040	

A frequently asked question is "What happens if the conditions in a sequence step are not met?" For example, if there is a condition that says "If ADDR = 1000 Then Trigger", what happens if the current sample has ADDR = 2000? The logic analyzer simply acquires the next sample and tries to execute this sequence step again. In essence, if the trigger condition is "ADDR = 1000", this is equivalent to "Keep acquiring more samples until you find one that has ADDR=1000". Therefore, if you set up a trigger condition that is never met, the logic analyzer will never trigger.

When the conditions are met in a sequence step, it is clear which sequence step will be executed next when a "Go To" action is used, but it is not necessarily clear if there is no "Go To". On some logic analyzers, if there is no "Go To", this means that the next sequence step should be executed. On other logic analyzers, it means the same sequence step should be executed again. Because of this confusion, it is good practice to always use a "Go To" action rather than relying on the default. The state and timing modules deal with this problem by automatically including a "Go To" or "Trigger" action in every sequence step. For example:

```
If ADDR = 1000 and DATA = 2000 then
Go to 1  <- This is automatically added
```

Next: Boolean Expressions (see [page 372](#))

Boolean Expressions

While multiple sequence steps imply a "followed by", within a sequence step Boolean expressions can be used. An example is:

```
If ADDR = 1000 and DATA = 2000
```

This expression means that for this expression to be met, ADDR must equal 1000 in the same sample that DATA equals 2000. In other words, ADDR equals 1000 at the same time that DATA equals 2000. Therefore, if you want to trigger on two events that occur at the same time, a Boolean expression should be used.

It's a common mistake to try to use two sequence steps when a Boolean expression should be used or to use a Boolean expression when two sequence steps should be used.

NOTE

Boolean expressions are used for events that happen at the same time, and multiple sequence steps are used when one event follows another.

Next: Branches (see [page 373](#))

Branches

Branches are similar to the *Switch* statement in the C programming language and the *Select Case* statement in Basic. They provide a method for testing multiple conditions. Each branch has its own actions. An example of multiple branches is shown below:

```

1. If ADDR < 1000 then Go To 2           <- This is a branch of Level 1
   Else If ADDR > 2000 then Go To 3     <- This is a 2nd branch of Level 1
   Else If DATA = 2000 then Trigger   <- This is a 3rd branch of Level 1
2. If DATA <= 7000 then Trigger
3. If there is a Rising Edge on SIG1, then Trigger

```

In sequence step 1, there are three branches, so there are three possible actions that can be taken.

When the condition of one branch is met, none of the branches below it are tested. In other words, there is no way for more than one branch to be executed based upon a single sample, even if the sample causes the conditions for more than one branch to be met. In other words, each branch is an "Else If".

Next: Edges (see [page 373](#))

Edges

Edges represent a transition from low to high or high to low on a single signal. Typically, edges are specified as "rising edge", "falling edge", or "either edge", where "rising edge" indicates a transition from a low to a high. On most logic analyzers, up to two edges can be included in the trigger sequence although some allow only one.

Next: Ranges (see [page 373](#))

Ranges

Ranges are a convenient method for specifying a range of values, such as "ADDR in range 1000 to 2000". Most logic analyzers also support a "not in range" function as well. Ranges are a convenient shortcut so that you don't have to specify "ADDR >= 1000 and ADDR <= 2000".

Next: Flags (see [page 373](#))

Flags

Flags are Boolean variables that are used to send signals from one module to another. They can be set when a condition occurs in one module and tested later by another module. In the example below, flag 1 is used to keep track of what happens in the trigger sequence of Module 1 so that this information can be used in Module 2.

Trigger Sequence for Module 1:

```

1. If ADDR < 5000 then
   Set Flag 1
   Trigger and fill memory

```

Trigger Sequence for Module 2:

```

1. If DATA = 5000 and Flag 1 is set then Trigger
   Else if DATA = 1000 and not Flag 1 then Trigger

```

Next: Counters (see [page 373](#))

Counters

Occurrence Counters are used in situations where you want to find the Nth occurrence of an event. For example, if you want to trigger on the 5th time that ADDR = 1000, you could set up the trigger as:

```
If ADDR = 1000 occurs 5 times then Trigger
```

Global Counters are like integer variables. They are more flexible than Occurrence Counters because they can be used to count complex events such as an edge followed by another edge. Global Counters can be incremented, tested, and reset. By default, Global Counters begin with zero and don't need to be reset unless they have already been used in the trigger sequence. In general, Occurrence Counters should be used in place of Global Counters, if possible, because they are easier to use and because there is a limited number of Global Counters.

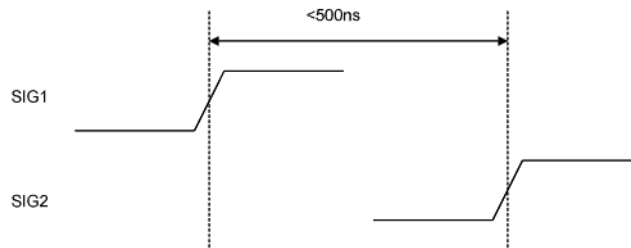
Event counters are similar to global counters in that these are like integer variables used to count events and can be incremented or reset on the occurrence of the specified event. However, these counters cannot be used in an "If" clause to test a condition to be true or false.

Next: Timers (see [page 374](#))

Timers

Timers are used to check the amount of time that has elapsed between events. For example, if you want to trigger on one edge followed by another edge that occurs within 500 ns, use a timer. The most critical point to remember in using timers is that they need to be started before they are tested. In other words, timers do not start automatically.

The key to setting up a timer is to identify where it should be started and where it should be tested. Consider the example in the following figure. The timer should be started when the rising edge on SIG1 is detected and it should be tested when the rising edge occurs on SIG2.



An edge followed by an edge with a time limit

An example trigger sequence to set up this measurement is:

1. If there is a Rising Edge on SIG1, then
 Start Timer1
 Go to 2
2. If there is a Rising Edge on SIG2 AND Timer1 < 500ns then
 Trigger

While the above trigger sequence seems correct, it actually has a critical flaw. What happens if there is a rising edge on SIG1 but SIG2 doesn't occur within 500 ns? The logic analyzer will never trigger, because timer1 will keep running and condition "Timer1 < 500 ns" will never be met. There might be another rising edge on SIG1 that is followed within 500 ns by the rising edge on SIG2 that occurs later on, so this situation is unacceptable.

To fix this problem, whenever the timer exceeds 500 ns without triggering, the sequence should loop back to Level 1 to look for another rising edge on SIG1. The following shows an example of the correct sequence:

1. If there is a Rising Edge on SIG1, then
 - Start Timer1
 - Go to 2
2. If there is a Rising Edge on SIG2 AND Timer1 < 500ns then
 - Trigger
 - Else If Timer1 >= 500ns then
 - Reset Timer1
 - Go to 1

Occasionally, you may run out of timers. A counter can be used in place of a timer if the logic analyzer is sampling at regular intervals (that is, if it's in the timing sampling mode). A timer can be simulated by counting the number of samples that are acquired. For example, if the logic analyzer acquires a new sample every 10 ns and seven samples are acquired, this represents 70 ns.

Next: Storage Qualification (see [page 375](#))

Storage Qualification

Storage qualification is used to determine if an acquired sample should be stored (that is, placed in memory) or thrown away. This keeps the logic analyzer memory from being filled with samples that are not needed.

Default Storage The simplest method to set up storage qualification is by setting up the Default Storage. Default Storage means "unless a sequence step specifies otherwise, this is what should be stored". As an example, you may want to only store samples if ADDR is in the range 1000 to 2000, so you should set the Default Storage to:

```
ADDR In Range 1000 to 2000
```

By default, the Default Storage is set to store all samples acquired. You can also set the Default Storage to store nothing, which means that no samples will be stored unless a sequence step overrides the default storage.

Sequence Step Storage Sequence step storage qualification means that within a particular sequence step only certain samples will be stored. This means that until a "Go To" or "Trigger" action is used to leave this sequence step, the storage qualification applies. This is useful when you want different storage qualification for each sequence step. For example, you may want to store nothing until ADDR = 1000 and then store only samples with ADDR in the range 1000 to 2000 for the rest of the measurement.

Setting up sequence step storage requires the use of an additional branch. For example, if you want to store only samples with ADDR in the range 5000 to 6FFF while looking for DATA = 005E, the following sequence step could be used in some situations:

1. If DATA = 005E then Trigger
 - Else If ADDR in range 5000 to 6FFF then
 - Store Sample
 - Go to 1

Note the use of the store sample action. This means "store the most recently acquired sample in memory now". It does *not* mean, "From now on, start storing". It should be noted that since the store sample action is never executed unless ADDR is in the range 5000 to 6FFF, this branch essentially means "While in this sequence step, store only samples with ADDR between 5000 and 6FFF".

The above example seems to imply that only samples with ADDR between 5000 and 6FFF will be stored. However, this depends upon how the default storage has been set up. Using the previous example, if the default storage is set to "Store Everything", and a sample is outside of the range 5000 to 6FFF, then the Else If branch is not executed and the Default Storage is applied. In essence, the sequence step has said what to do when a sample has a value in a particular range, but it doesn't say what to do for samples outside the range. Therefore, if you want to specify the sequence step storage unambiguously, use the following:

1. If DATA = 005E then Trigger
 - Else If ADDR in range 5000 to 6FFF then
 - Store Sample
 - Go to 1
 - Else If ADDR not in range 5000 to 6FFF then
 - Don't Store Sample
 - Go to 1

Alternatively, if the default storage is set to "Store Everything", use the following:

1. If DATA = 005E then Trigger
 - Else If ADDR not in range 5000 to 6FFF then
 - Don't Store Sample
 - Go to 1

In summary, Sequence Step Storage always overrides the Default Storage, but only for the conditions specifically mentioned in the Sequence Step Storage. You must be very careful that you account for the interaction between Default Storage and Sequence Step Storage.

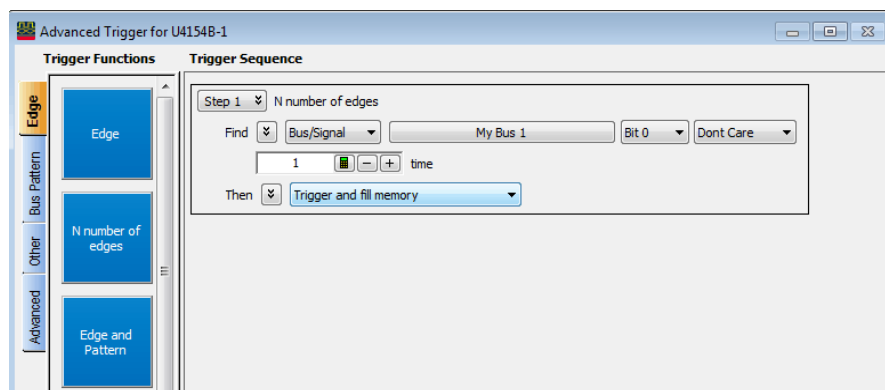
Next: Strategies for Setting Up Triggers (see [page 376](#))

Strategies for Setting Up Triggers

- Trigger Functions (see [page 376](#))
- Setting Up Complex Triggers (see [page 378](#))
- Save and Document Your Trigger Sequences (see [page 378](#))

Trigger Functions

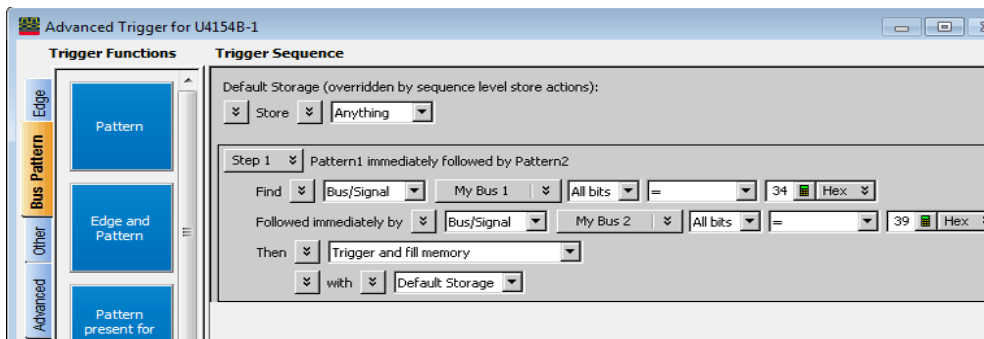
While setting up logic analyzer triggers can be difficult, *trigger functions* can greatly simplify the process. Trigger functions are commonly-needed building blocks that can be combined to set up a trigger. Because the functions cover most common triggers, you can set up your trigger simply by selecting the appropriate function and filling in the data. The logic analyzer trigger user interface is shown in the following figure. Note that trigger functions are prominently located at the left of the screen.



The trigger user interface

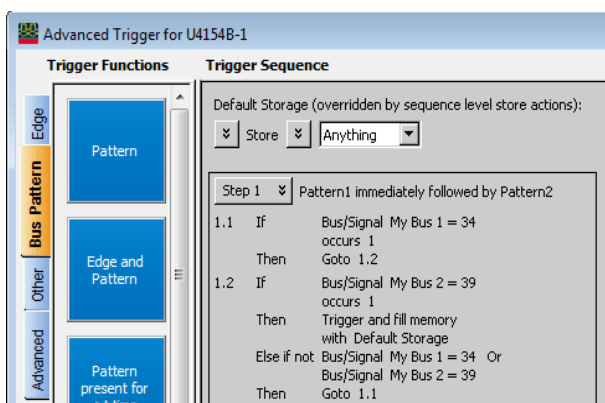
Note that a picture (which corresponds to the selected function) is provided by hovering over the trigger function button.

For example, if you want to trigger when a bus pattern is immediately followed by another bus pattern, you can drag-and-drop the "Pattern1 immediately followed by Pattern2" trigger function onto a trigger sequence step, as shown in the following figure.



Pattern1 immediately followed by Pattern2

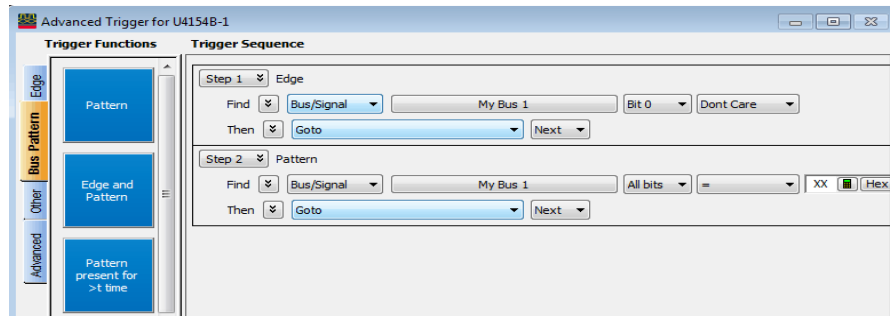
Once you have selected this function, you simply fill in the names of the buses and the patterns. Contrast the previous figure with the following figure, which is the same trigger created using If/Then statements. The trigger function is easier to use because the additional details of the If/Then statements have been hidden. However, if you want to see the details, you can *show trigger step as if/then*.



The same trigger as If/Then statements

Trigger functions can be modified. For example, if you start with the function "Find Edge", you can add another event, and it becomes the same as "Find Edge and Pattern". Therefore, a function that is not exactly correct can often be converted into the desired trigger. It is also possible to convert a trigger sequence step to advanced If/Then trigger functions and modify them.

Trigger functions are like building blocks because they can be used together in a trigger sequence. For example, if you want to set up a trigger as "Find edge followed by pattern", you can use a "Find Edge" function for Level 1 and a "Find Pattern" function for Sequence Level 2 (see the following figure). So, functions are useful both as an entire trigger sequence and as one step in a trigger sequence.



"Find Edge" and "Find Pattern" together

Next: Setting Up Complex Triggers (see [page 378](#))

Setting Up Complex Triggers

Frequently, the most difficult part of setting up a complex trigger is breaking down the problem. In other words, how do you map a complex trigger into sequence steps, branches, and Boolean expressions? Here are step by step instructions:

- 1 Break down the problem into events that don't happen simultaneously. These correspond to the sequence steps.
- 2 Scan the list of trigger functions to try to find some that match the events identified in Step #1.
- 3 Within all remaining events, break them down into Boolean expressions and their corresponding actions. Each Boolean expression/Action pair corresponds to a separate branch within a sequence step. Remember that "Store" branches may exist that are used only to handle storage qualification for that sequence step.

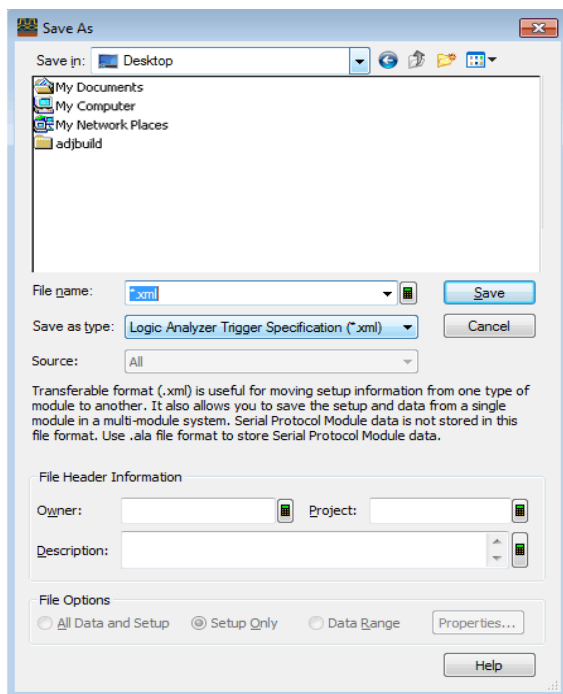
Next: Save and Document Your Trigger Sequences (see [page 378](#))

Save and Document Your Trigger Sequences

If a trigger sequence is important at one time, it is likely to be important again. This is why saving and documenting trigger sequences is so valuable. Complex trigger sequences generally are too difficult to understand without some accompanying explanation.

In Advanced If/Then trigger functions, you can include comments with the "If" clauses in a trigger sequence step. See ["To display or hide "If" clause comments"](#) on page 183.

When saving a trigger specification to a file, you can enter a description of the trigger sequence in the file header information (see ["To store a trigger"](#) on page 194 and the following figure).



Also, because the trigger specification file is in XML format, you edit the file and annotate steps with additional HTML-style comments (for example, `<!-- Comment. -->`).

Next: Conclusions (see [page 379](#))

Conclusions

Setting up logic analyzer triggers is very different than writing software. The job can be greatly simplified if other work can be leveraged by using pre-defined trigger functions and well-documented triggers that were written earlier. Only write your own trigger setup if there's nothing else available. Finally, when faced with a difficult trigger to set up, break down the problem into smaller chunks and deal with each one individually.

ALA vs. XML, When to Use Each Format

If you want to:	Then use:
Save and load sessions from a logic analysis system.	ALA Format – ALA format files are more complete, and the format is more efficient for saving and loading logic analyzer information.
Share captured data.	ALA Format – In <i>offline</i> mode, you can read ALA format configuration files in any instance of the <i>Keysight Logic Analyzer</i> application, without having licenses for licensed tools, windows, etc. that may be used in the configuration. This provides "read-only" capability; licenses are required to go <i>online</i> or add any licensed tools, windows, etc. XML Format – You can also share captured data using XML format configuration files. When loading an XML format file with data, you are forced into <i>offline</i> mode. Also, when viewing data from XML format configuration files, you need licenses for any licensed tools that may be used in the configuration.
Transfer module setup information between similar logic analyzers.	ALA Format – ALA format configuration files can only be loaded by logic analysis systems with compatible modules (for example, modules in the same or similar logic analyzer families).
Transfer module setup information between different logic analyzers.	XML Format – If you want to transfer setup information between incompatible modules, you must use XML format configuration files.
Transfer only part of a logic analyzer setup.	XML Format – You can load edited XML format configuration files.
Control the logic analysis system remotely using COM automation.	XML Format – The COM automation interface has commands for setting up parts of the logic analysis system with XML format strings. You can get these strings from XML format configuration files. (You can also load complete setups from ALA format configuration files using COM automation.)
Insert symbol information from software development tools.	XML Format – When a compiler-generated output file can't be loaded, you can save the configuration to an XML format file, edit it to include the symbol information (which has been translated into the logic analyzer's XML format), and open the XML format file again.

See Also

- To save a configuration file (see [page 198](#))
- To open a configuration file (see [page 208](#))
- To transfer module setups to/from multi-module systems (see [page 210](#))
- ALA Format (see [page 504](#))
- "XML Format" (in the online help)

Multiframe Logic Analysis Systems

If you need to make time-correlated measurements with more logic analysis channels than can be installed in a single frame/chassis, you can connect multiple M9505A, M9502A or M9595A chassis together.

NOTE

Not all logic analysis systems have multiframe capability. If the back of your instrument does not have an "Input" and "Output" connector, it does not support multiframe. For example, the 16850-series logic analyzers do not support multiframe.

The [AXIe based Logic and Protocol Analysis – Installation Guide](#) describes how to connect multiple frames/chassis. Basically, each M9505A, M9502A or M9595A chassis has two multiframe connectors, labeled "Input" and "Output". A multiframe cable connects the output of one frame to the input of another frame. You can chain as many frames as you like together this way. The *master frame* is the one with the open input connector; all other frames are *slave frames*, and the one with the open output connector is the *terminating slave frame*.

CAUTION

Failing to follow one of the recommended multiframe configurations in the installation guide can result in unpredictable software behavior and/or poor analyzer performance!

In addition to the multiframe cables, the frames must also be connected to a network. Usually, this is a Gbit LAN network. It can be a private or public network (see installation guide). The multiframe cable is used for time correlation, cross-triggering, and multiframe setup; all other inter-frame communication and data transfer take place over the network.

NOTE

If (and only if) you encounter problems after changing multiframe connections, try rebooting all frames in the multiframe set.

To ensure the smoothest multiframe startup, the following startup sequence is recommended:

- First, start up the system configured as the master frame. Let it fully boot and then log in as the preferred user. Once the desktop is available, wait until the Keysight Notification Center Icon (viewable in the system Tray) is displayed and in the green "ready" state.
- Proceed to start the next intermediate slave system in the multiframe chain, again waiting until the instrument is in the ready state.
- Continue sequentially with the rest of the slave systems, allowing each to get to the ready state.
- Lastly, start the terminating slave system and allow it to reach the ready state.

You may now start the controlling Logic Analyzer application, which can run on the master system, any of the slave systems, or even remotely from another personal computer.

A "System" is considered to be a combination of the host controller and the instrument chassis. For the 16850 series of standalone instruments, the host controller and instrument chassis is contained in one, single frame package. For the M9505A, M9502A or M9505A chassis with one or more instrument modules installed, "system" is the combination of the chassis connected via some network connection to either an external remote host computer or an embedded host controller module installed in the chassis.

Remotely hosted systems may have their own boot sequence requirements. Usually this is to boot the chassis first and then the host computer.

CAUTION

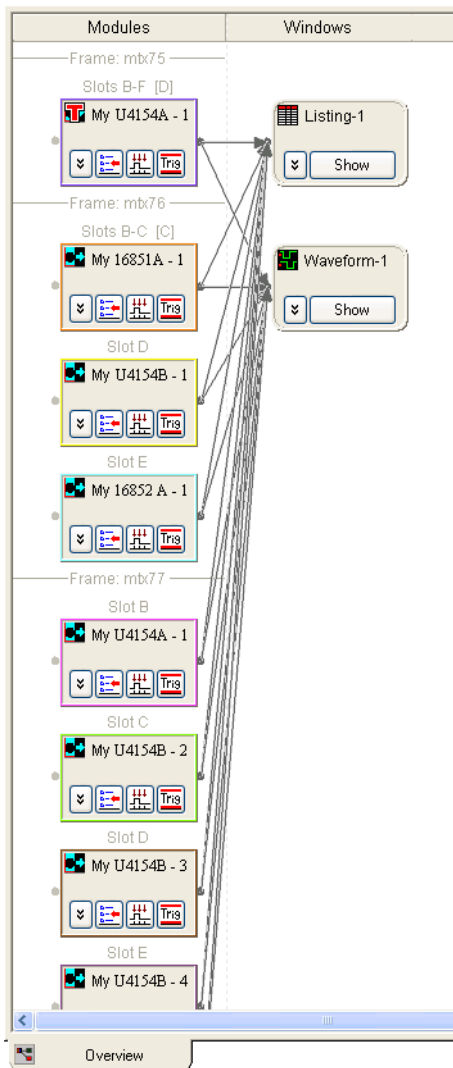
Changing multiframe cables and/or network cables while a user is connected (online) with the multiframe set will force them offline—losing any unsaved changes.

NOTE

When an operating application on an instrument is forced offline due to a multiframe connection being made, multiple offline warning messages are displayed which require a user response to continue. For smoother multiframe startup, it is recommended to remove any startup menu shortcuts for the Logic Analysis application from all systems in the multiframe scenario so that no instrument systems can auto-start at login and enter online mode.

Some things to consider when using multiframe logic analysis systems:

- Use the Overview window to tell which frames are connected and which modules are in each frame.



- When triggering from, or sending a trigger to, an external, non-multiframe instrument, you must use the **Trigger In** or **Trigger Out** BNC connectors on the *master frame*.
- When you arm between modules in different frames, an unused flag line is implicitly used to facilitate the arming. (Flag lines already used in the trigger setup or to arm the external Trigger Out are not used.)
- The *Keysight Notification Center Icon* (lower, right-hand corner of the desktop) will be present when logged onto any standalone instrument. You can double-click this icon to get frame and module details on all frames in the multiframe set. If connected remotely from a PC, this information is available via the Overview window (mentioned above) and the System Summary dialog.

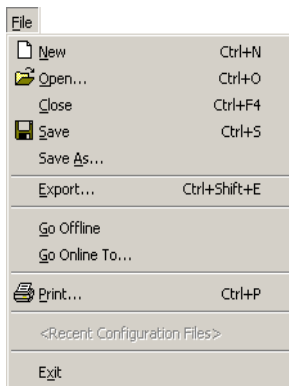
13 Reference

- Menus / 386
- Toolbars / 398
- Marker Measurement Display Bar / 403
- Windows / 404
- Dialogs / 421
- Trigger Functions / 478
- Data Formats / 504
- Object File Formats Supported by the Symbol Reader / 513
- General-Purpose ASCII (GPA) Symbol File Format / 514
- Product Overviews / 520
 - U4164A Logic Analyzer Product Overview / 520
 - U4154A Logic Analyzer Product Overview / 522
 - U4154B Logic Analyzer Product Overview / 523
 - 16850-Series Logic Analyzer Product Overview / 524
 - 16860-Series Logic Analyzer Product Overview / 528
 - Keysight Logic and Protocol Analyzer Application Product Overview / 532
- Logic Analyzer Notes / 536
 - U4164A Logic Analyzer Notes / 536
 - U4154A Logic Analyzer Notes / 544
 - U4154B Logic Analyzer Notes / 549
 - 16850-Series Logic Analyzer Notes / 553
 - 16860-Series Logic Analyzer Notes / 555
- Specifications and Characteristics / 587
 - U4164A Logic Analyzer Specifications and Characteristics / 587
 - U4154A Logic Analyzer Specifications and Characteristics / 592
 - U4154B Logic Analyzer Specifications and Characteristics / 596
 - 16850-Series Logic Analyzer Specifications and Characteristics / 600
 - 16860-series Logic Analyzer Specifications and Characteristics / 604

Menus

- File Menu (see [page 386](#))
- Edit Menu (see [page 387](#))
- View Menu (see [page 388](#))
- Setup Menu (see [page 389](#))
- Tools Menu (see [page 390](#))
- Markers Menu (see [page 391](#))
- Run/Stop Menu (see [page 392](#))
- Overview Menu (see [page 393](#))
- Listing Menu (see [page 393](#))
- Waveform Menu (see [page 394](#))
- Compare Menu (see [page 394](#))
- Source Menu (see [page 395](#))
- Protocol Viewer Menu (see [page 395](#))
- Window Menu (see [page 396](#))
- Help Menu (see [page 397](#))

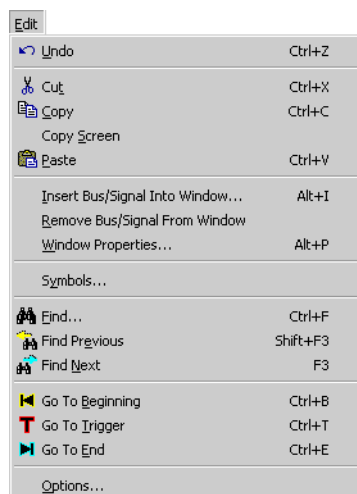
File Menu



Menu	Description
New	Creates a new logic analyzer configuration file.
Open...	Opens a previously saved logic analyzer configuration file.
Close	Closes the active window after asking whether to save its data.
Save (see page 198)	Saves changes to the currently open configuration file.
Save As... (see page 198)	Saves the currently open configuration file to a new name.
Export... (see page 199)	Saves captured data to comma-separated value (CSV) files.
Go Offline	Disconnects the <i>Keysight Logic Analyzer</i> application from the currently connected frame.

Menu	Description
Go Online To Local Frame	Connects the <i>Keysight Logic Analyzer</i> application to the local frame. If there is no local frame, the Offline Startup Options dialog (see page 447) opens.
Go Online To...	Opens the Select System to Use dialog (see page 459) for choosing a frame to connect the <i>Keysight Logic Analyzer</i> application to.
Print...	Opens the Printing Data dialog (see page 451) for printing displayed data within a defined range.
Recent Configuration Files (see page 210)	Lists recently opened files for quick reference or access.
Exit	Closes the logic analyzer user interface window.

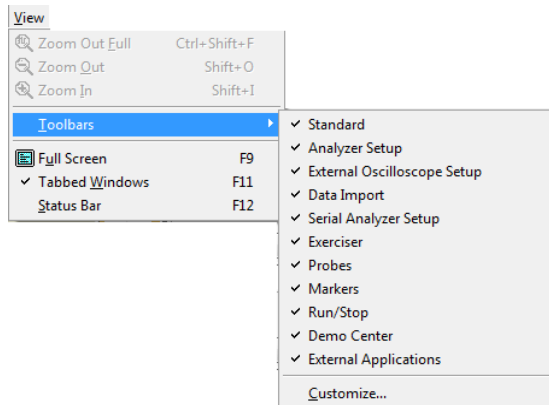
Edit Menu



Menu	Description
Undo	Undo the last user action. This includes any properties that have changed such as column color, column width, column move, column insert, column delete, etc. Items that cannot be undone include scrolling, acquisition runs and simple trigger modifications.
Cut	Cuts the selection from alphanumeric fields in listing and waveform windows. Alphanumeric fields in lower level modal dialogs are cut using keyboard commands (see page 533) (accelerator keys). Cut selections are pasted to the clip board.
Copy	Copies the selection from alphanumeric fields in listing and waveform windows. Alphanumeric fields in lower level modal dialogs are copied using keyboard commands (see page 533) (accelerator keys). Copied selections are pasted to the clip board.
Copy Screen (see page 315)	Copies the current screen to a bitmap and places it on the system clip board.
Paste	Pastes the cut or copied data that is stored in the clip board into the alphanumeric field. Alphanumeric data is pasted into fields in lower level modal dialogs using keyboard commands (see page 533) (accelerator keys).

Menu	Description
Insert Bus/Signal Into Window...	Inserts a predefined bus or signal into the display.
Remove Bus/Signal From Window	Deletes the highlighted bus or signal from the display window.
Window Properties... (see page 452)	Accesses the window properties dialog.
Symbols...	Opens the Symbols dialog (see page 471) for setting up symbols for the selected bus/signal.
Find... (see page 264)	Locates specific data in the acquisition.
Find Previous (see page 264)	Locates the previous occurrence of the specified data.
Find Next (see page 264)	Locates the next occurrence of the specified data.
Go To Beginning (see page 249)	Places the beginning of the captured data trace at center screen.
Go To Trigger (see page 249)	Places the trigger point at center screen.
Go To End (see page 249)	Places the end of the captured data trace at center screen.
Options...	Accesses the System Options dialog (see page 447).

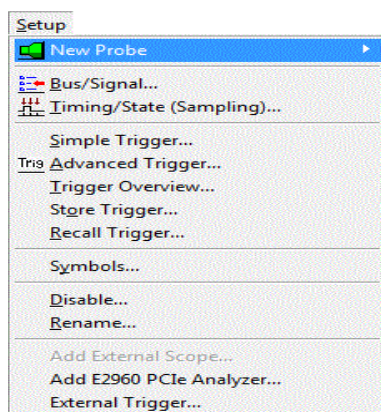
View Menu



Menu	Description
Zoom Out Full	Zooms out on an active window as far as possible.
Zoom Out (see page 221)	Zooms out on an active window.
Zoom In (see page 221)	Zooms in on an active window.
Toolbars (see page 398)	Access the Toolbar dialog window.

Menu	Description
Full Screen	Enables or disables full screen display.
Tabbed Windows (see page 313)	Enables or disables Listing and Waveform tabs.
Status Bar	Enables or disables the status bar.

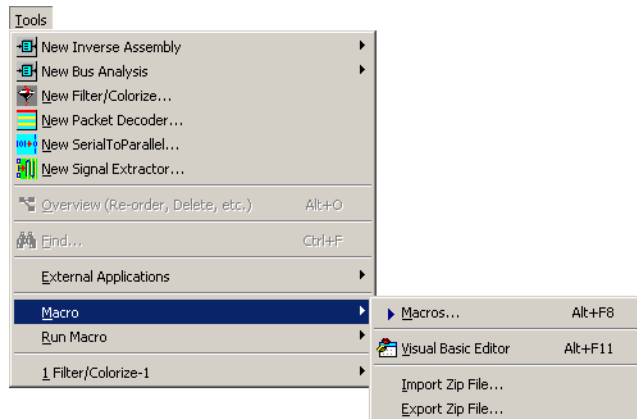
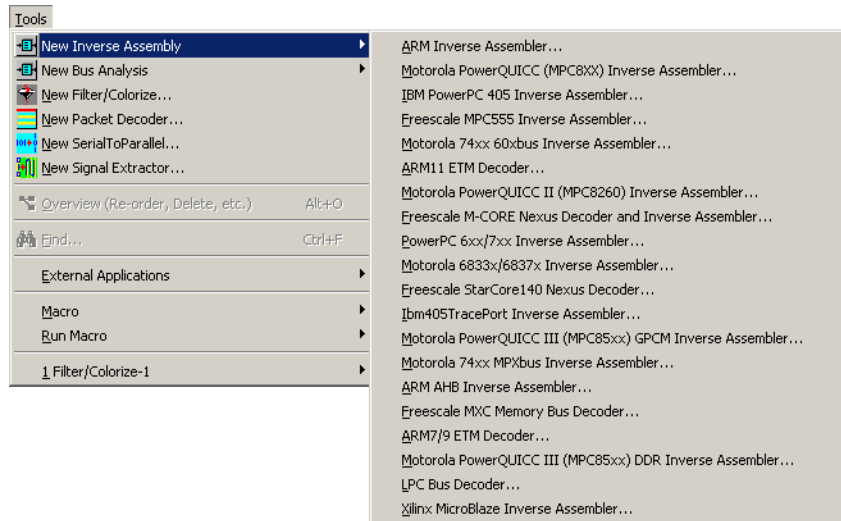
Setup Menu



Menu	Description
New Probe	Lets you set up the FPGA Dynamic Probe (for probing signals internal to an FPGA) or set up definitions for other probes that are used (see "Setting Up Probes" (in the online help)).
Bus/Signal...	Accesses the Buses/Signals (see page 426) tab of the Analyzer Setup dialog (see page 426).
Timing/State (Sampling)...	Accesses the Sampling (see page 428) tab of the Analyzer Setup dialog (see page 426).
Simple Trigger...	See Specifying Simple Triggers (see page 146).
Advanced Trigger...	Accesses the Advanced Trigger (see page 424) dialog.
Store Trigger...	Stores current trigger.
Recall Trigger... (see page 453)	Accesses a list of most recently used triggers.
Symbols...	Opens the Symbols dialog (see page 471) for setting up symbols for the selected bus/signal.
Disable.../ Enable...	Disabling a module prevents its captured data from being sent to tools and display windows; this will speed up the processing of data from other modules.
Rename...	Lets you rename the logic analyzer module.

Menu	Description
" Add External Scope..." (in the online help)	Runs the <i>Add External Oscilloscope</i> wizard for connecting an external oscilloscope to the logic analyzer.
Delete External Scope...	Removes the setup for an externally connected oscilloscope.
External Trigger...	Opens the External Trigger dialog (see page 437) for setting up triggers between the logic analyzer and other, external instruments.
Target Control Port...	Opens the Target Control Port dialog (see page 475) for outputting signals on the logic analysis system frame's <i>target control port</i> .
Skew & System Trigger...	Opens the Module Skew and System Trigger dialog (see page 446) for specifying which <i>module</i> (see page 616) is the <i>system trigger</i> (that is, which module's trigger reference point is <i>Time=0</i>) and for specifying the trigger reference point skew for modules that are not the <i>system trigger</i> .

Tools Menu



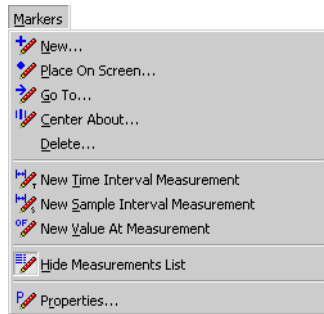
All add-in tools are grouped under the tools menu. The *Keysight Logic Analyzer* application comes with a filter/colorize tool built in. If you are using inverse assemblers, bus analysis tools, or other third-party tools, the tools will show up in the Tools menu under New.

As tools are created, they are added to the bottom of the Tools menu. The menus above show one active tool.

Menu	Description
New ... (see page 307)	Creates a new inverse assembly, bus analysis, filter/colorize, packet decoder, serial to parallel, or signal extractor tool.
Overview (see page 307)	Lets you manage the active tools.
Find... (see page 264)	Locates specific data in the acquisition.
External Applications>	Lets you run external applications from the <i>Keysight Logic Analyzer</i> application's menu. Setup... on the submenu opens the External Application Setup dialog (see page 436) that lets you add, edit, arrange, or remove items from the submenu.
Macro>	Lets you: <ul style="list-style-type: none"> · Open the Macros dialog for "choosing a Visual Basic macro to run" (in the online help). · Open the Visual Basic Editor for "editing programs" (in the online help). · "Export COM/DCOM project code to .zip files" (in the online help). For more information, refer to the Visual Basic online help.
Run Macro>	Opens the Add-In Manager dialog which lets you register an add-in (a customized tool that adds capabilities to the Visual Basic development environment), load or unload it, and set its load behavior. For more information, click Help in the Add-In Manager dialog. Runs a sample macro. As shipped from the factory, the submenu contains: <ul style="list-style-type: none"> · FindEdges (macro for displaying the time between two edges and placing markers on certain edge pairs). · RepetitiveSaveToFile (macro for saving data from repetitive runs to incrementing file names). · SendToExcel (macro for sending logic analyzer data to Microsoft Excel). You can add your own COM/DCOM macros to this submenu by placing COM/DCOM project code .zip files in the directory: <pre><Drive letter>:\<Install directory>\ COM/DCOM\ For example: C:\Program Files\Keysight Technologies\ Logic Analyzer\COM/DCOM\</pre>
1 <i>tool name</i> (see page 310)	Edit an existing tool.

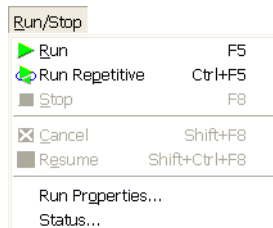
- See Also
- To add a new tool (see [page 307](#))
 - To change a tool (see [page 310](#))
 - To delete a tool (see [page 307](#))
 - "Using the Filter/Colorize Tool" (in the online help)

Markers Menu



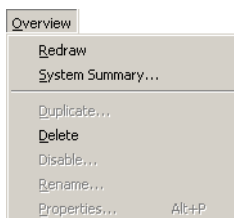
Menu	Description
New... (see page 246)	Creates a new marker.
Place on Screen... (see page 248)	Places a new or selected (existing) marker at the middle of the screen.
Go To... (see page 249)	Goes to a selected marker.
Center About... (see page 249)	Centers the display around a selected marker.
Delete...	Deletes selected markers.
New Time Interval Measurement (see page 251)	Creates a new time interval measurement.
New Sample Interval Measurement (see page 252)	Creates a new sample interval measurement.
New Value At Measurement (see page 253)	Creates a new value at measurement.
Hide/Show Measurements List	Hides or shows the marker measurement display bar.
Properties... (see page 255)	Accesses the markers properties dialog

Run/Stop Menu



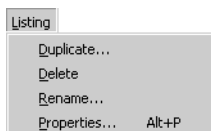
Menu	Description
Run	Starts sampling, fills logic analyzer memory with samples around the trigger, and stops.
Run Repetitive	Starts sampling, fills logic analyzer memory with samples around the trigger, and repeats.
Stop	Stops a logic analyzer measurement that is in progress.
Cancel	When searching, using a filter, or exporting captured data, Cancel stops the operation.
Resume	If you have used Cancel to stop a filter tool operation, Resume continues the filter operation.
Run Properties...	Opens the Run Properties dialog (see page 454) which lets you enable, and set the options for, saving captured data after each run and stopping after a certain number of repetitive runs.
Status...	Opens the Status dialog (see page 469).

Overview Menu



Menu	Description
Redraw	Re-paints the Overview window.
System Summary...	Opens the System Summary dialog (see page 472) which displays information about the <i>frames</i> (see page 615), <i>modules</i> (see page 616), <i>cards</i> (see page 614), and <i>slots</i> (see page 618) in the logic analysis system.
Delete	Closes the Overview window.

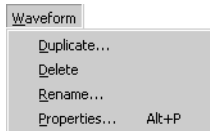
Listing Menu



Menu	Description
Duplicate...	Adds a new Listing window with the same properties as the window being displayed.
Delete	Closes the Listing window.
Rename...	Lets you rename the Listing window.
Properties...	Lets you change Listing window properties.

- See Also
- Analyzing Listing Data (see [page 235](#))
 - To set listing window properties (see [page 238](#))

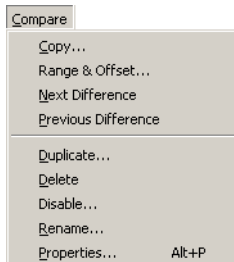
Waveform Menu



Menu	Description
Duplicate...	Adds a new Waveform window with the same properties as the window being displayed.
Delete	Closes the Waveform window.
Rename...	Lets you rename the Waveform window.
Properties...	Lets you change Waveform window properties.

- See Also
- Analyzing Waveform Data (see [page 220](#))
 - To set waveform window properties (see [page 227](#))

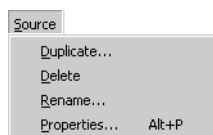
Compare Menu



Menu	Description
Copy...	Copies data to the reference buffer.
Range & Offset...	Lets you compare a range of samples and offset the reference data.
Next Difference	Finds the next difference (below the center reference).
Previous Difference	Finds the previous difference (above the center reference).
Duplicate...	Adds a new Compare window with the same properties as the window being displayed.
Delete	Closes the Compare window.
Disable..., Enable...	Lets you disable or re-enable the Compare window.
Rename...	Lets you rename the Compare window.
Properties...	Lets you change Compare window properties.

- See Also
- Compare Display Window (see [page 411](#))
 - Comparing Captured Data to Reference Data (see [page 274](#))
 - To set Compare window properties (see [page 276](#))

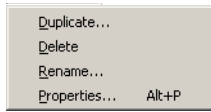
Source Menu



Menu	Description
Duplicate...	Adds a new Source window with the same properties as the window being displayed.
Delete	Closes the Source window.
Rename...	Lets you rename the Source window.
Properties...	Lets you change Source window properties.

- See Also
- Source Display Window (see [page 412](#))
 - Viewing Source Code Associated with Captured Data (see [page 277](#))
 - To set Source window properties (see [page 281](#))

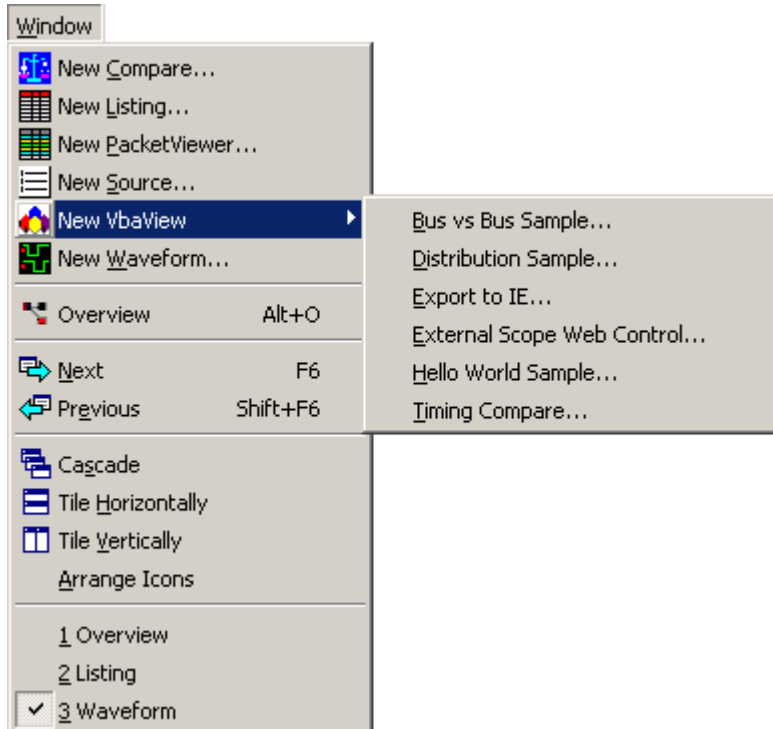
Protocol Viewer Menu



Menu	Description
Duplicate...	Adds a new Protocol Viewer window with the same properties as the window being displayed.
Delete	Closes and deletes the Protocol Viewer window.
Rename...	Lets you rename the Protocol Viewer window.
Properties...	Lets you change Protocol Viewer window properties.

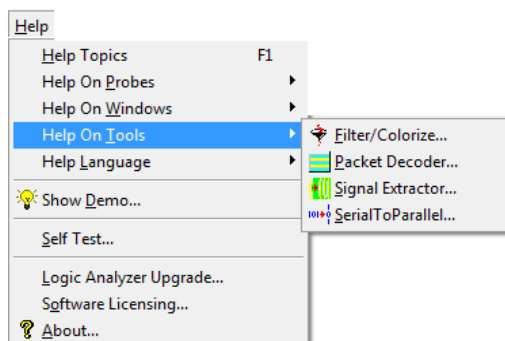
- See Also
- Protocol Viewer Display Window (see [page 413](#))
 - Analyzing Packet Data (see [page 283](#))
 - Changing Protocol Viewer Window Properties (see [page 302](#))

Window Menu



Menu	Description
New Compare...	Creates a new Compare window (see page 411).
New Listing...	Creates an additional Listing window (see page 409).
New Protocol Viewer...	Creates an additional Protocol Viewer window (see page 413).
New Source...	Creates a new Source window (see page 412).

Help Menu



Menu	Description
Help Topics	Accesses the online help.
Help On Probes	Opens online help for probes.
Help On Windows	Opens online help for the Waveform, Listing, Compare, or Source windows.
Help On Tools	Opens online help for tools.
Help Language	Lets you choose between the English and Japanese versions of the online help (see Accessing Japanese Online Help (see page 345)).
Show Demo...	Launches the Demo Center (see page 55) application that demonstrates logic analysis system features.
Self Test...	Accesses the Logic Analyzer Self-Tests dialog (see Running Self Tests (see page 343)).
Logic Analyzer Upgrade...	Accesses the Keysight Logic Analyzer Upgrade dialog (see page 425) which provides information for upgrading hardware in logic analyzer modules.
Software Licensing...	Opens the Software Licensing dialog (see page 461) for managing software licenses used by the logic analysis system.
Software Update...	Opens the Add or Remove Keysight Logic Analyzer Software (see page 327) tool for managing your logic analyzer software and keeping it up to date.
About...	Displays product version and copyright information.














Toolbars

Toolbars are located under the menu bar, and are used to quickly access a function or perform a task. By default, not all toolbars, or individual tools within a given toolbar are displayed. For a complete list of all available toolbars, choose **View>Toolbars>**. For a complete list of all tools within a given toolbar, choose **View>Toolbars>Customize...**; then, select the **Commands** tab in the Customize dialog.

- Standard (see [page 398](#))
- Analyzer Setup (see [page 399](#))
- External Oscilloscope Setup (see [page 399](#))
- Data Import (see [page 400](#))
- Probes (see [page 400](#))
- Markers (see [page 400](#))
- Run/Stop (see [page 401](#))
- Visual Basic (see [page 401](#))
- Customize... (see [page 402](#))












Standard Toolbar



Menu	Description
	New - Creates a new logic analyzer configuration file.
	Open - Opens a previously saved logic analyzer configuration file.
	Save - Saves changes to the currently open configuration file.
	Print (see page 451) - Prints displayed data within a defined range.
	Find (see page 264) - Locates specific data in the acquisition.
	Find Previous (see page 264) - Locates the previous occurrence of the specified data.
	Find Next (see page 264) - Locates the next occurrence of the specified data.
	Go to Beginning (see page 249) - Centers the beginning of the acquisition data.
	Go to Trigger (see page 249) - Centers the trigger point of the acquisition.
	Go to End (see page 249) - Centers the end of the acquisition data.
	Zoom Out (see page 221) - Zooms in on an active window.
	Zoom In (see page 221) - Zooms out on an active window.
	Overview - Opens or displays the Overview window (see page 418).

NOTE




The following are optional standard toolbar icons.

Menu	Description
	Cuts the selection and places it on the clip board.
	Copies the selection and places it on the clip board.
	Pastes the data that is stored on the clip board.
	Provides online help information about the <i>Keysight Logic Analyzer</i> application.
	Undo last user action.
	Enables or disables full screen display.
	Activate the next window.
	Activates the previous window.
	Arranges windows as cascaded overlapping tiles.
	Arranges windows as non-overlapping horizontal tiles.
	Arranges windows as non-overlapping vertical tiles.

See Also • To create a custom toolbar (see [page 402](#))

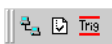
Analyzer Setup Toolbar






Menu	Description
	Bus/Signal - Accesses the Buses/Signals (see page 426) tab of the Setup dialog.
	Timing/State (Sampling) - Accesses the Sampling (see page 428) tab of the Setup dialog.
	Advanced Trigger - Accesses the Advanced Trigger (see page 424) dialog.

See Also • To create a custom toolbar (see [page 402](#))

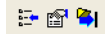
External Oscilloscope Setup Toolbar






Menu	Description
	Sets up the external oscilloscope connection attributes.
	Sets up the external oscilloscope option attributes.
	Sets up the external oscilloscope trigger attributes.

See Also · "Infiniium Oscilloscope Time Correlation" (in the online help)

Data Import Toolbar




Menu	Description
	Lets you edit the data import module bus/signal definitions.
	Displays data import module file information.
	Re-reads the data import module file.

See Also · Using Data Import Modules (see [page 211](#))





Probes Toolbar



Menu	Description
	Opens the properties dialog for a particular probe. <ul style="list-style-type: none"> · "Using the Xilinx FPGA Dynamic Probe" (in the online help) · "Using the FPGA Dynamic Probe for Altera FPGAs" (in the online help) · "Using General Purpose Probes" (in the online help) · "PCI Express Analysis Probe" (in the online help) · "Serial ATA Analysis Probe" (in the online help)






Markers Toolbar



Menu	Description
	New (see page 246) - Creates a new marker.
	Go To (see page 249) - Centers the display around the selected marker.
	Creates a new value at a measurement (see page 253).
	Creates a new time interval measurement (see page 251).

NOTE






The following are optional markers toolbar icons.

Menu	Description
	Place Maker (see page 248) - Places a new or selected (existing) marker at the middle of the screen.
	Center About (see page 249) - Centers the display around two selected markers.
	Creates a new sample interval measurement (see page 252).
	Hides or shows the marker measurement display bar.
	Accesses the markers properties (see page 255) dialog.

See Also • To create a custom toolbar (see [page 402](#))

Run/Stop Toolbar





Menu	Description
	Starts sampling, fills logic analyzer memory with samples.
	Starts Sampling, fills logic analyzer memory with samples around the trigger, and repeats.
	Stops a logic analyzer measurement in progress, for example, when the trigger condition is not found.
	Cancels the current operation.
	Resumes the cancelled operation.

See Also • To create a custom toolbar (see [page 402](#))

Visual Basic Toolbar



Menu	Description
	Runs a COM/DCOM macro.
	Opens the Visual Basic Editor.

See Also • "Using the Advanced Customization Environment (ACE)" (in the online help)

To customize toolbars

- To add icons to a toolbar (see [page 402](#))
- To remove icons from a toolbar (see [page 402](#))
- To create a new toolbar (see [page 402](#))
- To restore a toolbar to its original icons (see [page 402](#))

To add icons to a toolbar

- 1 From the menu bar, select **View>Toolbars>Customize...**
- 2 Select the **Commands** tab.
- 3 Select the **Category** that you want to add icons from.
- 4 Drag the desired icon from the **Buttons** area to the desired position on the toolbar; then, release the mouse button to insert the tool icon.
- 5 Repeat for any other icons you wish to add.

To remove icons from a toolbar

- 1 From the menu bar, select **View>Toolbars>Customize...**
- 2 Select the **Commands** tab.
- 3 Drag the icon from the toolbar and drop it onto the **Buttons** area of the Customize dialog.
- 4 Repeat for any other icons you wish to remove.

To create a new toolbar

- 1 From the menu bar, select **View>Toolbars>Customize...**
- 2 In the Customize dialog's Toolbars tab, click **New...**
- 3 In the New Toolbar dialog, enter the name of the new toolbar, and click **OK**.
- 4 Drag the new toolbar window to the desired position in the toolbar dock.
A second row of toolbars can be created by dragging a toolbar to the bottom of an existing toolbar row.
If a toolbar is hidden off-screen, drag a visible toolbar to create a second row of toolbars; that should then reveal the hidden toolbar.

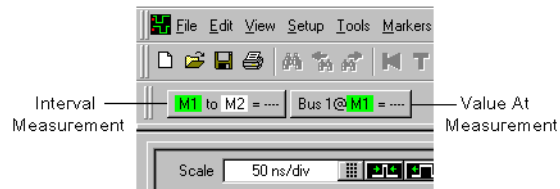
Once you have created a new toolbar, you can add or remove icons as described above.

To restore a toolbar to its original icons

- 1 From the menu bar, select **View>Toolbars>Customize...**
- 2 In the Customize dialog's Toolbars tab, select the name of the **Toolbar** you want to restore.
- 3 Click **Reset**.

Marker Measurement Display Bar

Marker "interval" and "value at" measurements are displayed below the menu bar with the other toolbars.

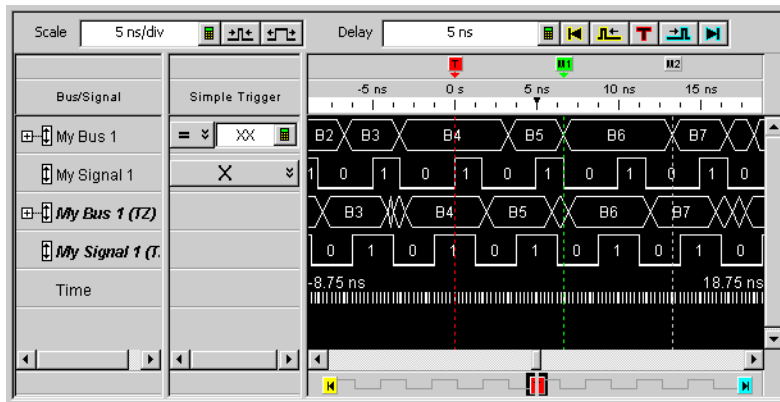


- To create a new time interval measurement (see [page 251](#))
- To create a new sample interval measurement (see [page 252](#))
- To create a new value at measurement (see [page 253](#))
- To hide/show measurement display bar (see [page 391](#))

Windows

- Waveform Display Window (see [page 404](#))
 - Markers Display Bar (see [page 407](#))
 - Markers Overview Bar (see [page 408](#))
- Listing Display Window (see [page 409](#))
- Compare Display Window (see [page 411](#))
- Source Display Window (see [page 412](#))
- Protocol Viewer Display Window (see [page 413](#))
- Overview Window (see [page 418](#))

Waveform Display Window



The Waveform window is accessed through the menu bar's **Window>Waveform**. If you have Tabbed Windows (see [page 313](#)) turned on, you can also select a tab at the bottom of the window.

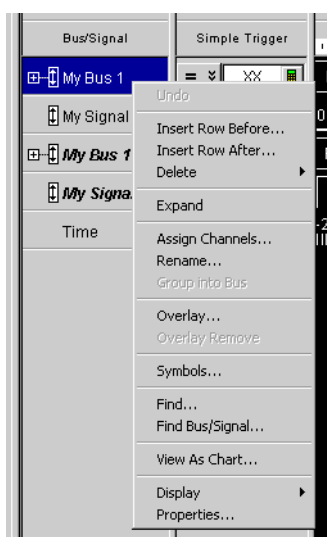
The Waveform window displays captured data as a digital waveform. You can configure the window to display selected buses and signals with time or pattern markers in the data. You can also set up bus pattern triggers and signal trigger options.

The Waveform window consists of the following areas:

- Bus/Signal Configuration (see [page 404](#))
- Simple Trigger (see [page 146](#))
- Markers Display Bar (see [page 407](#))
- Waveform Display Area (see [page 406](#))
- Markers Overview Bar (see [page 408](#))
- Delay Controls (see [page 222](#))
- Scale (time/division) Controls (see [page 221](#))

Bus/Signal Configuration

To access the following Bus/Signal configuration options, right-click on any bus or signal name in the Bus/Signal column.



Status	Description
Undo	Undo the last action performed.
Insert Row Before...	Inserts a bus/signal before the highlighted row.
Insert Row After...	Inserts a bus/signal after the highlighted row.
Delete>	Deletes the bus/signal in the highlighted row or deletes all buses/signals.
Expand	Expands the highlighted bus into separate displayed channels.
Collapse	Collapses displayed channels to a single displayed bus.
Assign Channels...	Access to the Buses/Signals tab of the Analyzer Setup dialog for mapping (assigning) the highlighted bus/signal to the desired pod and channel connection of the probes.
Rename...	Access a keypad to rename the highlighted bus/signal.
Group into Bus (see page 227)	Groups highlighted signals into a bus.
Overlay... (see page 223)	Overlays the highlighted bus or signal with another selected bus or signal.
Overlay Remove	Separates overlaid bus/signals.
Symbols...	Opens the Symbols dialog (see page 471) for setting up symbols for the selected bus/signal.
Find...	Opens the Find dialog for searching the captured data (see page 264).
Find Bus/Signal...	Searches for a bus/signal row.
View As Chart... (see page 224)	Opens the View As dialog for viewing the bus data as a chart or a bus.
Display>	Lets you show or hide parts of the Waveform window (see page 226).
Properties... (see page 227)	Access to properties dialog for waveform window, bus/signal row, bus/signal column, and marker properties.

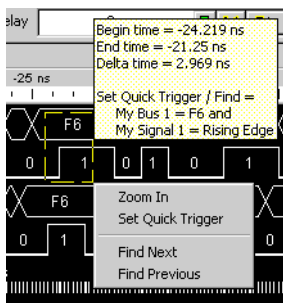
Waveform Display Area

To access waveform display options, right-click anywhere in the display area.



Status	Description
Undo	Undo the last action performed.
Zoom Out (see page 221)	
Zoom In (see page 221)	
Go To (see page 222)	
Place Marker> (see page 247)	
Center About>	Centers the display about a marker pair (see page 249) or waveform edges (see page 221).
Find... (see page 264)	
Find Next (see page 264)	
Find Previous (see page 264)	
Properties... (see page 227)	

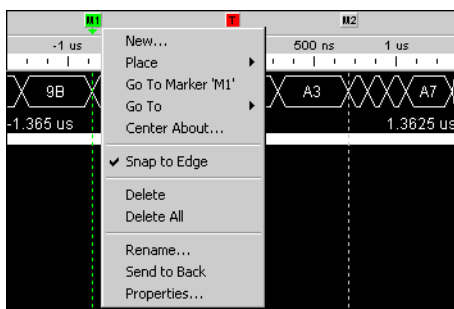
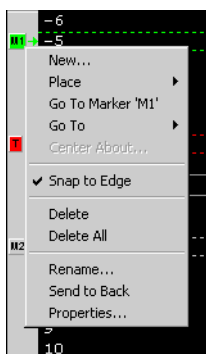
Drawing Rectangle in Data



Menu	Description
Zoom In (see page 221)	
Set Quick Trigger (see page 143)	Alternative way to set a Simple Trigger.
Find Next	Data value on left edge of rectangle becomes Find search (see page 264) criteria and next occurrence of that data value is placed at center screen.
Find Previous	Data value on left edge of rectangle becomes Find search (see page 264) criteria and previous occurrence of that data value is placed at center screen.

- See Also
- Analyzing Waveform Data (see [page 220](#))
 - Marking, and Measuring Between, Data Points (see [page 245](#))
 - Specifying Advanced Triggers (see [page 152](#))
 - Setting Up Symbols (see [page 132](#))
 - To add or delete display windows (see [page 313](#))
 - To turn window tabs on/off (see [page 313](#))

Markers Display Bar

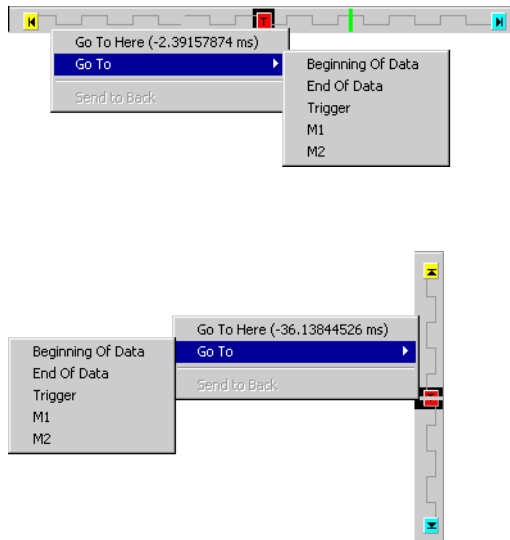


To access these tasks, right-click anywhere in the marker display bar.

Menu	Description
New... (see page 246)	To create new markers.
Place (see page 247)	To place markers in data.
Go To (see page 249)	To go to a marker.
Center About... (see page 249)	To center the display about a marker pair.
Snap to Edge (see page 250)	To toggle a marker's snap to edge property.
Delete (see page 250)	To delete a marker.
Delete All (see page 250)	To delete all markers.
Rename... (see page 254)	To rename a marker.
Send to Back (see page 254)	To send a marker to the back.
Properties... (see page 255)	To set marker properties.

- See Also
- To read the markers display and overview bars (see [page 245](#))
 - Markers Menu (see [page 391](#))
 - Markers Toolbar (see [page 400](#))
 - Markers Overview Bar (see [page 408](#))

Markers Overview Bar



To access these menus, click anywhere in the marker overview bar.

Menu	Description
Go To (see page 249)	To go to a marker.
Send to back (see page 254)	To send a marker to the back.

- See Also
- To read the markers display and overview bars (see [page 245](#))
 - Markers menu (see [page 391](#))
 - Markers toolbar (see [page 400](#))

Listing Display Window

Sample Number	Bus 1	Bus 2	Signal 1	Signal 2	Time
-7	F9	3FF9	1	1	-350
-6	FA	3FFA	1	1	-300
-5	FB	3FFB	1	1	-250
-4	FC	3FFC	1	1	-200
-3	FD	3FFD	1	1	-150
-2	FE	3FFE	1	1	-100
-1	FF	3FFF	1	1	-50
0	00	0000	0	0	0
1	01	0001	0	0	50
2	02	0002	0	0	100
3	03	0003	0	0	150
4	04	0004	0	0	200
5	05	0005	0	0	250
6	06	0006	0	0	300
7	07	0007	0	0	350

The Listing window is accessed through the menu bar's **Window>Listing**. If you have Tabbed Windows (see [page 313](#)) turned on, you can also select a tab at the bottom of the window.

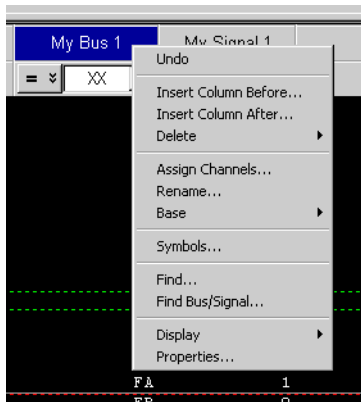
The Listing window displays your captured data as a state listing. You configure the window to display selected buses and signals in columns. Within the listed data, you can insert time or pattern markers. You can also configure the bus pattern triggers and signal trigger options.

The Listing window consists of the following areas:

- Column Configuration (see [page 409](#))
- Simple Trigger (see [page 146](#))
- Markers Display Bar (see [page 407](#))
- Listing Display Area (see [page 410](#))
- Markers Overview Bar (see [page 408](#))

Column Configuration

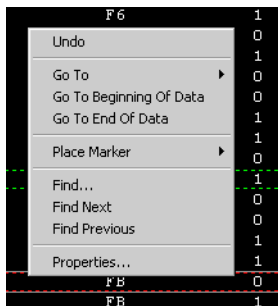
To access the following column configuration options, right-click on any bus or signal name in the column head.



Menu	Description
Undo	Undo the last action performed.
Insert Column Before... (see page 238)	
Insert Column After... (see page 238)	
Delete > (see page 238)	
Assign Channels... (see page 83)	
Rename...	Access a keypad to rename the highlighted bus/signal.
Base > (see page 241)	
Symbols...	Opens the Symbols dialog (see page 471) for setting up symbols for the selected bus/signal.
Find...	Opens the Find dialog for searching the captured data (see page 264).
Find Bus/Signal...	Searches for a bus/signal column.
Display >	Lets you show or hide parts of the Listing window (see page 237).
Properties... (see page 238)	Access to properties dialog for Listing window, bus/signal column, and marker properties.

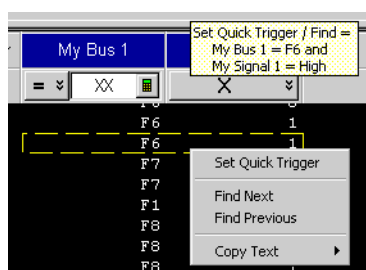
Listing Display Area

To access the Listing display options, right-click anywhere in the display area.



Menu	Description
Undo	Same as Edit>Undo (see page 387).
Go To (see page 236)	
Place Marker> (see page 247)	
Find... (see page 264)	
Find Next (see page 264)	
Find Previous (see page 264)	
Properties... (see page 238)	

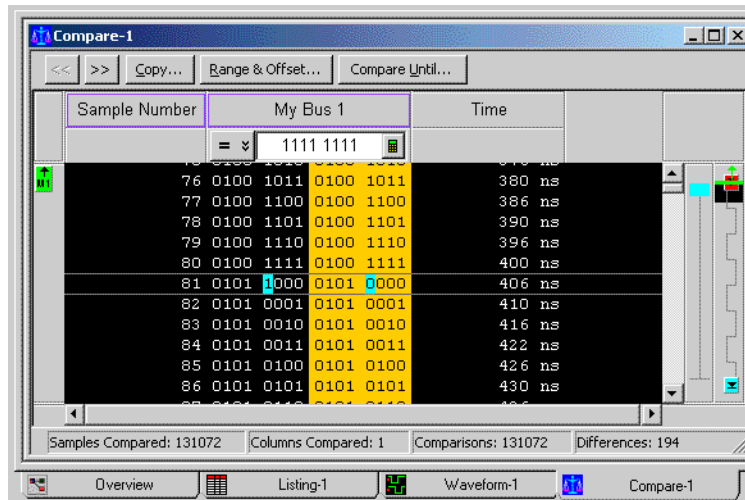
Draw Rectangle in Data



Menu	Description
Set Quick Trigger (see page 143)	Alternative way to set a simple trigger.
Find Next	Data value on top edge of rectangle becomes Find search (see page 264) criteria and next occurrence of that data value is placed at center screen.
Find Previous	Data value on top edge of rectangle becomes Find search (see page 264) criteria and previous occurrence of that data value is placed at center screen.
Copy Text (see page 314)	Copies data as text into the system clip board.

- See Also
- Analyzing Listing Data (see [page 235](#))
 - Setting Up Symbols (see [page 132](#))
 - Marking, and Measuring Between, Data Points (see [page 245](#))
 - To add or delete display windows (see [page 313](#))
 - To turn window tabs on/off (see [page 313](#))

Compare Display Window



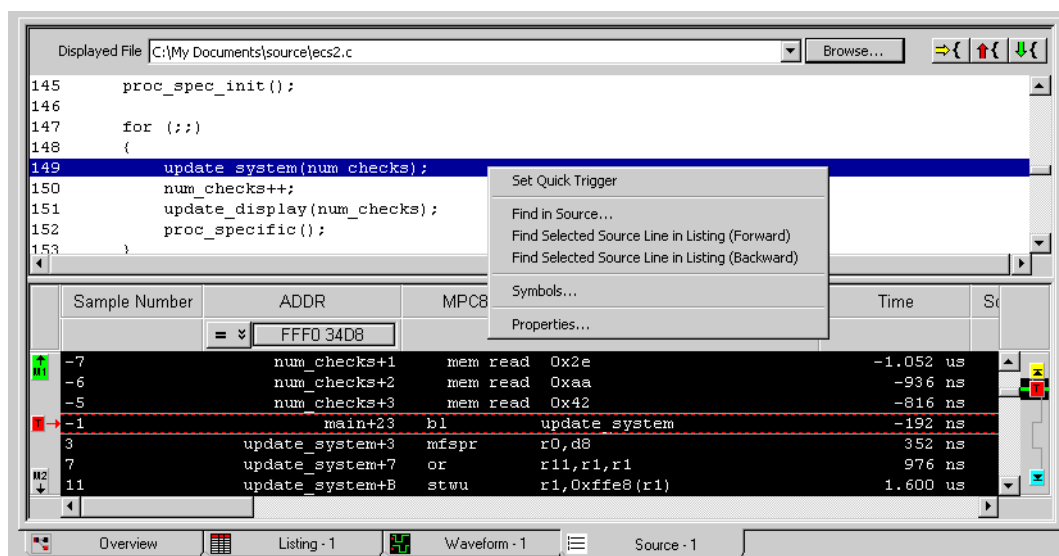
The Compare window lets you compare acquired (input) data to data that has been saved in a reference buffer. The reference data has a colored background, and differences between between the input data and the reference data are highlighted.

The Compare window is accessed through the menu bar's **Window>Compare**. If you have Tabbed Windows (see [page 313](#)) turned on, you can also select a tab at the bottom of the window.

Except for the Compare window's ability to display the differences between captured data and reference data, and its inability display colored data (from the Filter/Colorize tool), the Compare window is just like the Listing window.

- See Also
- Comparing Captured Data to Reference Data (see [page 274](#))
 - To copy data to the reference buffer (see [page 274](#))
 - To find differences in the compared data (see [page 274](#))
 - To compare only a range of samples (see [page 275](#))
 - To offset the reference data (see [page 275](#))
 - To run until a number of compare differences (see [page 275](#))
 - To set Compare window properties (see [page 276](#))
 - Analyzing Listing Data (see [page 235](#))

Source Display Window



The Source window lets you view the high-level source code that is associated with captured data.

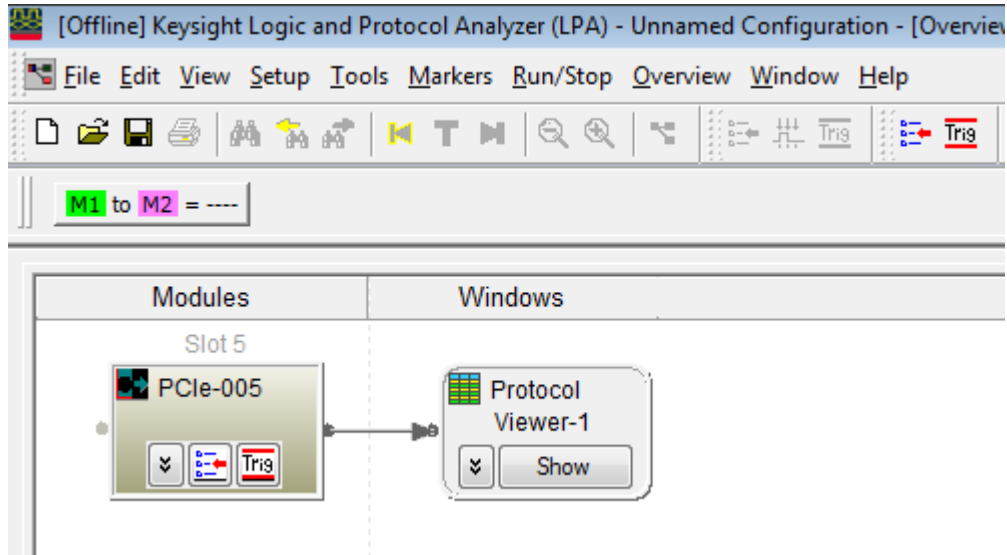
The Source window is accessed through the menu bar's **Window>Source** command. If you have Tabbed Windows (see [page 313](#)) turned on, you can also select a tab at the bottom of the window.

The Source window has two panes: the top pane displays the high-level source code associated with the captured data, and the bottom pane is the same as a Listing window.

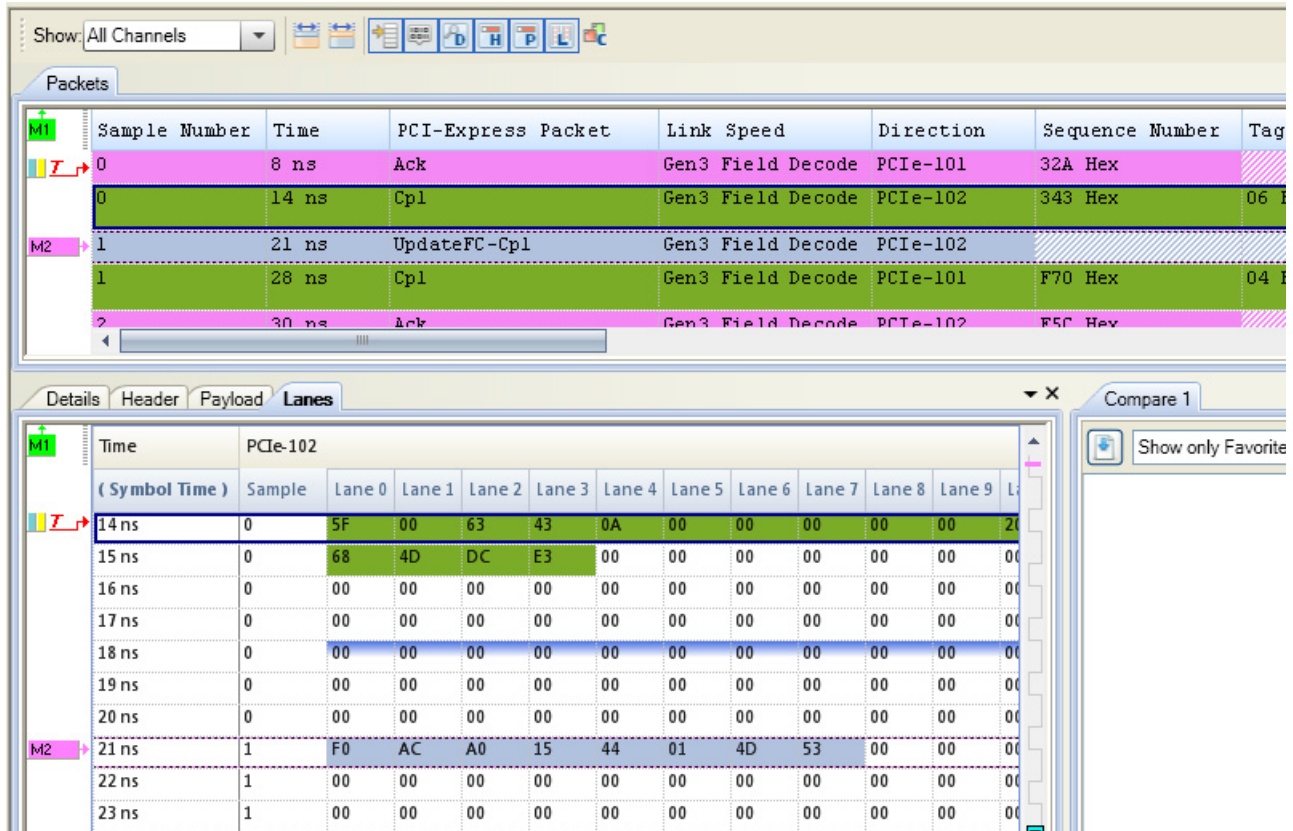
- See Also
- Viewing Source Code Associated with Captured Data (see [page 277](#))
 - To step through captured data by source lines (see [page 277](#))
 - To go to captured data associated with a source line (see [page 278](#))
 - To browse source files (see [page 278](#))
 - To search for text in source files (see [page 279](#))
 - To set a Quick Trigger in the Source window (see [page 145](#))
 - To go to the source line associated with the listing center rectangle (see [page 279](#))
 - To edit the source code directory list (see [page 279](#))
 - To select the correlation bus (see [page 280](#))
 - Changing Source Window Properties (see [page 281](#))
 - Analyzing Listing Data (see [page 235](#))

Protocol Viewer Display Window

You can use a Protocol Viewer window to display data captured by an Keysight instrument module such as the U4301A PCIe Gen3 Analyzer module. The following screen displays an instance of Protocol Viewer added to the U4301A module in the Logic Analyzer application to display the PCI Express data captured by the module.



On clicking the **Show** button on Protocol Viewer, the Protocol Viewer window is displayed with the captured data in the Logic Analyzer application.



Unlike the Listing window, the Protocol Viewer window lets you view summarized and detailed packet information at the same time within two panes.

The upper pane lists the captured packets. On selecting a packet in the upper pane, specific details of that packet are displayed in various tabs of the lower pane.

Adding a Protocol Viewer window instance

You can add an instance of the Protocol Viewer window to a module or a tool in the Overview tab of the Logic Analyzer application. To do this, right-click the module or tool, select **New Window** and then select **Protocol Viewer**.

If you are using the U4301A PCIe Gen3 Analyzer module, an instance of the Protocol Viewer window is automatically added to this module in the Overview tab of the Logic Analyzer application.

Upper pane of Protocol Viewer


The upper pane of Protocol Viewer provides a summarized listing of the captured packets. To know more about how to use the upper pane of Protocol Viewer, refer to the topic "[Viewing the Packet Summary](#)" on page 285.

Lower pane of Protocol Viewer

The lower pane displays the details of a packet that you selected in the upper pane. The packet details are organized in the following tabs.

Menu	Description
Details	This tab displays the details of a selected packet. You can also compare the details of a packet with another packet's details. To know how to use this tab, refer to the topic " To view and compare packet details " on page 294.
Header	This tab displays the header information for a selected packet. To know how to use this tab, refer to the topic " To view a packet header " on page 297.
Payload	This tab displays the payload information for a selected packet. To know how to use this tab, refer to the topic " To view a packet payload " on page 298.
Lanes	This tab displays a vertical listing of a selected packet's data with respect to the logical lanes. To know how to use this tab, refer to the topic " To view a packet's lanes " on page 299.
Traffic Overview	This tab provides an overview of the protocol traffic that is displayed in the upper pane of Protocol Viewer. You can use this tab to get a count of captured packets categorized on the basis of packet types. To know how to use this tab, refer to the topic Viewing the captured PCIe Traffic statistics in U4301A PCIe Gen3 Analyzer module online help.
LTSSM Overview	This tab displays a sequential list of the LTSSM states and their transitions and the packets exchanged during each state. You can use the information displayed in this tab to in verify the link training process and find out reasons for any failure in this process. To know how to use this tab, refer to the topic Viewing LTSSM States and Transitions in the U4301A PCIe Gen3 Analyzer module online help.
Transaction Decode	This tab allows you to compute and view transactions decoded from the captured PCIe traffic. This is a licensed feature. To know how to use this tab, refer to the topic Viewing Decoded Transactions in the <i>U4301A PCIe Gen3 Analyzer module online help</i> .
PCIe Performance Overview	This tab allows you to perform post processing on the captured PCIe traffic and generate an offline performance summary of bus utilization. This is a licensed feature. To know how to use this tab, refer to the topic Viewing Offline Performance Summary in the <i>U4301A PCIe Gen3 Analyzer module online help</i> .

NOTE

You can export the captured packet information to a specified .csv file using Protocol Viewer. You can either export all the data captured and displayed in Protocol Viewer or a specified time/marker based range of data. You do this using the  toolbar button in the Protocol Viewer window.

ONFi Analysis Window

You can use the ONFi Analysis window to display and analyze a NAND Flash memory device's data captured by a Keysight logic analyzer module such as the U4154B or U4164A module.

NOTE

You need the software license option B4661A-4FP Performance Analysis to get the full feature set and capabilities of the ONFi Analysis window. Without this license, it is possible to compute only limited number of NAND Flash memory transactions from the captured data.

Adding an ONFi Analysis Window Instance

You can add an instance of the ONFi Analysis window to any of the following modules/tools in the Logic and Protocol Analyzer GUI:

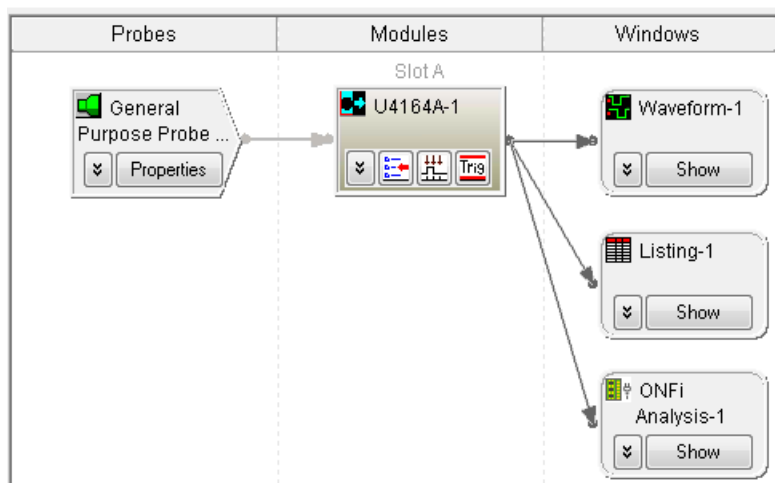
- A logic analyzer module such as U4164A, U4154B, or 16860 series
- A Data Import tool

NOTE

You should NOT attach the ONFi Analysis window to a Filter tool. Filtering can impair some of the memory analysis features if some data such as clock speeds is filtered.

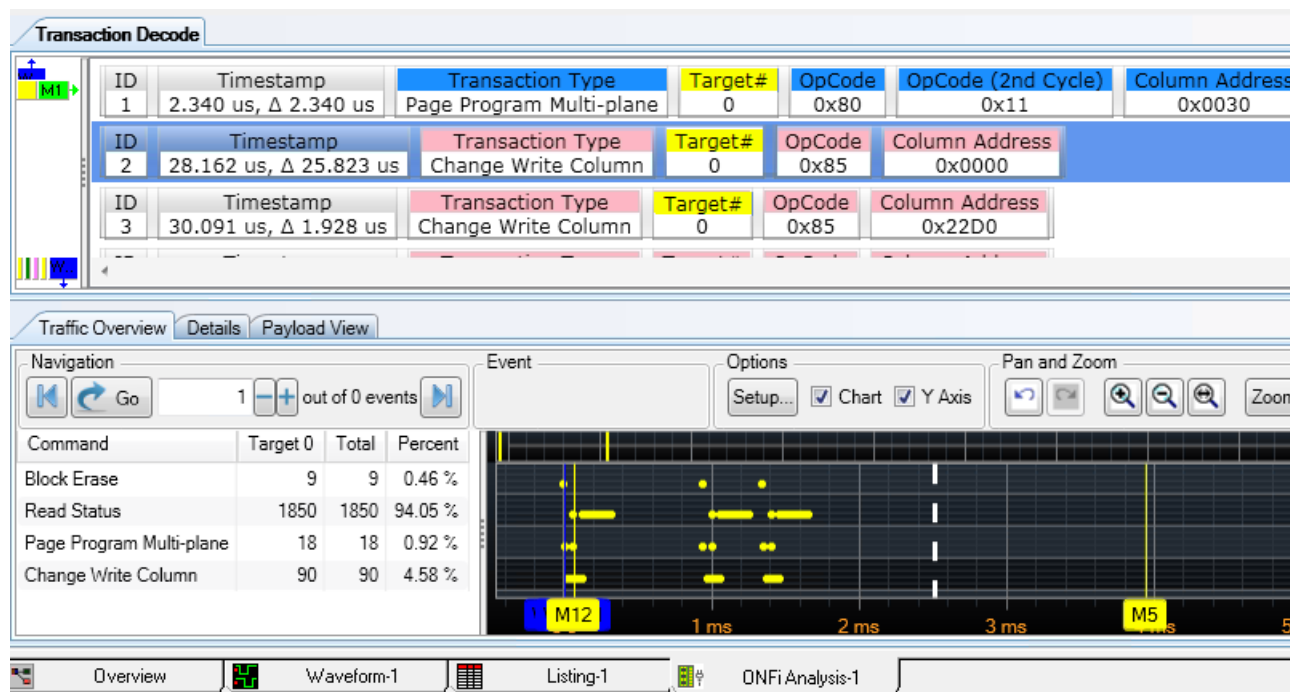
To do this, right-click the module or tool in the Overview tab, select New Window and then select ONFi Analysis. An instance of the ONFi Analysis window is then automatically added to the module in the Overview tab of the Logic and Protocol Analyzer GUI.

The following screen displays an instance of the ONFi Analysis window added to the U4164A module in the Logic and Protocol Analyzer GUI.



On clicking the Show button on the ONFi Analysis instance, the ONFi Analysis window is displayed. In this window, you can initiate the compute process to allow the Logic and Protocol Analyzer software to compute the decoded NAND Flash memory transactions from the captured data.

A sample of the NAND Flash memory data as presented in the ONFi Analysis window is displayed below.



Upper Pane of ONFi Analysis

The upper pane lists the decoded ONFi transactions that the Logic and Protocol Analyzer software computed from the captured NAND Flash memory data.

To know more about how to use the upper pane, refer to the topic Computing and Viewing Decoded ONFi Transactions.

Lower Pane of ONFi Analysis

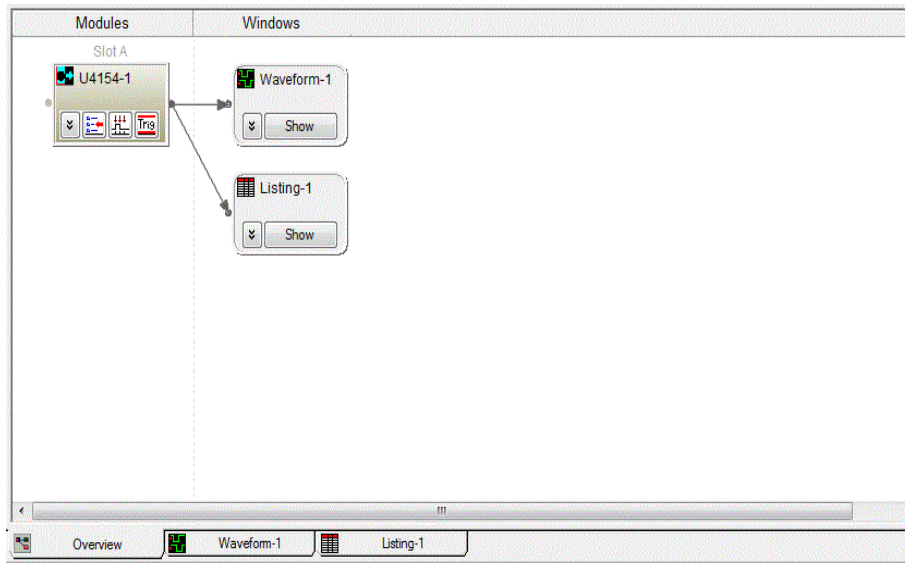
Besides displaying the decoded ONFi transactions in the upper pane, the ONFi Analysis window also provides various tabs in its lower pane.

These tabs provide statistical data and charts from the decoded transactions to help you analyze different aspects of your NAND Flash memory device.

The memory analysis details are organized in the following tabs of the lower pane.

Tab	Description
Traffic Overview	This tab provides a tabular as well as a graphical overview of the captured NAND Flash memory traffic. You can use this tab to get a count of ONFi commands categorized on the basis of command types. To know how to use this tab, refer to the topic Analyzing ONFi Traffic Statistics in the ONFi Analysis Window User Guide.
Details	This tab displays the details of an ONFi transaction selected in the upper pane. You can also visualize an ONFi operation as a set of logically grouped commands in a sequence in this tab. To know how to use this tab, refer to the topic Viewing Details of an ONFi Transaction in the ONFi Analysis Window User Guide.
Payload View	This tab displays the payload for an ONFi transaction selected from the upper pane. To know how to use this tab, refer to the topic Viewing the Payload of an ONFi Transaction in the ONFi Analysis Window User Guide.

Overview Window



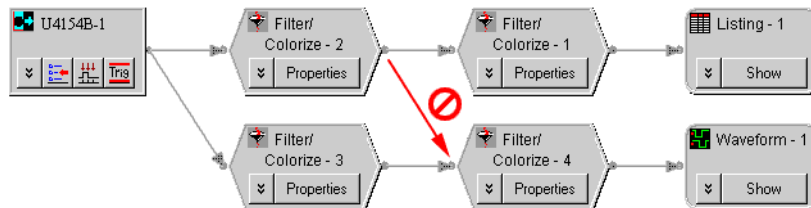
The Overview window lets you specify how the data is sent from the logic analyzer data acquisition module to post-processing tools and display windows.

The Overview window is accessed through the menu bar's **Window>Overview** command. If you have Tabbed Windows (see [page 313](#)) turned on, you can also select a tab at the bottom of the window.

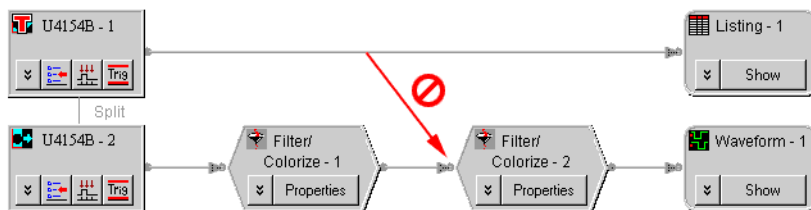
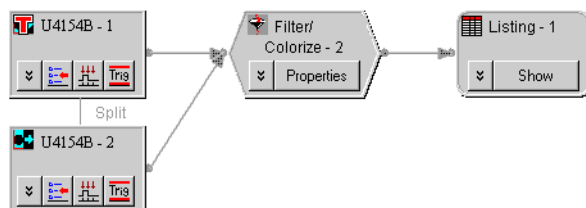
Connection Rules

There are a few rules that govern how you are able to add/connect tools and display data.

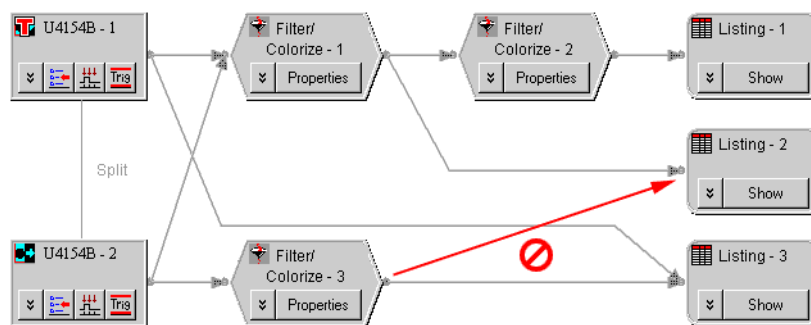
- **Rule 1: Fan out to tools only occurs directly after modules.**



- Rule 2: Fan in to tools occurs only directly after modules.



- Rule 3: Display windows cannot show two versions of the same bus/signal.



- See Also
- Analyzing the Same Data in Different Ways (Using the Overview Window) (see [page 307](#))
 - To open or display the Overview window (see [page 307](#))
 - To add, duplicate, or delete windows and tools (see [page 307](#))
 - To add new windows (see [page 308](#))
 - To delete windows (see [page 308](#))
 - To add new tools (see [page 308](#))

- To delete tools (see [page 309](#))
- To duplicate windows (see [page 309](#))
- To delete connections (see [page 309](#))
- To add connections (see [page 310](#))
- To edit window or tool properties (see [page 310](#))
- To rename windows, tools, and modules (see [page 311](#))
- To redraw the Overview window (see [page 311](#))
- To delete the Overview window (see [page 311](#))

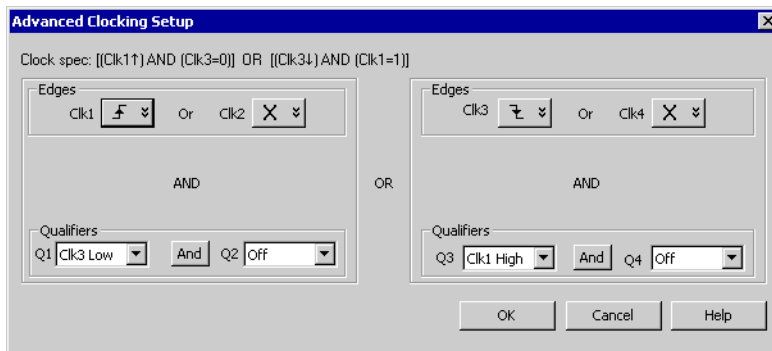
Dialogs

- Advanced Clocking Setup Dialog (see [page 422](#))
- Advanced Trigger Dialog (see [page 424](#))
- Keysight Logic Analyzer Upgrade Dialog (see [page 425](#))
- Analyzer Setup Dialog (see [page 426](#))
- Chat Dialog (see [page 429](#))
- Chat Select Destination Dialog (see [page 430](#))
- Choose a Protocol Family and Bus Dialog (see [page 430](#))
- Create a New Configuration Dialog (see [page 431](#))
- DDR Accumulate Dialog (see [page 431](#))
- E-mail Dialog (see [page 432](#))
- Event Editor Dialog (see [page 433](#))
- Export Dialog (see [page 434](#))
- Export File Selection Dialog (see [page 435](#))
- External Application Setup Dialog (see [page 436](#))
- External Trigger Dialog (see [page 437](#))
- Find Dialog (see [page 438](#))
- Frame/Module Information Dialog (see [page 439](#))
- "General Purpose Probe Set Dialog" (in the online help)
- Instrument Details Dialog (see [page 441](#))
- Import Dialog (see [page 442](#))
- Import Setup Dialog (see [page 442](#))
- Module Mapping Dialog (see [page 444](#))
- Module Skew and System Trigger Dialog (see [page 446](#))
- Netlist Import Dialog (see [page 446](#))
- Offline Startup Options Dialog (see [page 447](#))
- Options Dialog (see [page 447](#))
- Pod Assignment Dialog (see [page 450](#))
- Printing Data Dialog (see [page 451](#))
- Properties Dialog (see [page 452](#))
- Range Properties Dialog (see [page 452](#))
- Recall Trigger Dialog (see [page 453](#))
- Run Properties Dialog (see [page 454](#))
- Search Dialog (see [page 455](#))
- Select Symbol Dialog (see [page 458](#))
- Select System to Use Dialog (see [page 459](#))
- Software Licensing Dialog (see [page 461](#))
- Source Viewer Properties Dialog (see [page 466](#))
- Specify Mapping Dialog (see [page 468](#))
- Status Dialog (see [page 469](#))
- Symbols Dialog (see [page 471](#))
- System Summary Dialog (see [page 472](#))
- Target Control Port Dialog (see [page 475](#))

- TimingZoom Setup Dialog (see [page 476](#))

Advanced Clocking Setup Dialog

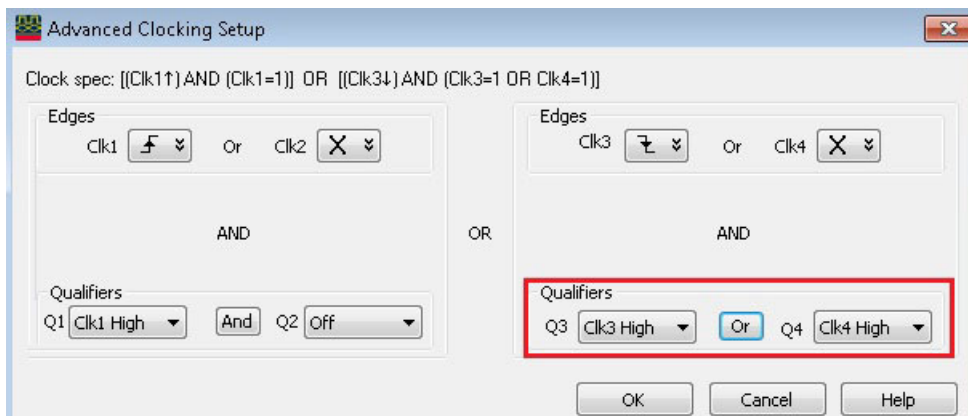
The Advanced Clocking Setup dialog lets you specify more complex clock setups than you can with the normal clock edge and qualifier selections in the Sampling tab of the Setup dialog. This dialog box is available only when you are setting up multiple clocks. To know how to access this dialog box, refer to the topic [To set up advanced clocking](#).



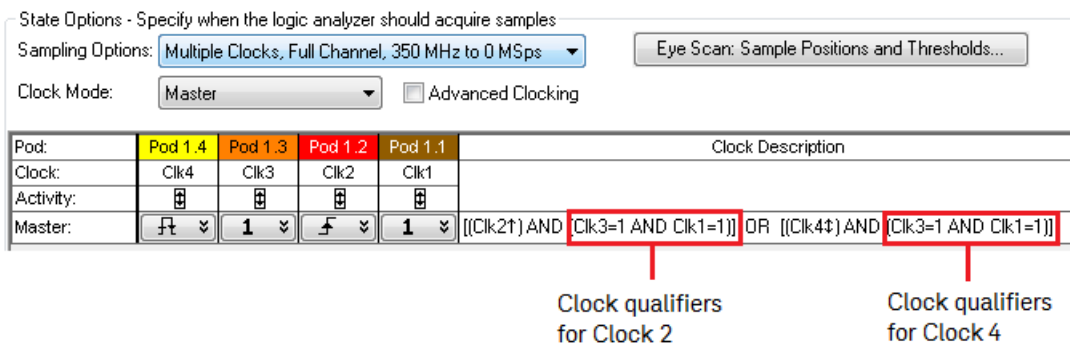
Menu	Description
Clock spec:	A textual description of the clocking setup that you defined in this dialog box.
Edges	Lets you choose from Don't Care, Rising Edge, Falling Edge, or Both Edges for each of the available clock inputs (on Pod 1, 2, 3, and 4). Edges are set in an Ored combination.
Qualifiers	Lets you turn Off clock qualifiers or select Low or High levels from the available clock inputs for each of the clock qualifier resources (Q1-Q4).
And/Or	Lets you toggle the And/OR conditional operators for the two clock qualifiers used for a clock.

Some situations in which this dialog box is particularly useful:

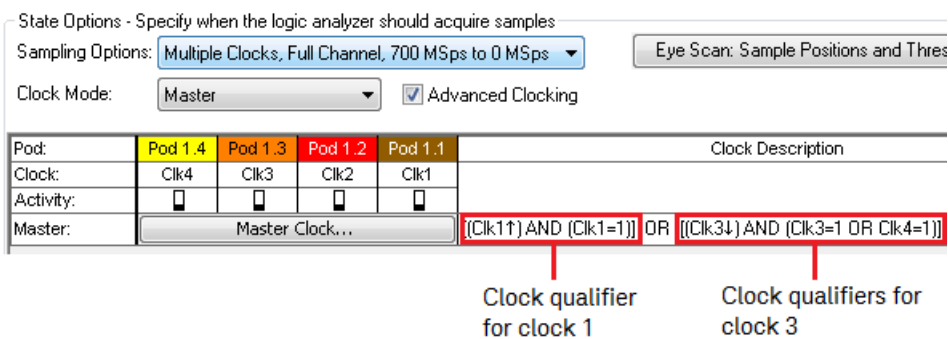
- By default, the clock qualifiers available for multiple clocks are set with the AND conditional operator but you can use the Advanced clocking feature to set these qualifiers with either "AND" or "OR" conditional operators.



- By default, the clock qualifiers that you set are used with all the clocks that you set but you can use the Advanced clocking feature to set different qualifiers for different clocks.



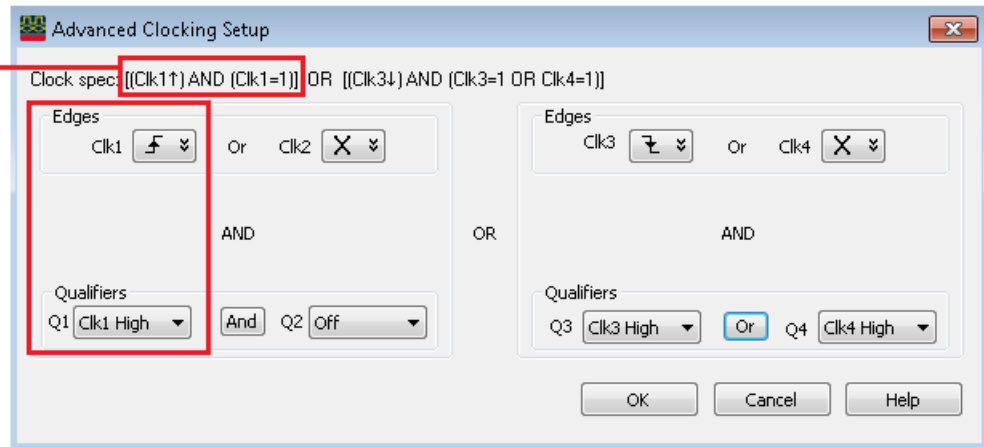
Multiple clocks with same clock qualifiers



Multiple clocks with different clock qualifiers

- By default, a clock channel can be used either as an edge or a qualifier in a multiple clock description but you can use the Advanced clocking feature to use a clock channel both as an edge and a qualifier in the same clock description.

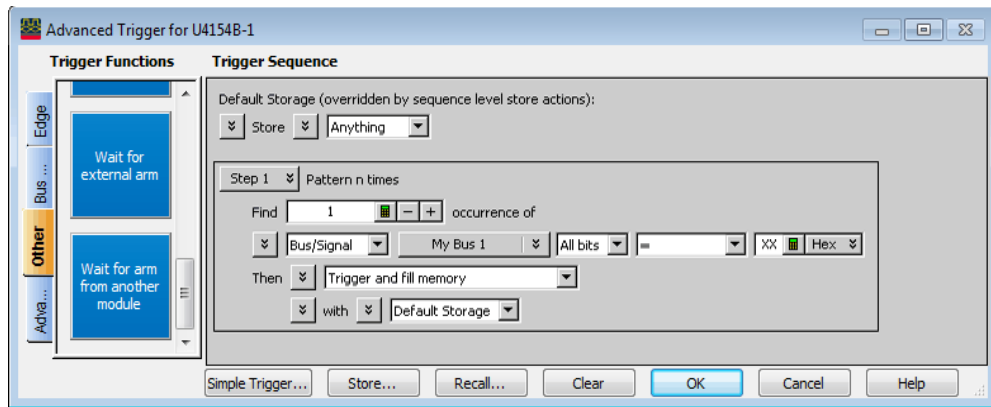
Clk 1 used both as an edge as well as a qualifier



- See Also
- To set up advanced clocking (see [page 127](#))
 - Pod and Channel Naming Conventions (see [page 361](#))

Advanced Trigger Dialog

The Advanced Trigger dialog lets you set up complex trigger specifications that cannot be set up with simple triggers (for example, you can trigger on a sequence of events in the device under test).



Menu	Description
Trigger Functions	Trigger functions are pre-defined trigger setups for common measurements. Trigger functions are drag-and-dropped into the Trigger Sequence area in the desired order; then, you fill-in the fields that specify the <i>events</i> to look for in the sampled data and the <i>actions</i> to take when the events are found.
Trigger Sequence	The trigger sequence describes the steps to take when searching for a sequence of events that will trigger the logic analyzer. <i>Default storage</i> specifies which samples to store unless there are overriding <i>storage control</i> actions within the trigger sequence steps.
Simple Trigger...	Opens an information dialog that explains how to specify simple triggers (see page 146).

Menu	Description
Store...	Opens the Store Trigger dialog for saving a trigger sequence setup as a favorite or saving it to a file.
Recall...	Opens the Recall Trigger dialog (see page 453) for recalling favorite, recently used, and stored trigger sequence setups.
Clear	Clears the current trigger sequence and sets up a default trigger sequence.

- See Also
- [Specifying Advanced Triggers \(see page 152\)](#)
 - [Trigger Functions \(see page 478\)](#)
 - [Specifying Simple Triggers \(see page 146\)](#)
 - [Storing and Recalling Triggers \(see page 194\)](#)

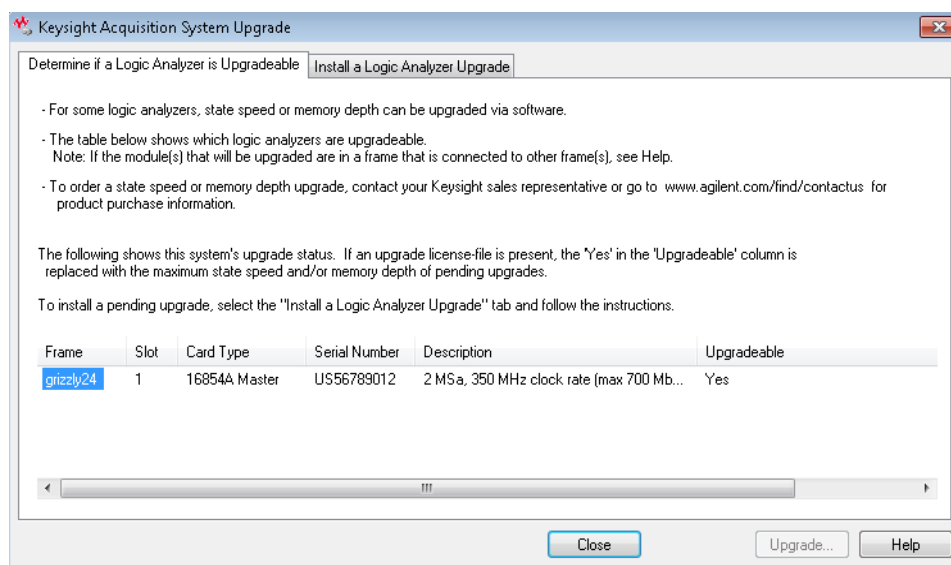
Keysight Logic Analyzer Upgrade Dialog

NOTE

When installing licensed hardware upgrades, you must run the *Hardware Update Utility* program on the frame that contains the cards you want to upgrade. In other words:

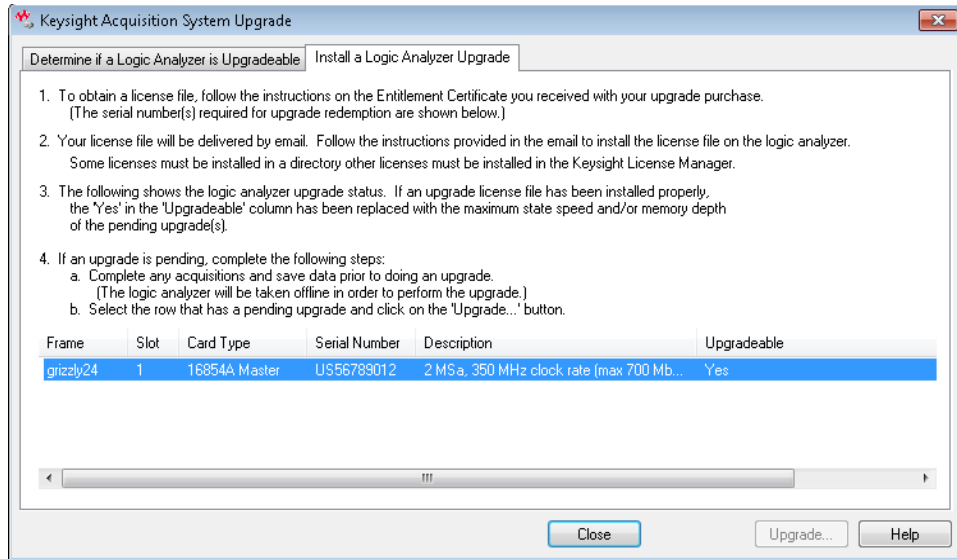
- In a multiframe logic analysis system, you must run the *Hardware Update Utility* program on each frame that has cards to be upgraded.
- You cannot install module upgrades over a remote connection (including remote connections via Remote Desktop, NetOp, or RealVNC).

Determine if a Logic Analyzer is Upgradeable Tab



This tab lists the logic analyzers and other cards in a frame and shows you whether they are upgradeable or if an upgrade is pending.

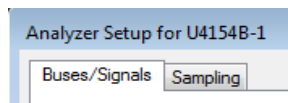
Install a Logic Analyzer Upgrade Tab



Menu	Description
Upgrade...	Performs the pending upgrade for the selected card.

See Also • Installing Licensed Hardware Upgrades (see [page 139](#))

Analyzer Setup Dialog



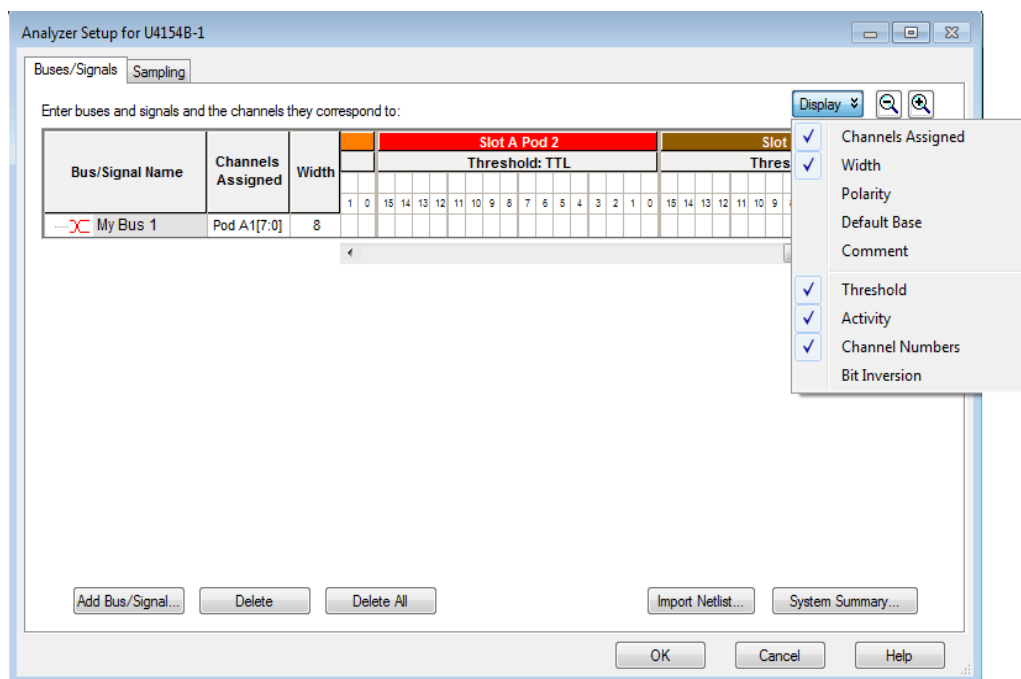
The Analyzer Setup dialog is accessed through the main menu's **Setup>(Logic Analyzer Module)>Bus/Signal...** or **Setup>(Logic Analyzer Module)>Timing/State (Sampling)...** commands.

The dialog consists of the following two tabs.

- **Buses/Signals** - The Buses/Signals tab is used to map bus and signal names in the interface to the pod and channel connections of the probes. Also, you can set a pod threshold, and assign a default number base and polarity to the bus or signal. See Buses/Signals Tab (see [page 426](#)).
- **Sampling** - The Sampling tab is used to name the analyzer, and select and configure the acquisition mode. In the timing acquisition mode, you set the channel width and sampling rate. In the state acquisition mode you configure the state clocks and qualifiers. See Sampling Tab (see [page 428](#)).

Buses/Signals Tab

The Buses/Signals tab is used to map (assign) bus and signal names in the interface to the pod and channel connections of the probes. You also use the Buses/Signal tab to set up thresholds, polarity, default number base, and enter user comments.



The Buses/Signals tab is accessed through the menu bar's **Setup>(Logic Analyzer Module)>Bus/Signal...** command.

Through the **Display** button, you can select what bus/signal setup information is displayed.

The bus and signal icons in the **Bus/Signal Name** column are normally red, but they turn gray if the bus/signal is locked by an inverse assembler.

Read Only Options

The following fields are read only and cannot be edited. The display of these items can be turned on/off under the **Display** button.

Menu	Description
Width	The Width column displays the number of assigned channels on each bus.
Activity	The Activity row displays the type of signal activity on each channel. <ul style="list-style-type: none"> Low bar = A stable low level. High bar = A stable high level. Transition arrows = An active signal transition between low and high.
Channel Numbers	The Channel Numbers row displays pod channel numbers

NOTE

In previous versions of the *Keysight Logic Analyzer* application, the Buses/Signals setup tab had a Define Probes... button; now, probes are defined differently (see "To define probes" (in the online help)).

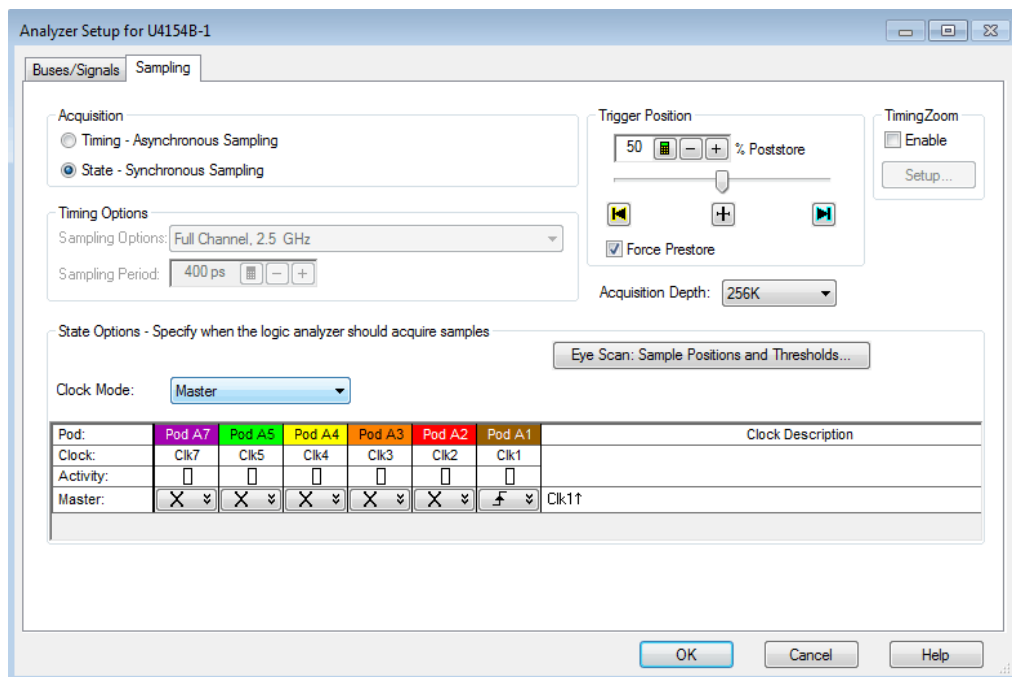
NOTE

If you enable the Advanced Probe settings for U4154A/B logic analyzer, a button named **APS** is displayed in the Buses/Signals tab. You use this button to enable or disable the peaking for the probing system used for these logic analyzers. See [“Changing Advanced Probe Settings for Logic Analyzers”](#) on page 563.

- See Also
- Defining Buses and Signals (see [page 83](#))
 - To add a new bus or signal (see [page 84](#))
 - To delete a bus or signal (see [page 84](#))
 - To rename a bus or signal (see [page 85](#))
 - To assign channels in the default bit order (see [page 88](#))
 - To assign channels, selecting the bit order (see [page 89](#))
 - To define buses and signals by importing netlist files (see [page 90](#))
 - To reorder bits by editing the Channels Assigned string (see [page 91](#))
 - To set the default number base (see [page 93](#))
 - To set polarity (see [page 94](#))
 - To add user comments (see [page 94](#))
 - To add a folder (see [page 95](#))
 - To alias a bus/signal name (see [page 95](#))
 - To sort bus/signal names (see [page 95](#))
 - Pod and Channel Naming Conventions (see [page 361](#))
 - Why Are Pods Missing? (see [page 362](#))
 - Logic Analyzer Notes (see [page 536](#))

Sampling Tab

The Sampling tab is access through the menu bar's **Setup>(Logic Analyzer Module)>Timing/State (Sampling)...** command. The Sampling setup tab is used to select and configure the sampling mode.

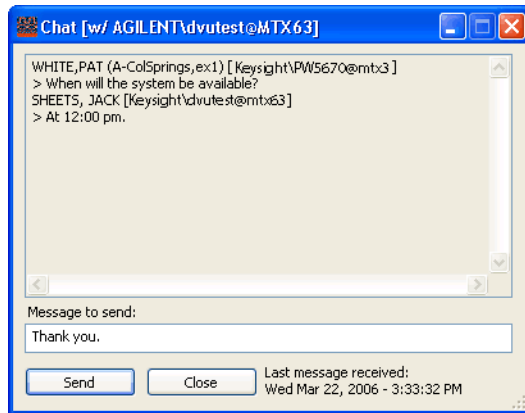


Menu	Description
Acquisition	Lets you select the Timing or State acquisition mode (see Choosing the Sampling Mode (see page 96)).
Timing Options	When the Timing acquisition mode is selected, you can specify its options (see Selecting the Timing Mode (Asynchronous Sampling) (see page 96)).
State Options	When the State acquisition mode is selected, you can specify its options (see Selecting the State Mode (Synchronous Sampling) (see page 98)).
Options	Lets you specify options that apply to both the Timing and State acquisition modes (see To specify the trigger position (see page 128) and To set acquisition memory depth (see page 129)).
TimingZoom	Lets you turn the timing zoom feature on or off and specify its settings (see Using Timing Zoom (see page 129)).

- See Also
- Choosing the Sampling Mode (see [page 96](#))
 - Logic Analyzer Notes (see [page 536](#))

Chat Dialog

The Chat dialog lets you enter and send messages to other logic analysis system users.

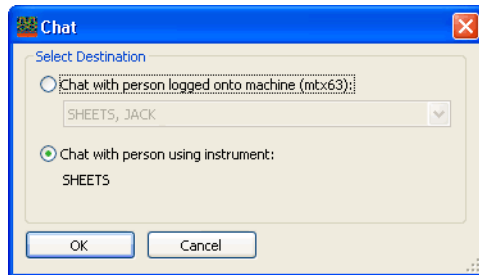


Menu	Description
Message to send	Lets you enter a message to send.
Send	Sends the message.
Close	Closes the dialog and the chat session.

- See Also
- Select System to Use Dialog (see [page 459](#))
 - Chat Select Destination Dialog (see [page 430](#))

Chat Select Destination Dialog

The Chat Select Destination dialog lets you select either the person logged into or the person connected to the logic analysis system.

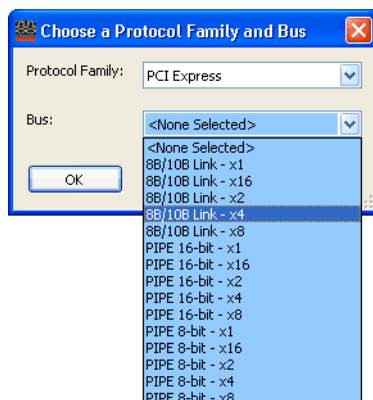


Clicking **OK** opens the Chat dialog (see [page 429](#)) where you can enter and send your message.

- See Also
- Select System to Use Dialog (see [page 459](#))

Choose a Protocol Family and Bus Dialog

The Choose a Protocol Family and Bus dialog lets you select a protocol family and bus for the "Find a packet" trigger function.

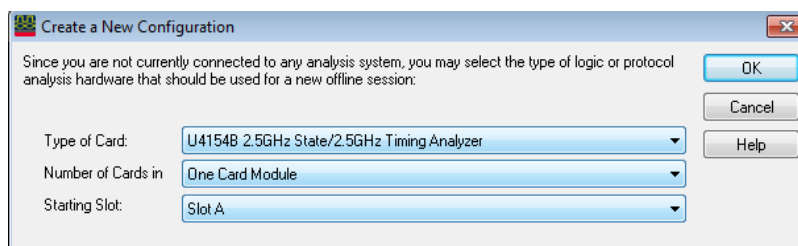


Menu	Description
Protocol Family	Lets you select the protocol family.
Bus	Lets you select the type of bus within the protocol family.

- See Also
- Find a packet (see [page 496](#)) trigger function
 - To specify packet events (in "Find a packet" trigger function) (see [page 164](#))

Create a New Configuration Dialog

The Select Offline Hardware dialog appears when you are in *offline mode* and you choose the **File>New** command to create a new logic analyzer configuration file. This dialog lets you specify the type of logic analyzer hardware to model in the configuration file.

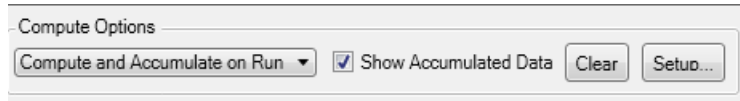


Menu	Description
Type of Card	Selects the type of logic analysis hardware to use in the new offline configuration file.
Number of Cards in	Specifies the number of cards in the hardware module.
Starting Slot	Selects the starting slot of the cards used in the new offline configuration file.

- See Also
- Offline Analysis (see [page 217](#))

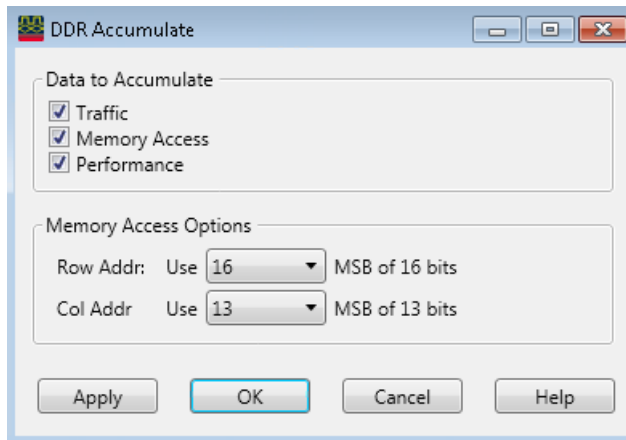
DDR Accumulate Dialog

This dialog is accessed by clicking the Setup button from the Compute Options groupbox in the DDR/LPDDR Memory Analysis Window.



When you select the Compute and Accumulate on Run compute option, the computed data from each consecutive run of logic analyzer is accumulated for the applicable tabs of the DDR/LPDDR Memory Analysis window.

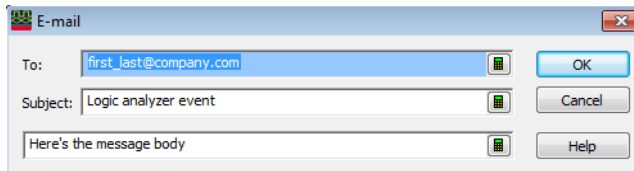
You use the **DDR Accumulate** dialog box to configure options for this accumulated data.



Option	Description
Data To Accumulate	By default, the computed data from each consecutive run of logic analyzer can be accumulated only for the Traffic Overview, Memory Access Overview, and Performance Overview tabs of the DDR/LPDDR Memory Analysis window. If you do not want data to be accumulated for any of these tabs, you can deselect its checkbox.
Memory Access Options	The options in this groupbox are applicable for data accumulation in the Memory Access Overview tab only.

See Also [Computing Decoded Memory Transactions and Analysis Data \(in the Memory Analysis Window User Guide\)](#)

E-mail Dialog



This dialog allows you to configure the e-mail settings that the Keysight Logic and Protocol analyzer application uses to compose e-mail messages for the specific situations for which you have configured the "send e-mail" feature. For instance, to send the specified e-mail message to the specified recipient when a trigger event occurs for which you have configured "send e-mail" as the trigger action.

You can access the E-mail dialog from the following dialogs that support the "send e-mail" feature.

- Advanced Trigger dialog (See - To specify a trigger sequence step's goto or trigger action on [page 166](#))
- Interval Properties dialog (See - To create a new time interval measurement on [page 251](#))
- Interval Properties dialog (See - To create a new sample interval measurement on [page 252](#))
- Value Properties dialog (See - To change a marker's position property on [page 248](#))
- Compare Properties dialog (See - To run until a number of compare differences on [page 275](#))

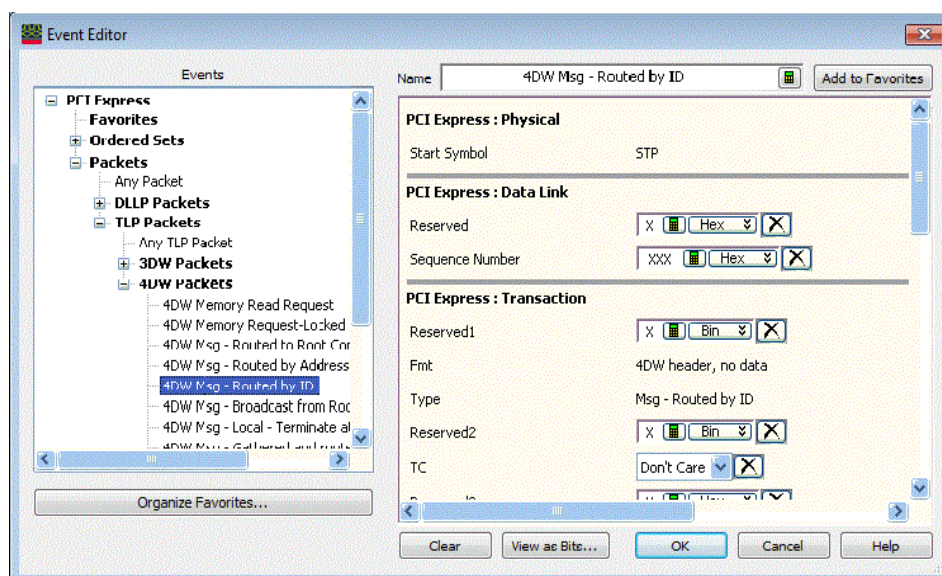
NOTE

Ensure that you have specified the SMTP mail server name and sender's email address in the **Options** dialog for the "send e-mail" feature to work. The Keysight Logic and Protocol Analyzer application communicates directly with this e-mail server to send the e-mail.

Menu	Description
To	Address(es) to which e-mail will be sent. You can specify multiple recipients by separating each e-mail address with a semicolon (;).
Subject	Subject of the e-mail.
(message)	Text of the message.

Event Editor Dialog

The Event Editor dialog lets you specify packet events in the "Find a packet" trigger function.

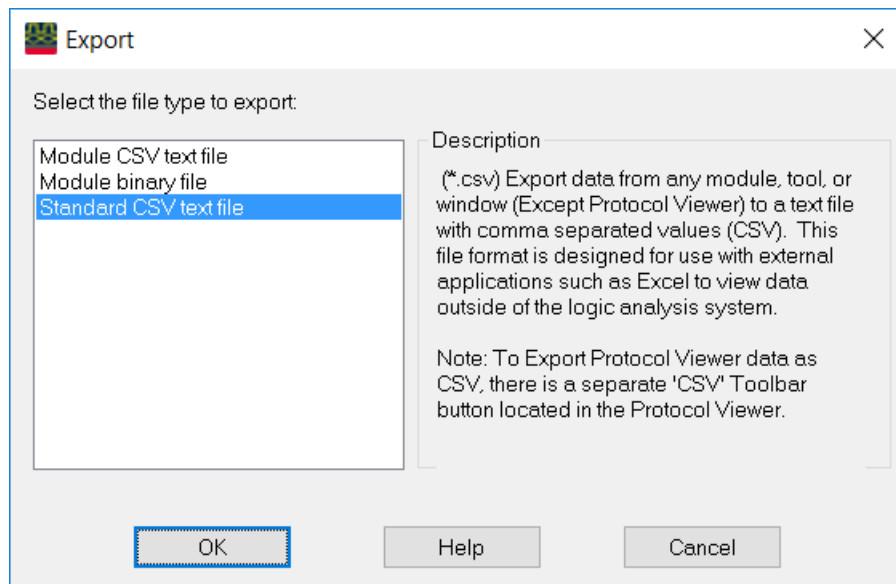


Menu	Description
Events	Hierarchically lists protocol event types that you can select.
(Fields)	(On the right side of the dialog). Lets you enter, select, or clear (✕) packet field values.
Name	Lets you enter or modify the name of the packet event.
Add to Favorites	Adds the packet event to the favorites list.
Organize Favorites...	Opens the Organize Favorites dialog for organizing the packet event favorites list (see To organize favorite packet events (see page 165)).
Clear	Clears all packet field values.
View as Bits...	Opens the View as Bits dialog for viewing the packet event in a format similar to specifications documents (see To view a packet event as bits (see page 165)).

- See Also
- Using the Packet Event Editor (see [page 164](#))
 - Find a packet (see [page 496](#)) trigger function
 - To specify packet events (in "Find a packet" trigger function) (see [page 164](#))
 - To find packet patterns in the captured data (see [page 267](#))
 - "To specify packet patterns to filter" (in the online help)

Export Dialog

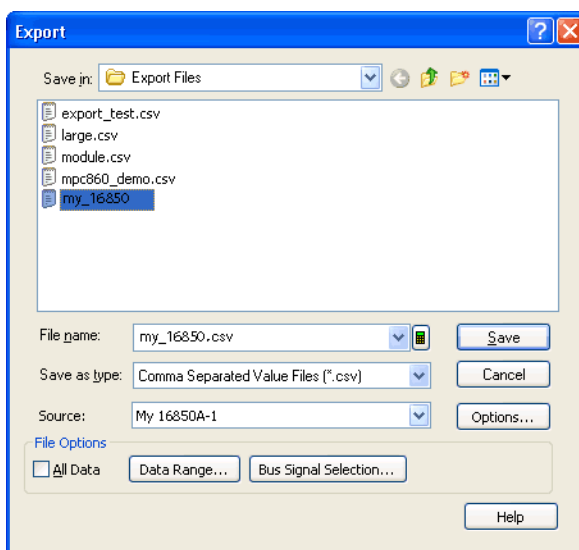
The Export dialog lets you export certain kinds of data into the *Keysight Logic Analyzer* application.



Menu	Description
Module CSV text file	Lets you export module data only to a comma separated value (CSV) format text file. This file type is for post-processing tools (see "To export data to module CSV format files" on page 201).
Module binary file	Lets you export module data only to a binary format file (see "To export data to module binary (ALB) format files" on page 203).
Standard CSV text file	Lets you export data from modules, tools, or windows to a comma separated value (CSV) format text file (see "To export data to standard CSV format files" on page 199).

Export File Selection Dialog

The Export file selection dialog lets you select the file to which data is exported, the data source, the export options, the range of data samples, and the buses/signals to export.

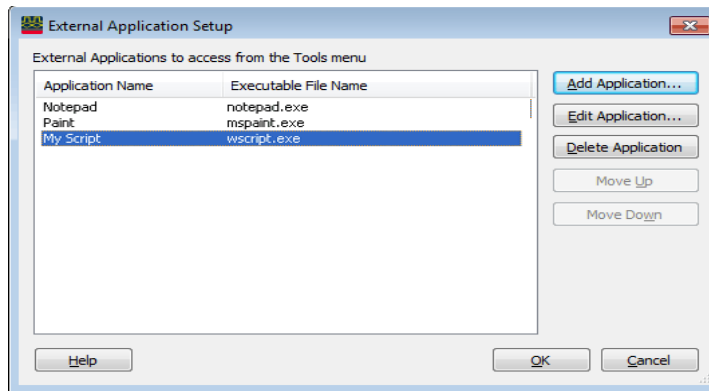


Menu	Description
File name	Lets you select the file to which data is exported.
Save as type	The appropriate type is already selected, based on the file type selected in the previous Export dialog (see page 434).
Source	Lets you select the data source. The list of sources you can select from is determined by the file type selected in the previous Export dialog (see page 434).
Options...	Opens the File Export Options dialog where you can select the appropriate options for the file type selected in the previous Export dialog (see page 434).
All Data	When checked, data from all samples and all buses/signals is exported.
Data Range...	When All Data is not checked, this button opens a Range Properties dialog for selecting the range of data samples to export.
Bus Signal Selection...	When All Data is not checked, this button opens a Range Properties dialog for selecting the buses/signals to be included in the export.

- See Also
- To export data to standard CSV format files (see [page 199](#))
 - To export data to module CSV format files (see [page 201](#))
 - To export data to module binary (ALB) format files (see [page 203](#))
 - "Exporting Vector Sequences to CSV Format Files" (in the online help)

External Application Setup Dialog

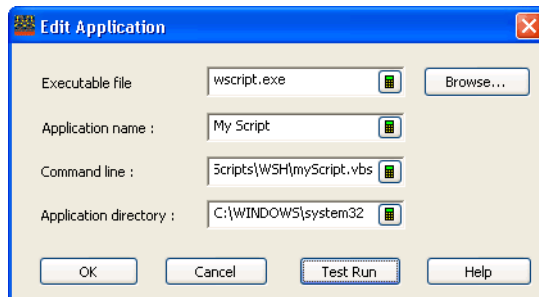
The External Application Setup dialog lets you add, edit, arrange, or remove items from the **Tools>External Applications>** menu.



Menu	Description
Add Application...	Opens the Add Application dialog (see page 436) for specifying a new menu item's parameters.
Edit Application...	Opens the Edit Application dialog (see page 436) for changing the selected menu item's parameters.
Delete Application	Deletes the selected application from the menu item list.
Move Up Move Down	Moves the selected application up or down within the menu item list.

External Application Add/Edit Dialog

The Add/Edit Application dialog lets you enter or modify the parameters for an item in the **Tools>External Applications>** menu.



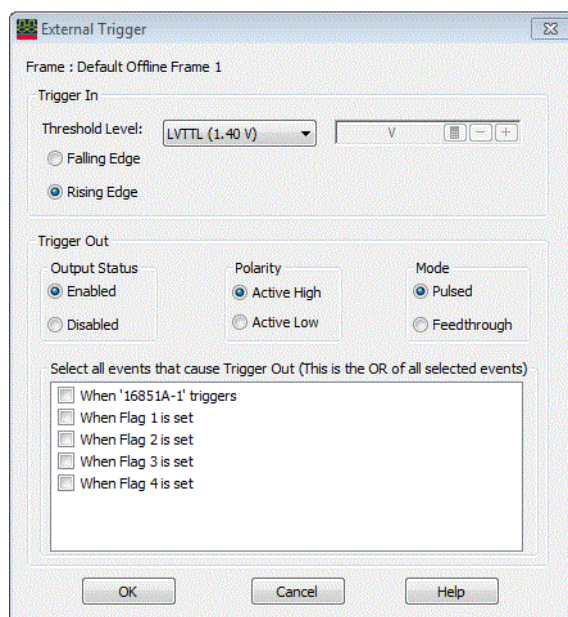
Menu	Description
Executable file	The name of the file to execute when the menu item is selected.
Application name	The name that appears on the menu item.
Command line	Command line options for the executable file.
Application directory	The directory that contains the executable file.
Test Run	For quick testing, this executes the application with the given parameters.

External Trigger Dialog

There are **Trigger In** and **Trigger Out** BNC connectors located on the 16850-series logic analysis system. These BNC connectors are used to connect the analyzer to an external instrument and either send or receive a trigger signal.

The External Trigger dialog is used for setting up triggers between the logic analyzer or logic analysis system and other, external instruments.

The 16850-series External Trigger dialog looks like:



Trigger In Lets you trigger the logic analyzer from another source. You can select whether a rising or falling edge indicates a trigger.

Input Signal Characteristic	16850-Series
Input signal level:	Selectable, ± 5 V Max.
Minimum signal amplitude:	200 mV

Trigger Out Sends a signal to another device when the logic analyzer triggers. You can select whether the trigger will appear as a rising or falling edge.

The trigger out signal is designed to drive a 50 Ohm load. It is recommended that for good signal quality, the trigger out signal be terminated in 50 Ohms to ground.

With a 16850-series logic analysis system frame, you can:

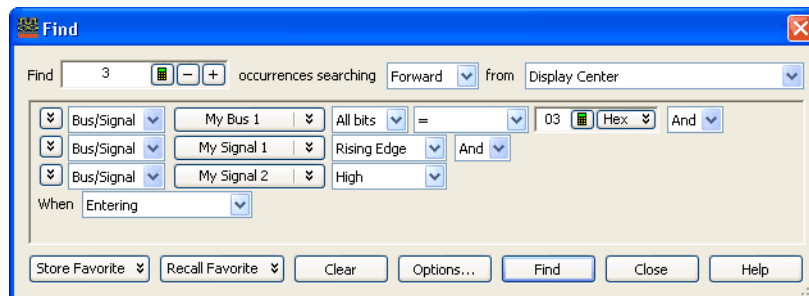
- Enable or disable (3-state high-impedance) the output.
- Choose the polarity of the output.
- Choose whether the output mode is **Pulsed** or **Feedthrough** (for observing flag settings).
- Select the events that will cause a trigger signal to be output.

Output Signal Characteristic	16850-Series
VOH (output high level):	>2.0 V (3.3 V avg.)
VOL (output low level):	<0.5 V (0 V avg.)
Pulse width:	Approx. 80-160 ns

- See Also
- To trigger other instruments - trigger out (see [page 191](#))
 - To trigger analyzer from another instrument - trigger in (see [page 192](#))

Find Dialog

The Find dialog lets you search for patterns in captured data. You can qualify your search by specific bits, data patterns, equality, and range operators. The search result is placed at the center of the display.



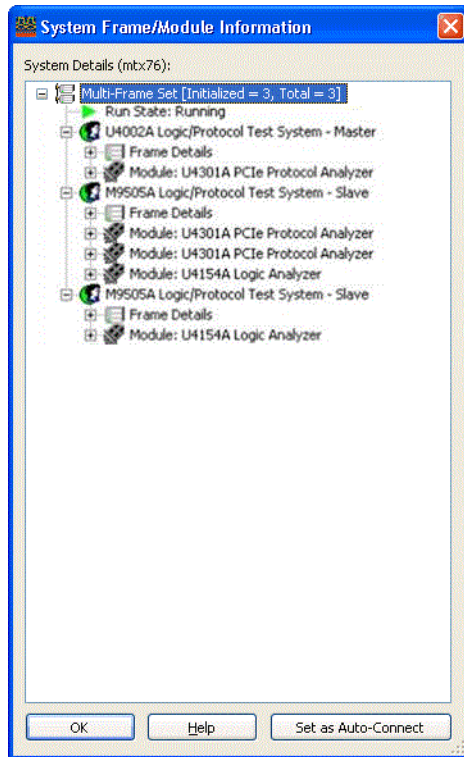
Menu	Description
Find N occurrences	Specifies the number of occurrences to search for.
searching	Specifies whether to search Forward or Backward.
from	Specifies the starting location (Display Center, Beginning Of Data, End Of Data, Trigger, or a marker).

Menu	Description
(pattern event)	<p>Specifies the pattern event you wish to locate.</p> <p>In addition to the usual pattern matching operators (=, !=, <, >, <=, >=, In Range, and Not In Range), there are three additional operators you can use:</p> <ul style="list-style-type: none"> · Entering – the first sample of one or more consecutive samples that match the pattern. (By comparison, the "=" equals operator considers every sample that matches the pattern as an occurrence.) · Exiting – the sample after one or more consecutive samples that match the pattern. · Transitioning – entering or exiting one or more consecutive samples that match the pattern.
When	<p>A find qualifier (which further qualifies the find criteria with a time duration or other operator):</p> <ul style="list-style-type: none"> · Present · Not Present · Present> (time duration) · Present>= (time duration) · Present< (time duration) · Present<= (time duration) · Present for Range (of time) · Not Present for Range (of time) · Entering · Exiting · Transitioning <p>The find qualifiers Present>, Present>=, Present<, Present<=, Present for Range, and Not Present for Range let you specify a time duration. This means the find event specified in the expression area will be found based upon the given time and operator.</p> <p>The other qualifiers (Present, Not Present, Entering, Exiting, and Transitioning) do not allow a time duration.</p>
Store Favorite	Lets you store favorite find patterns.
Recall Favorite	Lets you recall or delete favorite find patterns.
Clear	Clears the current find pattern.
Options...	Opens the Find Options dialog that lets you specify "found" marker placement.
Find	Performs the find without closing the Find dialog.
Close	Closes the Find dialog.

- See Also
- To quickly find bus signal patterns (see [page 264](#))
 - To find bus/signal patterns in the captured data (see [page 265](#))
 - To find packet patterns in the captured data (see [page 267](#))
 - To find complex patterns in the captured data (see [page 270](#))
 - To store, recall, or delete favorite find patterns (see [page 271](#))
 - To specify "found" marker placement (see [page 272](#))

Frame/Module Information Dialog

The Frame/Module Information dialog displays detailed information about a logic analysis system *frame* (see [page 615](#)).



Menu	Description
System Details	<p>Displays the detailed information about the logic analysis system. This includes both information on the frame as a whole and information on each module.</p> <p>If the frame is participating in a multiframe set (as is the case in the example above), frame and module details on each frame is shown. In the example above, the in-use icon shows a user is currently using the multiframe set.</p> <p>If module depth or speed is upgradeable (via hardware licensing), module upgrade information is also provided.</p> <p>Under the "Frame Details" category, you can find information about the frame's software version, host IP address, host name, whether or not the frame is password protected from remote-access, etc.</p>
Set as Auto-Connect	<p>Selects the logic analysis system frame as the one to use when the <i>Keysight Logic Analyzer</i> application starts. The words "auto-connect" will show up next to the frame selected as an auto-connect in the Select System to Use dialog (see page 459).</p> <p>This button will change to "Clear Auto-Connect" if this frame is already set as an auto-connect.</p>
Clear Auto-Connect	<p>When pressed, specifies that the logic analysis system frame not be automatically connected to when the <i>Keysight Logic Analyzer</i> application starts.</p>

- Notes:
- When no auto-connect is established, the *Keysight Logic Analyzer* application attempts to go online with the local frame or starts offline if there is no local frame.
 - The auto-connect frame selection is stored on a per-user basis—so each logged on user can establish a different auto-connect frame.

- If you are running the *Keysight Logic Analyzer* application on a standalone instrument or you are running the *Keysight Logic Analyzer* application on your PC with hosted instruments attached to your PC, the *Keysight Notification Center Icon* will be present on the taskbar (lower, right-hand corner of your desktop). This icon can be right-clicked to open an *instrument details* dialog showing the same type of information as the above dialog—but only for the local hardware.

See Also • To view logic analysis system details (see [page 64](#))

Instrument Details Dialog

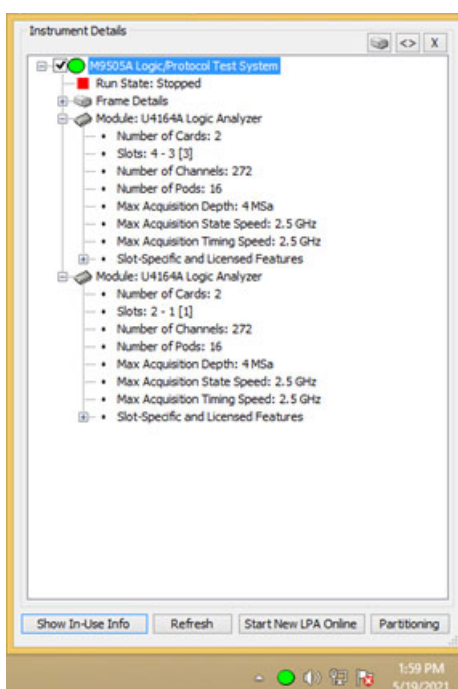
To access this dialog, use the Keysight Notification Center icon present on the Windows taskbar (lower, right-hand corner of your desktop).



The **Instrument Details** dialog box is displayed. This dialog box displays information on the modules currently installed in the chassis.

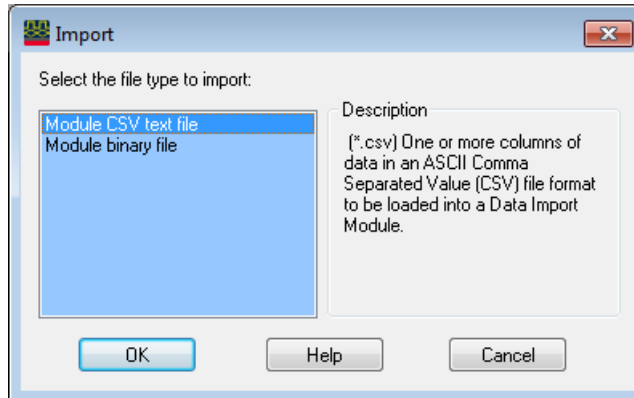
If you have created a multi-card set by interconnecting modules (cards) using cables, the dialog box also displays information about the number of cards in each set and the chassis slots in which you have installed these cards. For instance, in the screen below, there are two U4164A multi-card sets and each set has two U4164A modules. To know more, refer to ["Creating a Logic Analyzer Multi-card Set"](#) on page 74.

Once a multi-card set is created, you can use the **Partitioning** button in this dialog box to reconfigure (repartition) this set by including/excluding the interconnected modules from the set without physically connecting/disconnecting these modules. Refer to the topic ["Repartitioning a U4164A Multi-card Set"](#) on page 75 to know more.



Import Dialog

The Import dialog lets you import certain kinds of data into the *Agilent Logic Analyzer* application.



Menu	Description
Module CSV text file	Lets you import data from a CSV format text file into the logic analysis system using a <i>data import module</i> (see To create a data import module (see page 400)).
Module binary file	Lets you import data from a module binary (ALB) format file into the logic analysis system using a <i>data import module</i> (see To create a data import module (see page 400)).

Import Setup Dialog

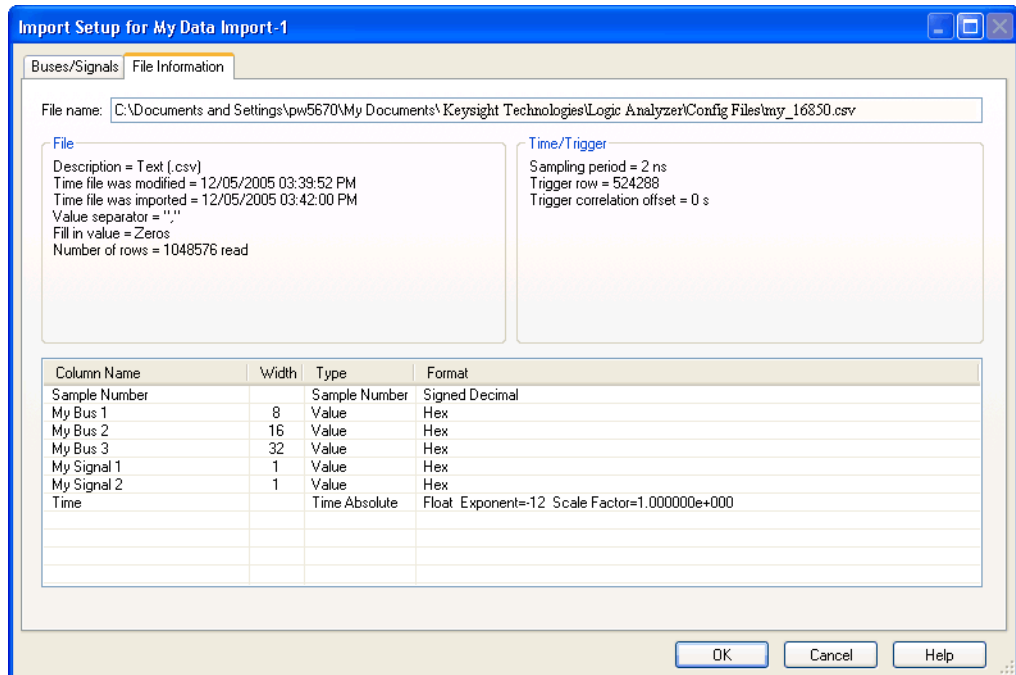
The Import Setup dialog is accessed through the menu bar's **Setup>(Data Import Module)>Bus/Signals...**

The dialog consists of the following two tabs.

- **Buses/Signals** - The Buses/Signals tab is used to edit bus and signal names in the data import module. Also, you can assign a default number base and polarity to the bus or signal. See [Buses/Signals Tab \(see page 442\)](#).
- **File Information** - The File Information tab describes the contents of the file that has been imported. See [File Information Tab \(see page 443\)](#).

Buses/Signals Tab

The Buses/Signals tab is used to edit bus and signal names in the data import module. You also use the Buses/Signal tab to set the polarity, set the default number base, and enter user comments.

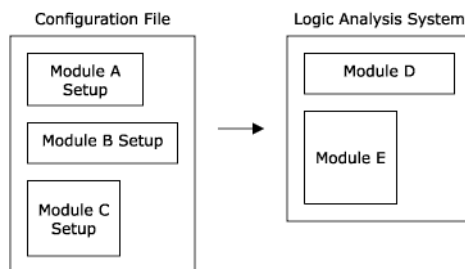


The File Information tab is accessed through the menu bar's **Setup>(Data Import Module)>File Info...** command.

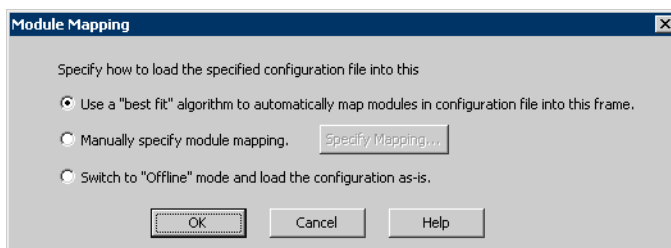
- See Also
- To view data import module file information (see [page 215](#))
 - Using Data Import Modules (see [page 211](#))

Module Mapping Dialog

The Module Mapping dialog helps you map module setup information from the configuration file you are opening to the modules in the logic analysis system you are using.



If you are opening an ALA format configuration file, modules must be compatible in order to setup a module with information from the configuration file. If you are opening an XML format configuration file, you can use setup information from any module; however, because of module differences, some settings may not transfer.



Menu	Description
Best-fit algorithm (see page 445)	Automatically maps configuration file modules into the current frame.
Manually map	The Specify Mapping... button opens the Specify Mapping dialog (see page 468) to specify how configuration file modules should be mapped into the current frame.
Offline mode	Loads the configuration file as-is in the <i>offline mode</i> .

See Also · To transfer module setups to/from multi-module systems (see [page 210](#))

Best-Fit Algorithm for Automatic Module Mapping

The best-fit algorithms differ slightly, depending on whether you're loading XML or ALA format configuration files.

When Loading XML
Format
Configuration Files:

For any given module in the configuration file:

- 1 Look for a module in the Overview window with the same name. If one is found, load the configuration into that module.
- 2 Look for the first module (based on the top to bottom order as shown in the Overview window) of the same *specific* type as the configuration file module. If one is found, load the configuration into that module.
- 3 Look for the first module of the same *general* type as the configuration file module.
- 4 If not found with any of the above, then:
 - a In online (either *Local* or *Remote*) mode, don't load the module.
 - b In offline mode, *create* the module.

When Loading ALA
Format
Configuration Files

For any given module in the configuration file:

- 1 Look for the first module (based on the top to bottom order as shown in the Overview window) of the same *specific* type as the configuration file module. If one is found, load the configuration into that module.
- 2 If not found, skip it.

NOTE

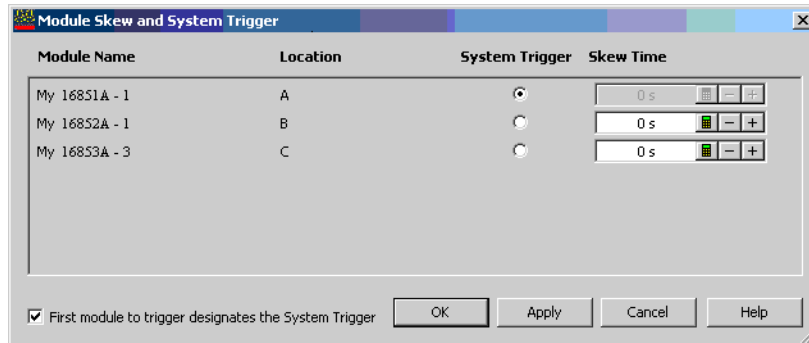
In offline mode:

When loading an ALA format configuration file, there is *always* a "clear" performed in the Overview window. Therefore, no matching algorithm is needed.

When loading an XML format configuration file, the XML best-fit algorithm is used.

See Also · Module Mapping Dialog (see [page 444](#))

Module Skew and System Trigger Dialog



This dialog is available when there are multiple *modules* (see [page 616](#)) in a logic analyzer or logic analysis system. It lets you:

- Specify which module is the *system trigger* (that is, which module's trigger reference point is *Time=0*).
- Specify the trigger reference point skew for modules that are not the *system trigger*.

When **First module to trigger designates the System Trigger** is checked, the first module to trigger after the next run is selected as the *system trigger*. Unchecking this option and checking it again causes the module that has currently triggered first to become the selected *system trigger*.

Disabled modules are grayed out.

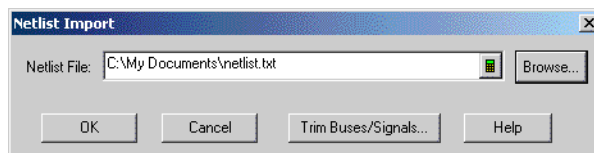
- See Also
- Setting the System Trigger and Skew Between Modules (see [page 312](#))
 - To disable and enable modules (see [page 72](#))

Netlist Import Dialog

The Netlist Import dialog lets you set up bus/signal names and assign them to logic analyzer channels by importing netlist files. Netlist files come from the Electronic Design Automation (EDA) tools used to design the device under test, and they contain information about the signals on the connectors built into the device under test for the logic analyzer probes.

NOTE

Before you can import bus/signal names from netlist files and assign them to logic analyzer channels, you must use the Define Probes dialog to identify the probes that are used with the logic analyzer.

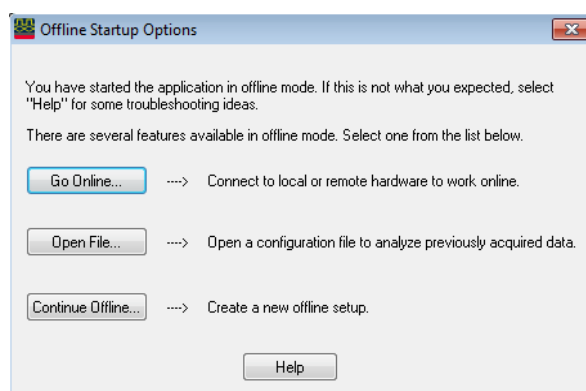


Menu	Description
Netlist File	Lets you enter or browse for the name of the netlist file to import.
Trim Buses/Signals...	Opens the Trim Bus/Signal Names dialog which lets you trim the bus/signal names imported from the netlist file.

See Also · To define buses and signals by importing netlist files (see [page 90](#))

Offline Startup Options Dialog

The Offline Startup Options dialog appears when you start the *Keysight Logic Analyzer* application and it wants to start in the *offline mode* (see [page 217](#)) (if this is unexpected, see *If starting in offline mode is unexpected* (see [page 332](#))). This dialog presents options for the tasks you can perform in the offline mode.



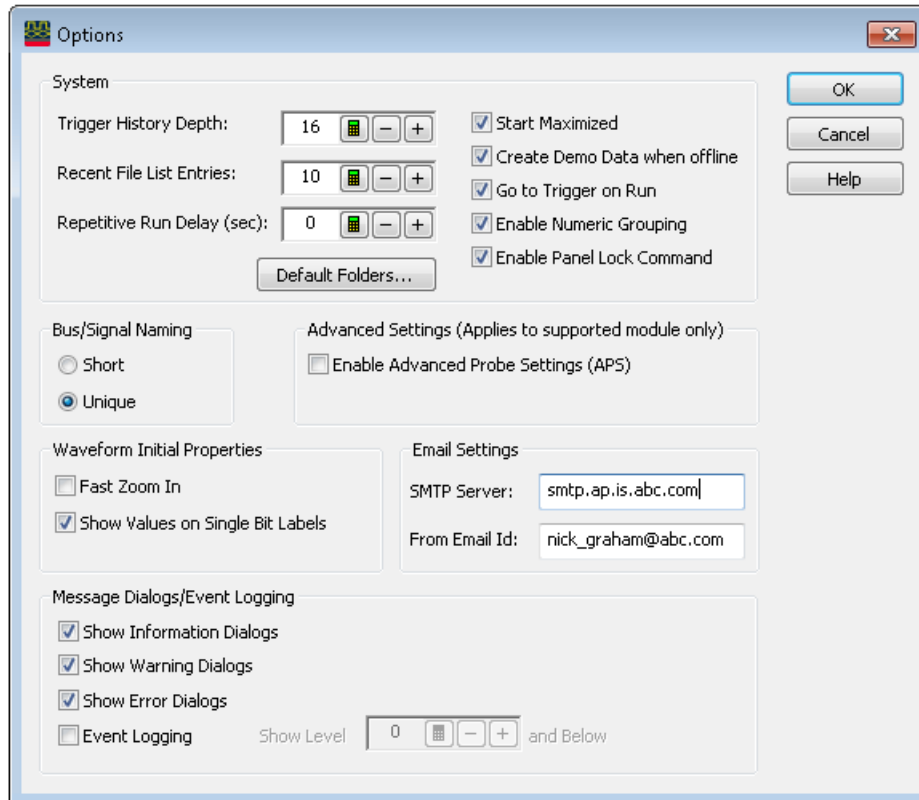
Menu	Description
Go Online...	Opens the Select System to Use dialog (see page 459) for choosing a frame to connect the <i>Keysight Logic Analyzer</i> application to.
Open File...	Lets you open a logic analyzer configuration file (see page 208).
Continue Offline...	Opens the Create a New Configuration dialog (see page 431) for creating a new logic analyzer configuration file.

See Also · If starting in offline mode is unexpected (see [page 332](#))

Options Dialog

To change your system options, select **Edit>Options...** from the menu bar. System options are written in the Windows registry file and persist across sessions.

- System options (see [page 448](#))
- Bus/Signal Naming options (see [page 449](#))
- Advanced Settings options (see [page 449](#))
- Email Settings options (see [page 449](#))
- Message Dialogs/Event Logging options (see [page 450](#))



System options

Menu	Description
Trigger History Depth	You can keep as many as 50 of the most recently used triggers. See To store a trigger (see page 194) for more information on re-using triggers. Trigger history is saved in the configuration files. The default is 10.
Recent File List Entries	This sets how many recently-loaded configuration files are shown in the File menu. The default is 4.
Repetitive Run Delay	Delay between repetitive measurements allows you to look at the captured data and decide whether to stop the measurement before the next run occurs.
Start Maximized	Specifies whether the <i>Keysight Logic Analyzer</i> application's main window is maximized when the application is started.
Create Demo Data when offline	When you run the analyzer, fake data will be created. This mode is useful when learning how to use the logic analyzer software. NOTE: The logic analyzer does not trigger with fake data. You can set the triggers, but will not get the same results you would with a real acquisition.
Go to Trigger on Run	Specifies, when a logic analyzer measurement is run, whether display windows are automatically positioned around the data that triggered the analyzer.
Enable Numeric Grouping	Numeric grouping adds spaces between every four hexadecimal and binary digits, spaces between every three octal digits, and commas between every three decimal digits (for example, FFFF FFFF, 1111 1111, 777 777, and 999,999).
Default Folders...	Opens the Default Folders dialog for specifying the default folder locations for configuration files and export files.

Bus/Signal Naming options

When you have defined the same bus/signal name in more than one module:

Menu	Description
Short	Bus/signal names are shown without the module name even if they are the same. In other words, it is possible to display two buses called "ADDR" that are not the same physical bus. (You can still see the module name in a tool tip by hovering over the bus/signal name.)
Unique	Module names are pre-pended to identical bus/signal names, for example, "Module 1:ADDR" and "Module 2:ADDR".

Advanced Settings options

Menu	Description
Enable Advanced Probe Settings (ASP)	<p>The Advanced Probe Settings are supported on U4154A/B logic analyzers and PCIe Gen3 analyzer.</p> <p>Checking this option enables:</p> <ul style="list-style-type: none"> the Probe Setup tab in the PCIe Gen3 analyzer's Setup dialog. The probe setup tab is used to adjust the analyzer's equalizing snoop probe (ESP) settings. See "Adjusting the Equalizing Snoop Probe (ESP) Settings" (in the online help). the APS button in the Buses/Signals tab of the U4154A/B logic analyzer's Setup dialog. The APS button is used to change the settings for the probes used with the U4154A/B logic analyzers. Probe settings include enabling peaking for the channels of these logic analyzers to compensate for the additional high frequency attenuation that some probing solutions provide on target signals. See "Changing Advanced Probe Settings for Logic Analyzers" on page 563.

Email Settings options

The Keysight Logic and Protocol Analyzer application supports the "send e-mail" feature in various dialogs of the application. For this feature to work, you need to specify the e-mail settings listed in the table below. The Keysight Logic and Protocol Analyzer application uses these settings to send e-mails for situations for which you have enabled the "send e-mail" feature. Some such situations are:

- when a trigger event occurs for which you have specified "send e-mail" as a trigger action. This is useful when you are waiting for a trigger event that may take a long time to occur and you want to do other things in the meantime.
- when markers in the logic analyzer meet certain measurement requirements (specified interval value) and you have configured "send e-mail" as an action.

Menu	Description
SMTP Server	<p>Enter the name of your mail server that is running the Simple Mail Transfer Protocol (SMTP). The Keysight Logic and Protocol Analyzer application will communicate directly with this e-mail server to send the e-mail.</p> <p>If you do not know your mail server name, ask your Network Administrator for it.</p>
From Email ID	<p>Enter the e-mail address that the Keysight Logic and Protocol Analyzer application will use to send an e-mail message. It is a good idea to use your e-mail address in this field so you will receive an e-mail informing you if the e-mail was unable to be delivered. You can specify multiple e-mail addresses by separating each address with either a space or a semicolon.</p>

NOTE

You can specify the rest of the e-mail settings such as the e-mail address(es) to which e-mail will be sent, subject, and text of the e-mail in the E-mail dialog. This dialog can be accessed from within various dialogs in which the "send email" feature is available such as the Advanced Trigger dialog. The E-mail dialog help topic lists the dialogs in which the "send e-mail" feature is available.

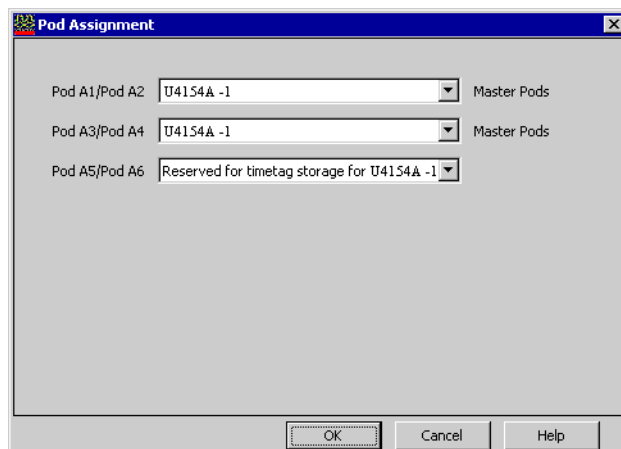
Message Dialogs/Event Logging Options

As with any other program, the *Keysight Logic Analyzer* application generates messages about events. You can choose which messages are displayed. Check the appropriate box to indicate you wish the dialogs displayed.

Menu	Description
Show Information Dialogs	Information dialogs offer tips such as the location of Simple Trigger, and do not indicate a failure.
Show Warning Dialogs	Warning dialogs occur when some setting may affect your data, such as being offline.
Show Error Dialogs	Error dialogs occur when an operation cannot be completed as specified.
Event Logging	You can choose to have all events recorded in a log file. Event logging will slow down your logic analyzer. Event logs can be viewed by navigating to Documents > My Documents > Keysight Technologies > Logic Analyzer > Logic Files in your local system. Set the event logging level according to the directions of your Keysight Technologies support person. Be sure to turn off event logging when resuming normal use.

Pod Assignment Dialog

The Pod Assignment dialog lets you reserve a *pod* (see [page 616](#)) or *pod pair* (see [page 616](#)) for time tag storage. If you choose the highest acquisition memory depth (see [page 129](#)) and there is a pod pair that has no buses or signals assigned to it, that pod pair is automatically reserved for time tag storage, and there is no need to use this dialog.

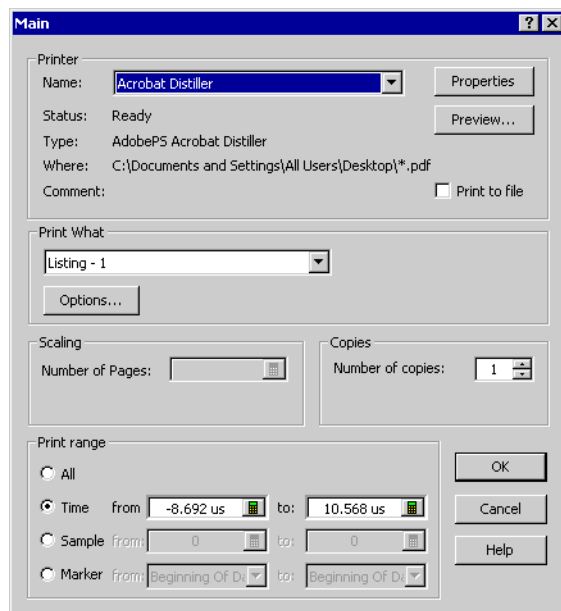


Menu	Description
Pod/Pod Pair	Indicates the pod or pod pair to be assigned or reserved.
(Module Selection)	For each pod or pod pair, selects the module to which it is assigned or reserves it for time tag storage.
Master Pods	Indicates that the pod or pod pair is on the master card.

- See Also
- Memory Depth and Channel Count Trade-offs (see [page 363](#))
 - Pod and Channel Naming Conventions (see [page 361](#))

Printing Data Dialog

After choosing **File>Print...**, this dialog lets you print the current measurement data from a display window.



NOTE

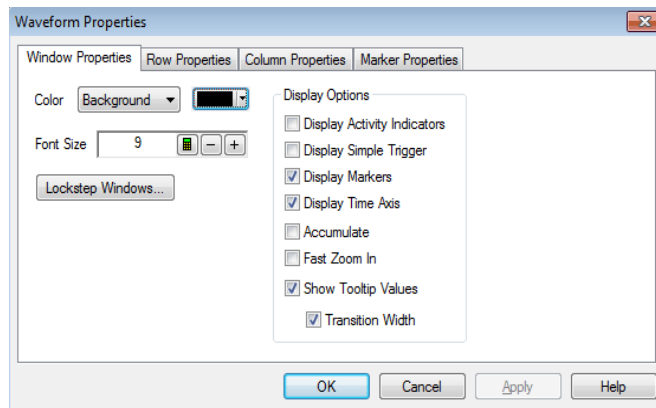
The first time you access the print dialog, you are asked to install a printer. Follow the directions in the printer install dialogs that appear.

Menu	Description
Printer	Lets you select the printer, change its properties, and preview the print out.
Print What	Lets you select which display window to print from and specify printing options.

Menu	Description
Scaling	Lets you specify the number of pages to print per sheet.
Copies	Lets you specify the number of copies to print.
Print range	You can print All data, or print just a defined range between times, sample numbers, or marker locations. Data is printed from the smallest time/sample to the largest.

- See Also
- Printing Captured Data (see [page 314](#))
 - To install a printer (see [page 315](#))
 - To connect a LAN (see [page 315](#))

Properties Dialog

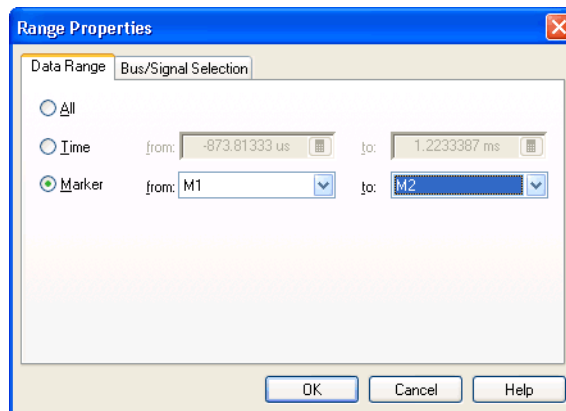


The Properties dialog is accessed through the menu bar's **Window>Properties....** Use it to set up how the window and the displayed data appear.

- Changing Waveform Window Properties (see [page 227](#))
- Changing Listing Window Properties (see [page 238](#))
- To set Compare window properties (see [page 276](#))
- Changing Source Window Properties (see [page 281](#))
- Changing Marker Properties (see [page 255](#))

Range Properties Dialog

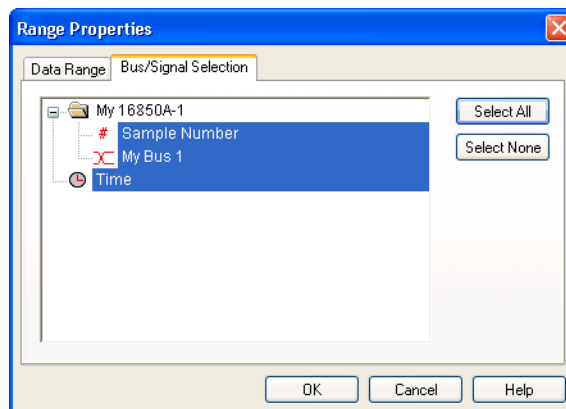
Data Range Tab Specifies the range of data to export.



Menu	Description
Time	Lets you specify the range of data to export by time.
Marker	Lets you specify the range of data to export by markers.

Bus/Signal Selection Tab

Specifies the buses/signals to export data from.



Menu	Description
Select All	Selects all buses/signals.
Select None	De-selects all buses/signals.

See Also • To export data to CSV format files (see [page 199](#))

Recall Trigger Dialog

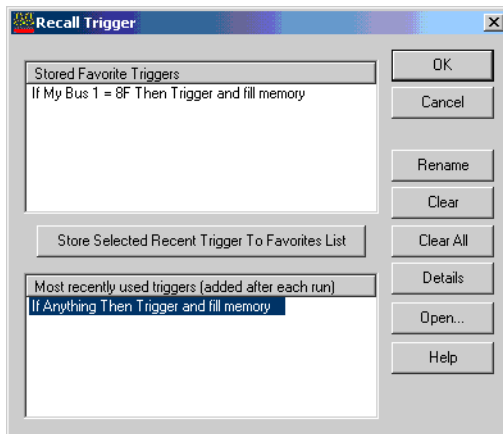
The Recall Trigger dialog lets you:

- Recall a previously-used trigger from:
 - The favorites list.
 - The recently used list.

- An XML format trigger specification file.
- Move a recently used trigger to the favorites list.
- Rename the trigger.
- Clear triggers from the favorites or recently-used list.
- View trigger details.

NOTE

The favorites list is saved with the logic analyzer configuration. If you load a new configuration file, the favorites list is overwritten.

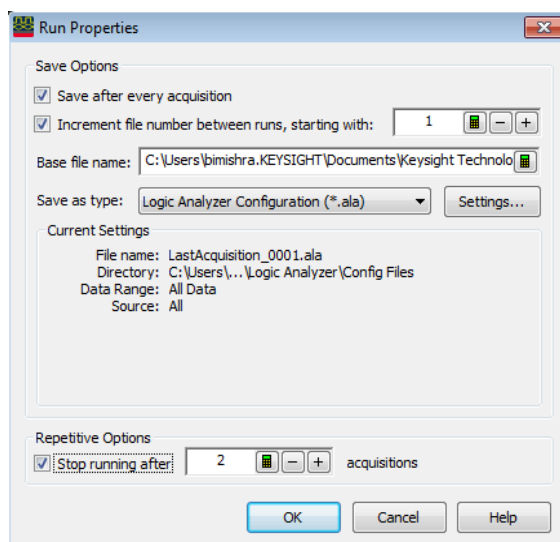


Menu	Description
Rename	Lets you edit the name of the highlighted trigger.
Clear	Clears the highlighted trigger from the list.
Clear All	Clears all triggers from recall lists.
Details	Shows complete definition of the highlighted trigger.
Open...	Lets you recall a trigger that was previously saved to an XML format trigger specification file. NOTE: When a trigger is stored to an XML format trigger specification file, trigger sequence steps are converted to advanced If/Then trigger functions. Therefore, the trigger may look different when you recall it from an XML format file; however, it is equivalent to the trigger that was saved.

- See Also
- To store a trigger (see [page 194](#))
 - To recall a trigger (see [page 194](#))
 - To set the trigger history depth (see [page 195](#))

Run Properties Dialog

Displays the options for saving captured data after each run and stopping after a certain number of repetitive runs.



Save Options

Menu	Description
Save after every acquisition	Enables or disables saving captured data after every run. When enabled, you can specify additional options for naming the data files.
Increment file numbers between runs, starting with	Enables or disables saving to consecutively numbered files. When enabled, you can specify the base file name and type.
Base file name	Lets you enter the base file name when saving to consecutively numbered files.
Save as type	Lets you specify the file type to use when saving to consecutively numbered files. You can choose from: <ul style="list-style-type: none"> Logic Analyzer Configuration (*.ala) Transferable Configuration (*.xml) Module CSV text file (*.csv) Module binary file (*.alb) Standard CSV text file (*.csv) For more information on these file types, see the topics under Saving Captured Data (and Logic Analyzer Setups) (see page 198).
Settings...	Opens the Save As dialog (see page 198) for specifying the settings of individual data files; for example, you can select the data source and sample range.
Current Settings	Shows the currently selected settings.

Repetitive Options

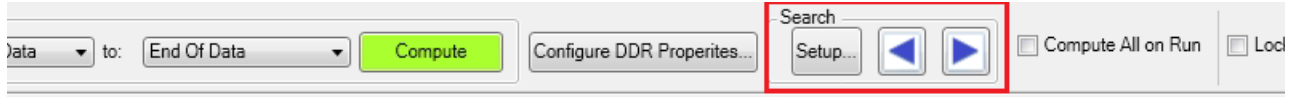
Menu	Description
Stop running after	Lets you stop repetitive runs after a certain number of acquisitions. When enabled, you can enter the number of acquisitions.

See Also • [Running/Stopping Measurements \(see page 196\)](#)

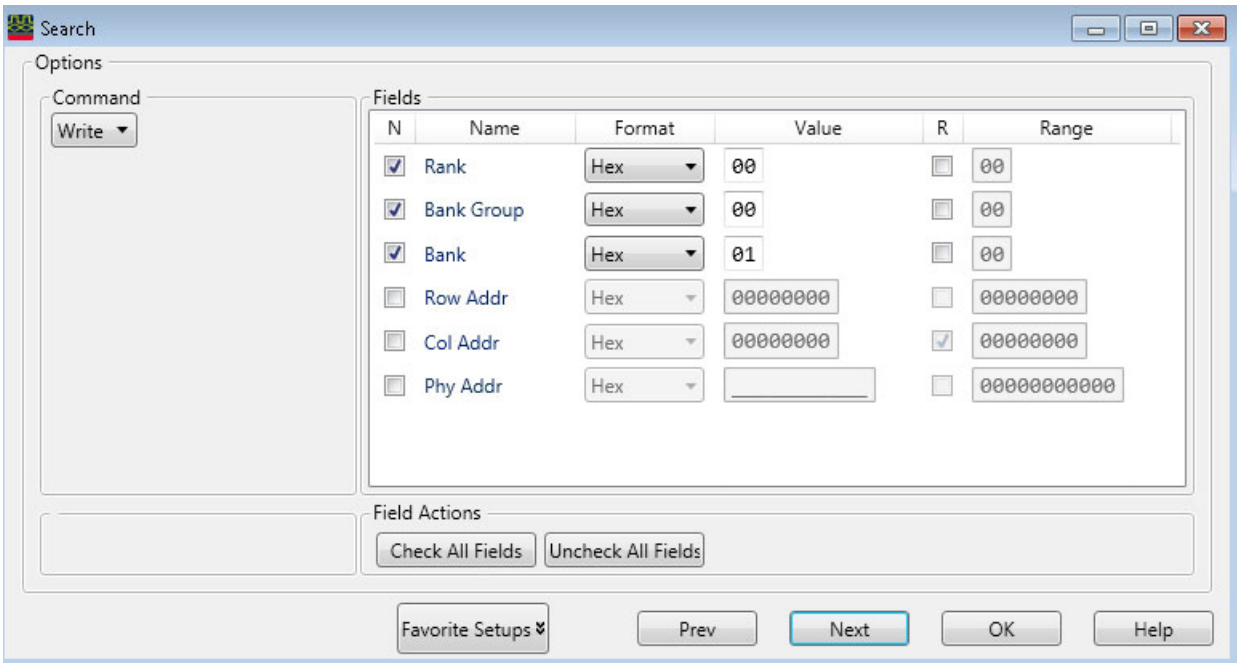
Search Dialog

The Search dialog box lets you specify the search criteria for quickly searching a particular command of interest from the list of decoded transactions displayed in the upper pane of the Memory Analysis or ONFi Analysis window.

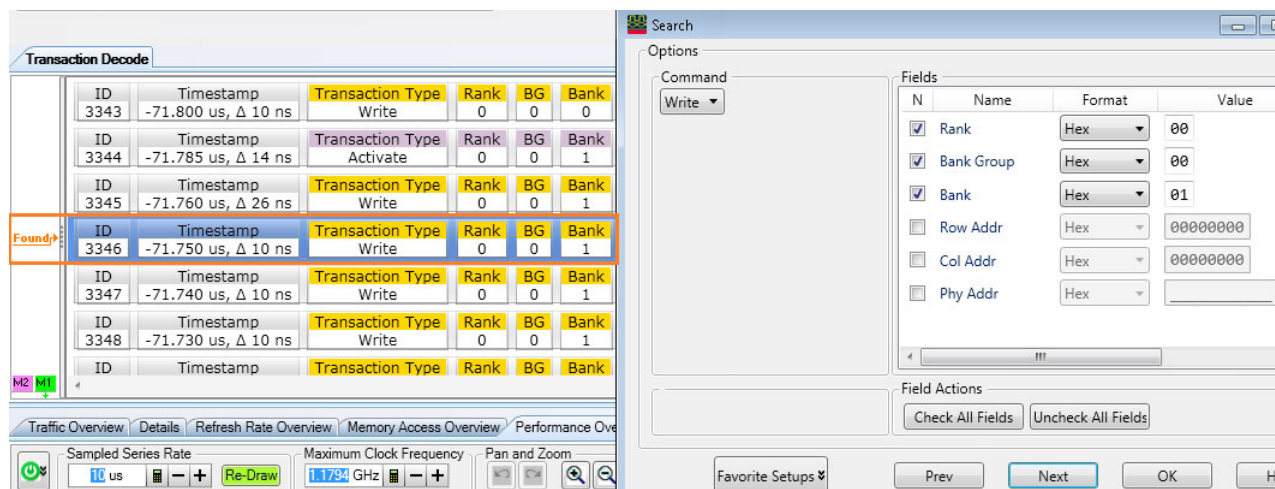
You access this dialog box by clicking the **Setup** button from the **Search** section of the **Memory Analysis** or **ONFi Analysis** windows.



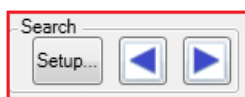
- 1 Click **Setup** in the **Search** section to define your search criteria. The **Search** dialog box is displayed.



- 2 In the left pane of Search, the Command listbox contains a list of commands supported by the Memory Analysis / ONFi Analysis window. Select the required command to be searched. On selecting a command, the right pane displays the fields relevant for that command. These fields help you refine and narrow down your search criteria for the selected command.
- 3 Select the checkbox displayed with each field that you want to include in the search criteria. Then specify the value of these fields. You can also select the R checkbox with the fields to define the range of field.
- 4 Click **Next** to begin the search based on the specified criteria. If a command matching the search criteria is found in the list of decoded transactions, then the first occurrence of that command is highlighted in the upper pane of the Memory Analysis / ONFi Analysis window. A "Found" marker is also placed at this occurrence. If a command matching the search criteria is not found in the list of decoded transactions, then a Not Found error message is displayed.

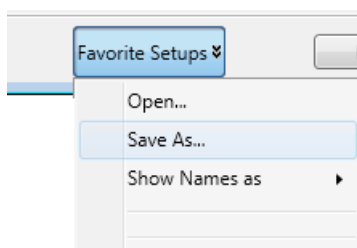


- 5 Traverse to the next occurrences of the command matching the search criteria using the Next button in the Search dialog box. Click **Prev** to move to the previous occurrence of the command in the decoded transactions.
- 6 Click **OK** to close the Search dialog box with the search criteria defined in it and then you can search based on this criteria using the Next and Previous buttons displayed in the Search section.



NOTE

You can save the search criteria that you defined in the Search dialog box for later use. You use the Save As option under Favorite Setups to save the search criteria in a Search Setup file.



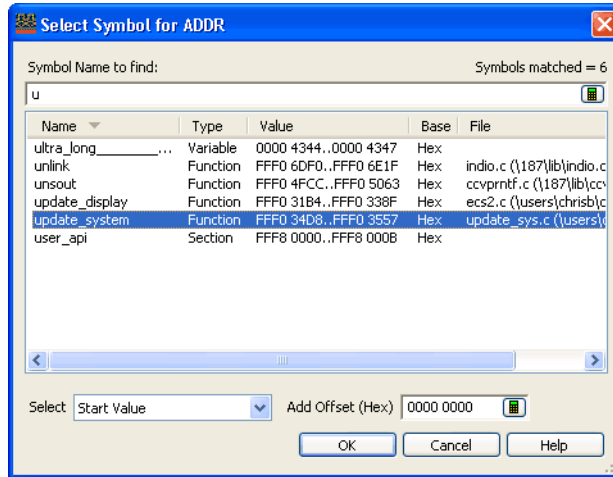
You can access a previously saved search setup using the Open option under Favorite Setups. Also, the recently used search setup files are displayed as a list under Favorite Setups. You may choose to display these recently used setup files as only filenames or as complete path and filenames in the list using the Show Names as option under Favorite Setups.

See Also [Chapter 9, “Analyzing ONFi Data using the ONFi Analysis Window”](#)

Analyzing Memory Data using the Memory Analysis Window

Select Symbol Dialog

Use the Select Symbol dialog to choose a symbol to use when the numeric base (see [page 241](#)) is set to Symbols.



The Select Symbol dialog becomes available when you use the Symbols number base in the following dialogs:

- Find (see [page 265](#))
- "Filter/Colorize" (in the online help)
- Specifying Simple Triggers (see [page 146](#))
- Advanced Trigger Dialog (see [page 424](#))

In the Select Symbol dialog:

Menu	Description
Symbol Name to find	Filter the list of symbols by typing characters in this field. You can also use the wildcard characters: <ul style="list-style-type: none"> · * (asterisk) to represent zero or more characters. · ? (question mark) to represent a single character. You can sort on any column in the symbol list by clicking the column header.
Name	Lists the symbol names defined or loaded for the bus (see Setting Up Symbols (see page 132)).
Type	Lists the symbol types (for example, section, variable, function, etc.). When sorting on this column, the symbols associated with each type are sorted by name.
Value	Lists the symbol values.
Base	Shows the number base for the symbol value.

Menu	Description
File	Shows the high-level source file that the symbol is from. When sorting on this column, the symbols associated with each file are sorted by value.
Select	When there is a range of values associated with the selected symbol, you can choose the Start Value of the range, the End Value of the range, and if you are selecting a symbol for a range setting, the Start and End Values of the range.
Add Offset (Hex)	<p>Lets you add an offset to the selected symbol value, for example, in the case where code is relocated.</p> <p>Note that when you use offsets, the system stores the "real" value, not the fact that it is a symbol plus an offset. This can cause the display of the symbol+offset to be different than what you entered. For example, for:</p> <pre> Symbol Value ----- RangeSymbol1 00 . . 10 RangeSymbol2 11 . . 20 Symbol3 21 Symbol4 22 Symbol5 FF </pre> <ul style="list-style-type: none"> · If you enter "Symbol3 + 1", the system interprets this as 22, which is also Symbol4. So, even though you entered "Symbol3 + 1", the system displays "Symbol4". · If you enter "RangeSymbol2(end value) + 1", it will be displayed as "Symbol3". · If you enter "RangeSymbol1(start value) + 12", it will be displayed as "RangeSymbol2+1". · Symbol values can "wrap", such as "Symbol5 + 1" where the result is 00. In this case, the you enter "Symbol5 + 1" and the resulting symbol is "RangeSymbol1".

- See Also
- Setting Up Symbols (see [page 132](#))
 - Symbols Dialog (see [page 471](#))

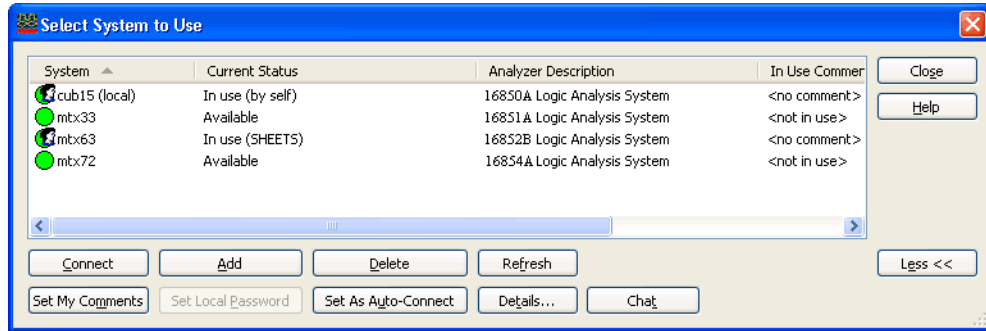
Select System to Use Dialog

The Select System to Use dialog lets you select a logic analysis system *frame* (see [page 615](#)) to connect to, and it lets you manage the list of frames.

A local system is a frame connected to the machine that runs the *Keysight Logic Analyzer* application. Remote systems are frames connected to machines elsewhere on your network. The remote system list can be managed any way you wish. The list of remote systems is stored on a per-user basis (each user has their own customizable list of remote systems).

The information in the dialog can be sorted by any of the columns by clicking on the column header.

You can right-click on any row in the dialog to get quick access to a menu of system-specific actions.



Menu	Description
System	Displays the hostname or IP address of a frame in the list. A green, yellow, or red indicator indicates whether you can connect to or obtain information from this frame (green), the frame hardware is initializing (yellow), or some other problem (red)—such as the host not having any frames, the host is offline, or the software on the target machine is incompatible.
Current Status	Displays whether a frame is available, offline (that is, powered down), currently in use, or has an incompatible remote service (that is, its software needs to be upgraded to match the version of software installed on the machine displaying this dialog).
Analyzer Description	Displays the type of logic analyzer frame.
In Use Comment	Displays the "system in use" comments set by the user currently using the frame.
Connect	Connects to the selected logic analysis system (either local or remote). You can connect to a frame even if it is in use. You will be warned if you elect to connect to a frame that is already in-use before bumping the other user offline. Taking a user offline by connecting to an in-use frame will result in the other user losing any unsaved acquisition data (included in the warning message). For this reason, it is always preferable to contact the user (perhaps using the chat feature) and ask them to gracefully take themselves offline—saving any important data. If the user is not reachable—this ability to claim the system remotely is a powerful feature.
Add	Lets you enter the hostname or IP address of a remote system to be added to the list.
Delete	Removes the selected system from the system list.
Refresh	Forces an immediate refresh of all information in this dialog. This dialog is auto-updating—so you should not need to push this button. It is provided as a fail-safe only.
More >> Less <<	Shows or hides the bottom row of buttons.
Set My Comments	Opens a dialog that lets you enter some in-use comments. If another <i>Keysight Logic Analyzer</i> application attempts to connect to the system that you are currently connected to, your "system in-use" comments will be displayed. These comments can be used to explain why you are using a particular system, give your contact information, etc. These comments can be changed before going online with a frame or while online with a frame. The changes will take effect immediately.

Menu	Description
Set Local Password	Opens the Remote Access Password Utility dialog for establishing a remote-connect password (see page 68). The icon next to a <i>remote</i> system will have a padlock superimposed on it if a password is required to connect to that machine. A remote-access password forces remote users to enter this password before connecting to any local instrument. The local system icon will never have a padlock on it because a remote-access password is never required for the local client. This button will change to "Clear Local Password" if a remote-access password is already set. Only Windows users with administrative credentials will have the ability to set or clear remote-access passwords.
Set as Auto-Connect	Selects the logic analysis system frame as the one to use when the <i>Keysight Logic Analyzer</i> application starts. The words "auto-connect" will show up next to the frame selected as an auto-connect. This button will change to "Clear Auto-Connect" if this frame is already set as an auto-connect.
Details...	Opens the Frame/Module Information dialog (see page 439) which displays detailed information about the selected logic analysis system. If the selected frame is a member of a multiframe set, frame and module information on all frames in the set is presented in this dialog.
Chat	Opens the Chat Select Destination dialog (see page 430) for communicating with someone on the selected system. You have the option of opening a two-way, interactive chat with any person logged on to the selected system or the user that is currently using the selected frame (if different).
Close	Closes the dialog without connecting to a local or remote system.

- Notes:
- If the systems in this dialog are slow to update, try removing any machines in the list that are unresponsive (red) or not used.
 - Slow network conditions can also slow the update rate of this dialog.
 - This dialog is auto-updating—so it is not necessary to push the refresh button to see status changes. The refresh button is provided as a fail-safe only.

- See Also
- [Connecting to a Logic Analysis System \(see page 59\)](#)

Software Licensing Dialog

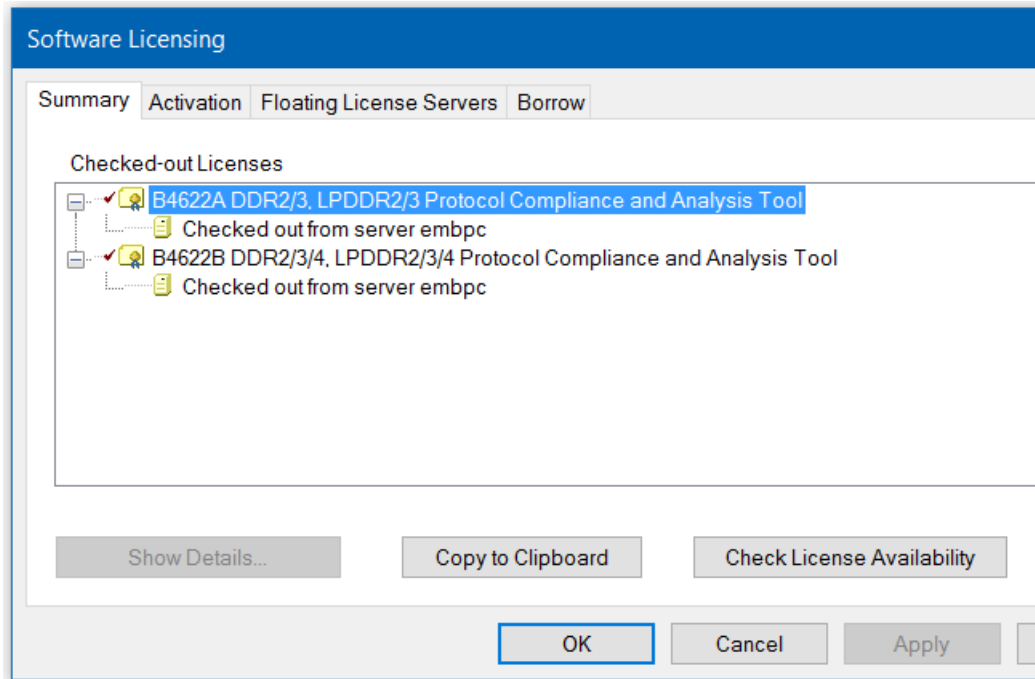
The Software Licensing dialog is used to manage the software licenses used by a logic analysis system. This dialog has four tabs:

- [Summary Tab \(see page 461\)](#)
- [Activation Tab \(see page 463\)](#)
- [Floating License Servers Tab \(see page 464\)](#)
- [Borrow Tab \(see page 465\)](#)

- See Also
- [Managing Software Licenses \(see page 321\)](#)

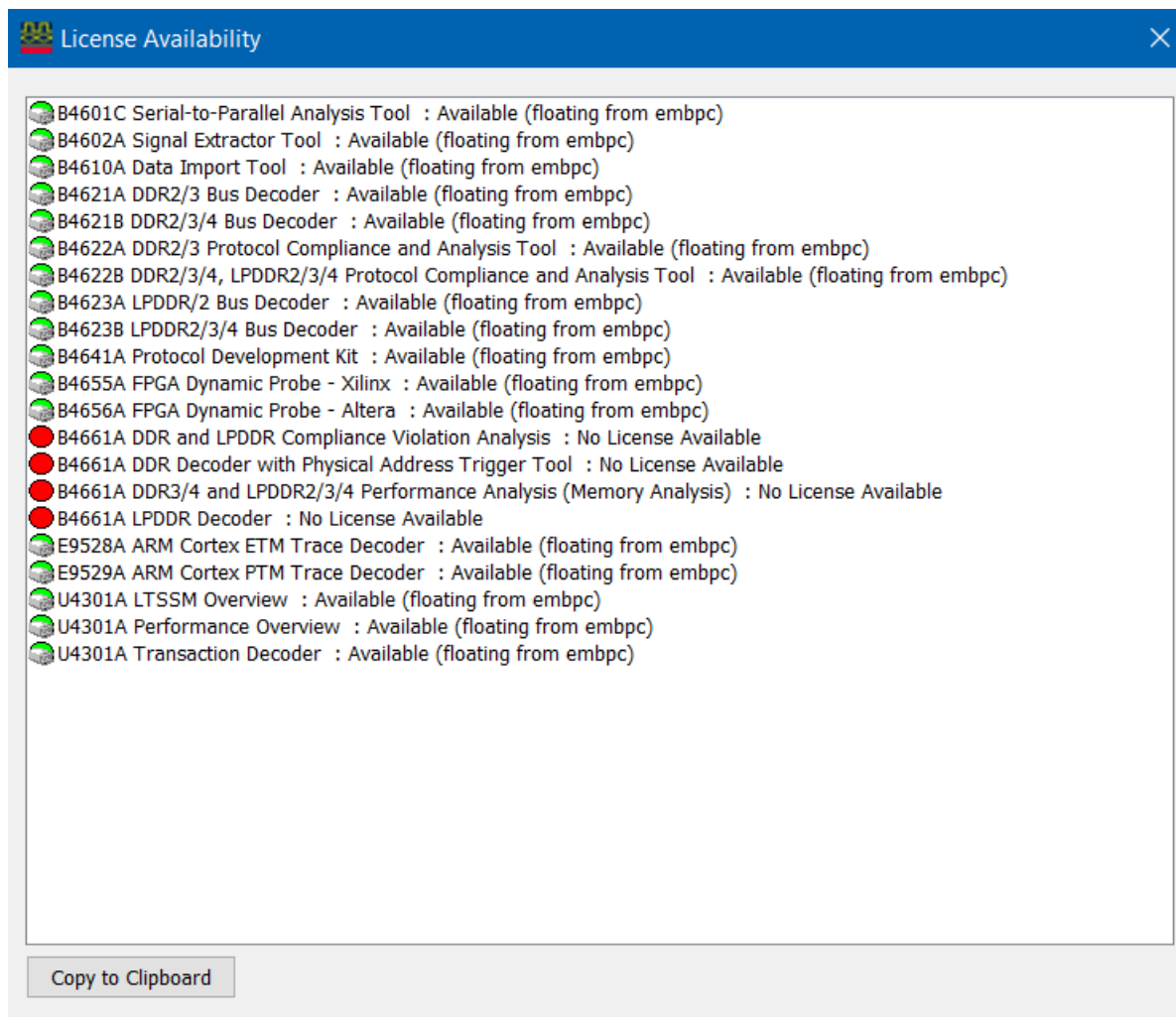
Summary Tab

The Software Licensing dialog's Summary tab lists the licenses that you have checked out from the license server for use in the logic analysis system. (You can add a license server using the Floating License Servers tab of this dialog box.) A license is checked out from a license server for use only if the license is currently available on this server.



Red check marks in the Checked-out Licenses list indicate floating licenses that are checked out and are currently in use. A red "X" next to a folder in this list indicates a floating license that has been checked out but the software associated to this license has not been installed.

Menu	Description
Show Details...	Opens a dialog that displays detailed information about the selected license.
Copy to Clipboard	Copies software licensing summary information to the clip board.
Check License Availability	Displays the current availability status of floating licenses on the license server. A sample screen is displayed below.

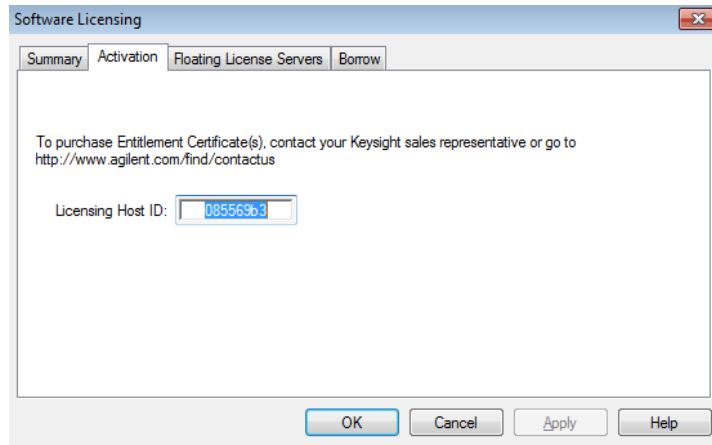


To know how to install and manage these licenses, refer to the **Keysight License Manager (KLM)** help. KLM is installed when you install the Keysight Logic and Protocol Analyzer software. It lets you easily manage right-to-use licenses for software and hardware capabilities on Keysight host systems.

See Also • To view active software license information (see [page 322](#))

Activation Tab

The Software Licensing dialog's Activation tab contains the Licensing Host ID which is needed to activate software licenses.



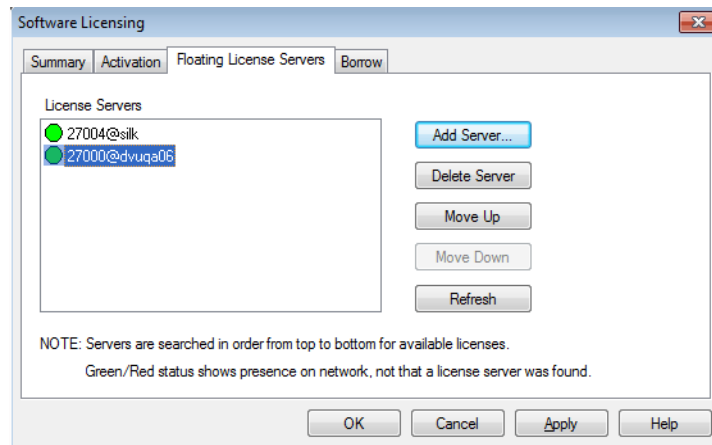
Menu	Description
Licensing Host ID	This ID is used when obtaining license files for software tools.

See Also • To activate software licenses (see [page 323](#))

Floating License Servers Tab

The Software Licensing dialog's Floating License Servers tab lets you add / delete and order the license server(s) from where floating licenses can be checked out for use.

NOTE: If you have currently checked out license(s) from a license server, then the Add Server, Delete Server buttons in this tab are disabled indicating that you cannot add or delete a server while its licenses are in use.



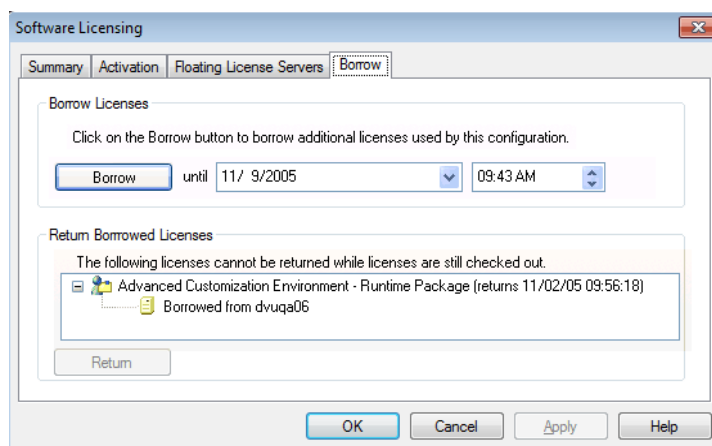
When licenses need to be checked out, the list of **License Servers** is searched in order, and the license is checked out from the first server having an available license.

Menu	Description
Add Server...	Opens a dialog for entering the computer name and port number of the floating license server. CAUTION Only enter names of computers (or logic analyzers) that are floating license servers. Otherwise, the license manager interface hangs up for many minutes trying to determine if the computer is really a floating license server.
Delete Server	Removes the selected server from the list.
Move Up	Raises the selected server in the search order.
Move Down	Lowers the selected server in the search order.
Refresh	Updates the green or red server availability indicators. This is only a check of whether the computer is on the network, not of whether the license server software is running on that computer.
Apply	License server changes must be applied before you can go to the Summary or Borrow tabs.

See Also · To access floating license servers (see [page 324](#))

Borrow Tab

The Software Licensing dialog's Borrow tab lets you borrow floating licenses from a server for a period of time, for example, if you're taking a logic analyzer (or a computer running the *Keysight Logic Analyzer* application) out of the office (or just off the network). When a borrowed license's time expires, the license is automatically returned to the server. However, you can also use the Borrow tab to return licenses early.



Menu	Description
Borrow Licenses	Lets you enter the amount of time you want to borrow a license for and borrow the license. The default time is seven days. The minimum time is ten minutes. See also Messages in the Borrow Licenses Area (see page 466).
Return Borrowed Licenses	Lets you return borrowed licenses early. All borrowed licenses must be returned at the same time. You are not able to return borrowed licenses while any licenses are checked out. See also Messages in the Return Borrowed Licenses Area (see page 466).

See Also · To borrow floating licenses and return them early (see [page 326](#))

Messages in the Borrow Licenses Area

These messages can appear in the Software Licensing dialog's Borrow tab.

Borrowing is not supported while in Demo Center Mode.

Licenses cannot be borrowed while using the Demo Center feature.

This configuration is not using any licenses. There is nothing to borrow.

No licenses are currently in use, so there is nothing to borrow.

This configuration is not using any floating licenses. There is nothing to borrow.

If all licenses being used are node-locked, there are no licenses that can be borrowed.

The licenses for this configuration cannot be borrowed because some are node-locked.

You cannot borrow licenses when floating licenses and node-locked licenses are in use at the same time.

Click on the Borrow button to borrow licenses used by this configuration.

When this message is displayed, you can borrow the licenses used by the current configuration.

Click on the Borrow button to borrow additional licenses used by this configuration.

When this message is displayed, some licenses are already borrowed, and you can borrow the additional licenses used by the current configuration.

All licenses needed by this configuration are already borrowed.

When this message is displayed, all licenses in use are already borrowed. To borrow additional licenses, open the feature that requires the license, and return to the Software Licensing dialog's Borrow tab.

Messages in the Return Borrowed Licenses Area

These messages can appear in the Software Licensing dialog's Borrow tab.

The following licenses cannot be returned while some servers are not configured.

If a license was checked out from a server that is not currently configured, licenses cannot be returned early.

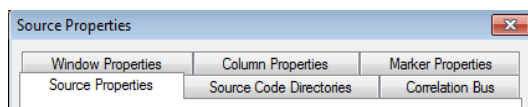
The following licenses cannot be returned while licenses are still checked out.

If any licenses are checked out (and not borrowed), borrowed licenses cannot be returned early.

The following licenses are currently borrowed for exclusive use on this system.

When this message is displayed, borrowed licenses can be returned early.

Source Viewer Properties Dialog



· Source Properties Tab (see [page 467](#))

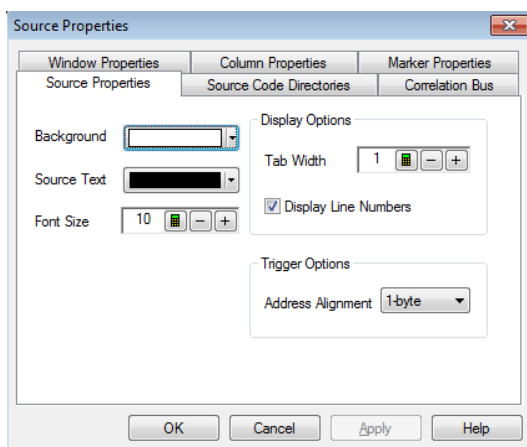
- Source Code Directories Tab (see [page 467](#))
- Correlation Bus Tab (see [page 468](#))

The **Listing Properties**, **Column Properties**, and **Marker Properties** tabs are the same as in the Listing window.

- See Also
- Changing Listing Window Properties (see [page 238](#))
 - Changing Bus/Signal Column Properties (see [page 240](#))

Source Properties Tab

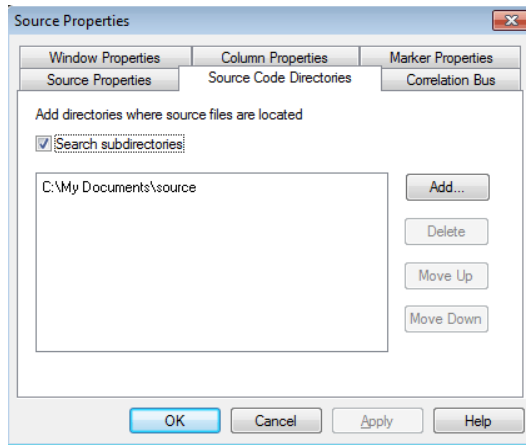
The Source Viewer Properties dialog's Source Properties tab lets you set the background color and font size as well as display options like the number of spaces to use for tabs and whether or not to display line numbers.



- See Also
- Changing Source Window Properties (see [page 281](#))
 - Source Viewer Properties Dialog (see [page 466](#))

Source Code Directories Tab

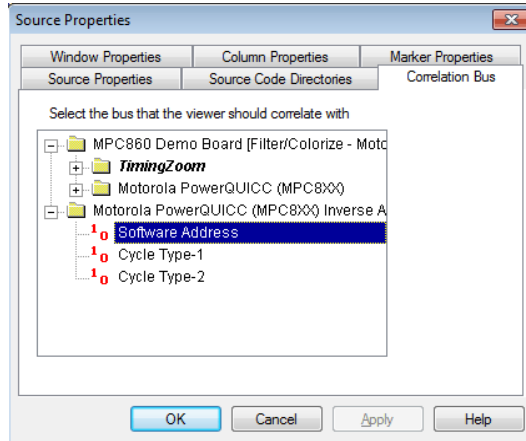
The Source Viewer Properties dialog's Source Code Directories tab lets you specify the directories where the source code is located. This is necessary because source file paths specified in the symbol file may not be valid if you compile on one computer and debug on another. You can specify multiple directories and change their order. Directories are searched in order, and you can specify whether subdirectories are searched.



- See Also
- To edit the source code directory list (see [page 279](#))
 - Source Viewer Properties Dialog (see [page 466](#))

Correlation Bus Tab

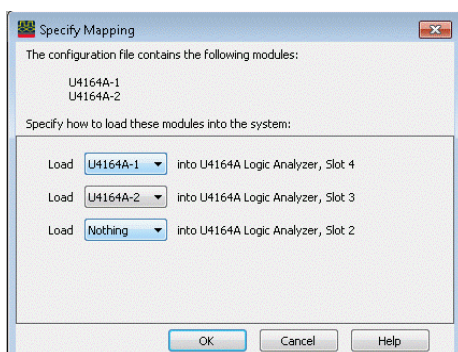
The Source Viewer Properties dialog's Correlation Bus tab lets you specify the bus whose line number symbols will be used for source correlation. Typically, you will select the "software address" bus generated by an inverse assembler tool or another address bus.



- See Also
- To select the correlation bus (see [page 280](#))
 - Source Viewer Properties Dialog (see [page 466](#))

Specify Mapping Dialog

The Specify Mapping dialog lets you manually map modules from a configuration file to modules in the logic analysis system.



The top of the Specify Mapping dialog lists the module configurations in the file being loaded. The bottom part of the dialog lists modules in the logic analysis system. The **Nothing** selection says not to use any of the module configurations being loaded.

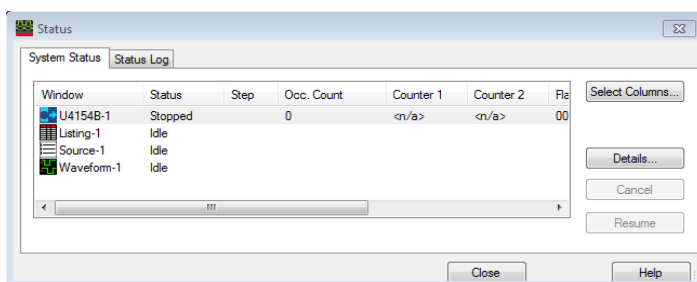
NOTE

If you cannot load setup information from an ALA format configuration file into a particular module, the modules are not compatible, and you need to use an XML format configuration file to transfer the module setup information (see [If an ALA format configuration file won't open](#) (see [page 333](#))).

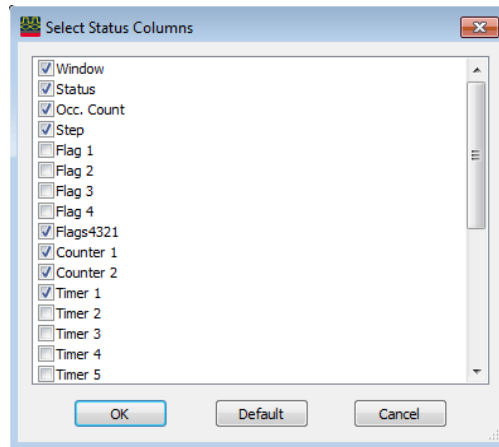
- See Also
- Module Mapping Dialog (see [page 444](#))
 - To transfer module setups to/from multi-module systems (see [page 210](#))
 - ALA vs. XML, When to Use Each Format (see [page 380](#))

Status Dialog

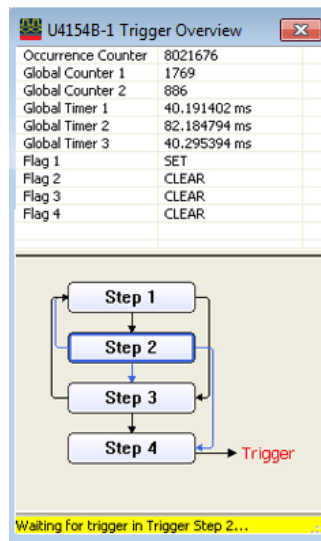
System Status Tab Displays the logic analysis system status.



Menu	Description
Select Columns...	For logic analyzer modules, opens the Select Status Columns dialog for selecting the columns to be displayed.



Default Columns	Resets the columns displayed in the System Status tab to the default set.
Details...	Displays details for the selected module, tool, or window.

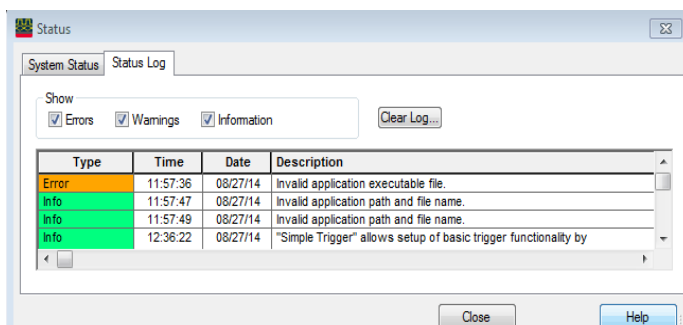


For example, module details include: graphical trigger overview, occurrence counter, global counter, flags, and timer status.

Cancel	Cancels processing for the selected module, tool, or window.
Resume	Resumes canceled processing for the selected module, tool, or window.

You can copy a module, tool, or window's status to the clip board by right-clicking and choosing **Copy** from the popup menu.

Status Log Tab Displays the logic analysis system status log.

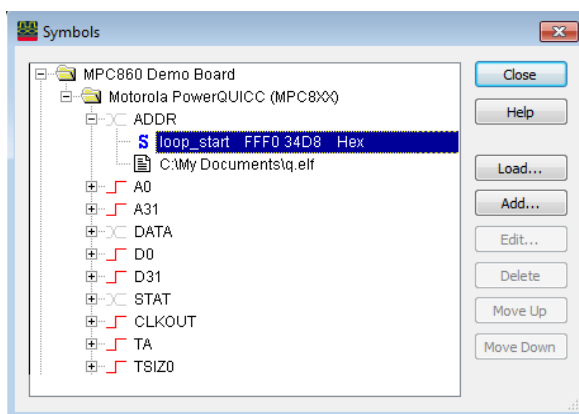


You can select the types of log messages to display, and you can clear the log.

See Also • Options Dialog (see [page 447](#))

Symbols Dialog

The Symbols dialog lets you to define, copy, and edit symbols for the entire system.



User-defined symbols are indicated by **S**.

Symbols files are indicated by . Symbol files can be either compiler-generated object files containing symbols or general-purpose ASCII (GPA) format symbol files.

Menu	Description
Load...	If a bus/signal is selected, this opens the Select Symbol File dialog for loading symbols from a compiler-generated object file or a general-purpose ASCII (GPA) format symbol file. If symbol file is selected, it is reloaded.
Add...	Adds a user-defined symbol to the selected bus/signal. To know more, see "To create and edit user-defined symbols" on page 132.

Menu	Description
Edit...	Edits the selected user-defined symbol. To know more, see "To create and edit user-defined symbols" on page 132.
Delete / Delete All	Removes the selected user-defined symbol or symbol file (or all symbols if a bus/signal is selected) from the list.
Move Up Move Down	Moves the selected user-defined symbol or symbol file up or down within the list. When the system looks for a symbol that corresponds to a bus/signal value, the first match is used. (More than one symbol can match a given value.)

- See Also
- Setting Up Symbols (see [page 132](#))
 - Displaying Names (Symbols) for Bus/Signal Values (see [page 243](#))

System Summary Dialog

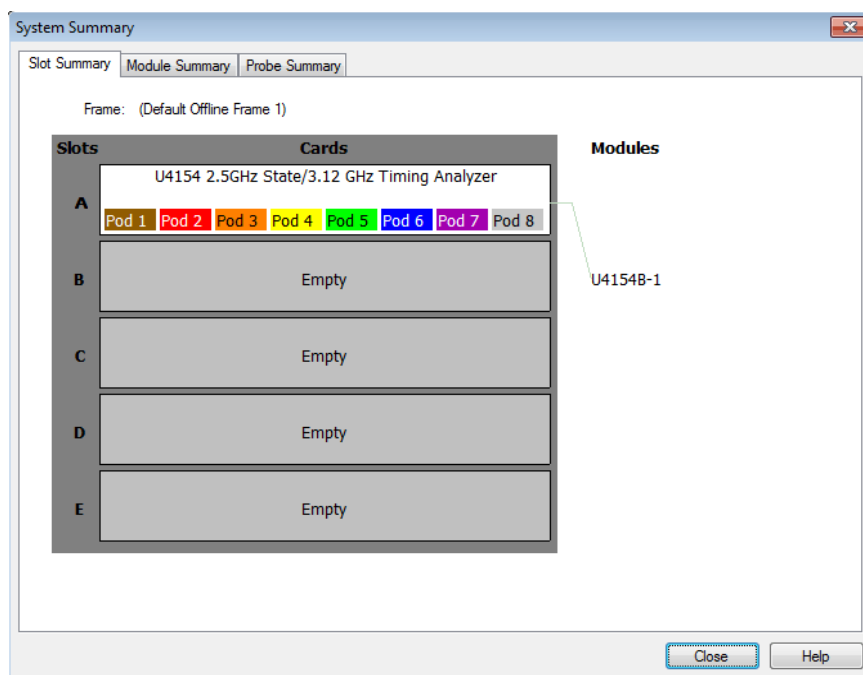
The System Summary dialog is used to see summary information about all modules in the system including a physical view of the cards in the system (Slot Summary (see [page 472](#))), a list of all of the modules and their trigger times (Module Summary (see [page 474](#))), and a list of how to connect all of the probes (Probe Summary (see [page 474](#))).

- Slot Summary Tab (see [page 472](#))
- Module Summary Tab (see [page 474](#))
- Probe Summary Tab (see [page 474](#))

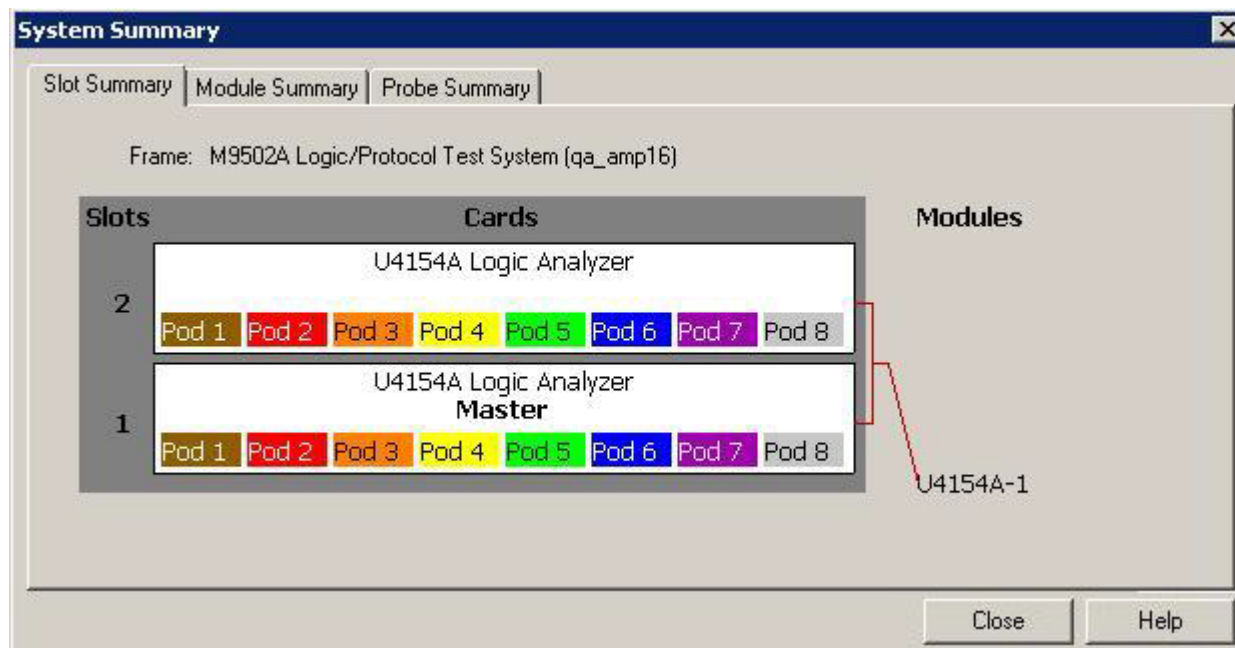
- See Also
- Analyzer Setup Dialog (see [page 426](#))

Slot Summary Tab

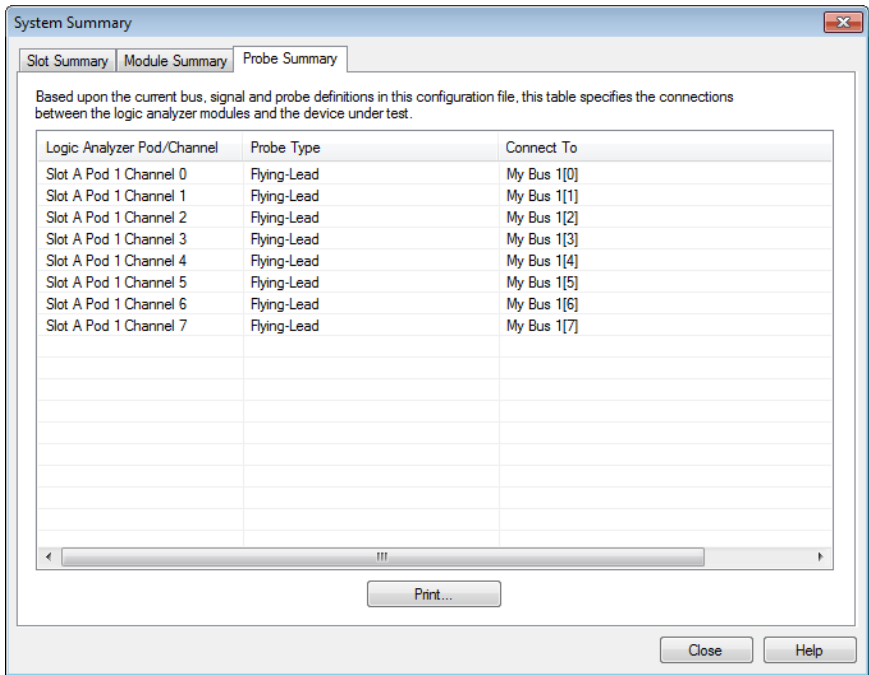
For the selected *frame* (see [page 615](#)) (if there are multiple frames), the Slot Summary tab shows: the *slots* (see [page 618](#)), *cards* (see [page 614](#)), and *modules* (see [page 616](#)) in the logic analysis system frame, or the *cards* (see [page 614](#)) and *modules* (see [page 616](#)) in the standalone logic analyzer.



U4154A/B Logic Analyzer Slot Summary



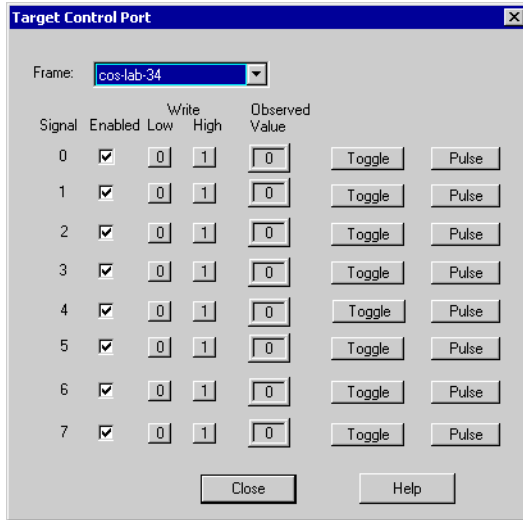
If you are using U4154A/B Logic Analyzer, the Slot Summary tab displays a graphical view of the slots of the AXIe chassis in which you installed the U4154A/B Logic Analyzer. If you are using a two-slot chassis, then the placement of the U4154A/B cards in these two slots is displayed. If you are using a 5-slot chassis, then all the five slots are displayed. You can also view which U4154A/B card in a multi-card set is considered as the master card.



- See Also
- "To define probes" (in the online help)
 - System Summary Dialog (see [page 472](#))
 - "Connecting Probes Using the Probe Summary" (in the online help)

Target Control Port Dialog

The 16850-series logic analysis system frames (see [page 615](#)) have a *target control port*, an 8-bit, 3.3V port that can be used to send signals to a device under test. The target control port does not function like a pattern generator, but more like a remote control for switches in the device under test. For the selected frame (if there are multiple frames), the Target Control Port dialog lets you set the output signal levels on the port.

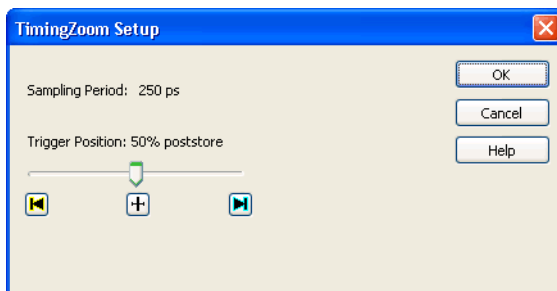


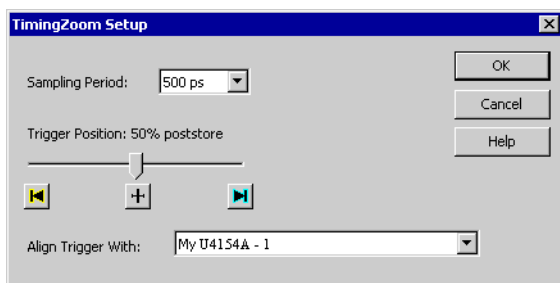
Menu	Description
Signal	Identifies the target control port signal.
Enabled	Lets you enable or disable an output signal on the target control port. When disabled, the signal has a tri-state high-impedance output. When enabled, the signal outputs a standard 3.3V logic level with 1 high and 0 low.
Write	Gives you two buttons, 0 and 1, for changing the signal output value to a standard 3.3V logic low or high, respectively.
Observed Value	Displays the signal value observed on the target control port signal.
Toggle	Flips the settings of the signal and leaves them that way. For example, if your signal is set to 1 and you click Toggle , the setting changes to 0.
Pulse	Flips the settings for one clock cycle, which is at least 16 ms. The pulse may last longer. You cannot specify the duration of the pulse.

See Also · To control signals in the device under test (see [page 58](#))

TimingZoom Setup Dialog

Timing zoom collects additional high-speed timing data around the trigger of the logic analyzer.





Menu	Description
Sampling Period	Displays the timing zoom sampling period. With some logic analyzers (see Logic Analyzer Notes, Timing Zoom (see page 536)), you can change the sampling period to see more or less sampling resolution around the trigger.
Trigger Position	Lets you position the timing zoom acquisition memory around the event that triggers the logic analyzer.

- See Also
- Using Timing Zoom (see [page 129](#))
 - Logic Analyzer Notes, Timing Zoom (see [page 536](#))

Trigger Functions

The trigger functions available in the Advanced Trigger dialog give you pre-configured trigger setups for common measurements. If the trigger function you need is not available, start with a trigger function that is close, convert the trigger sequence step to advanced If/Then trigger functions, and edit the If/Then trigger functions.

- Timing Mode Trigger Functions (see [page 478](#))
- State Mode Trigger Functions (see [page 490](#))

- See Also
- To show a trigger sequence step as Advanced If/Then trigger functions (see [page 182](#))
 - To convert a trigger sequence step to Advanced If/Then trigger functions (see [page 183](#))
 - Triggering From, and Sending Triggers To, Other Modules/Instruments (see [page 186](#))

Timing Mode Trigger Functions

The following trigger setup examples are available as Trigger Functions in the Advanced Trigger dialog when in the timing acquisition mode. To see these trigger setups in the context of an example measurement refer to Making a timing analyzer measurement (see [page 50](#)).

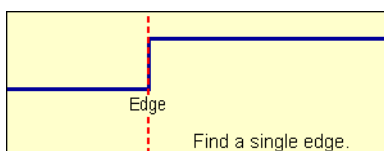
- Edge
 - Edge (see [page 479](#))
 - "N" number of edges (see [page 479](#))
 - Edge and Pattern (see [page 480](#))
 - Edge followed by edge (see [page 480](#))
 - Edges too far apart (see [page 481](#))
 - Edge followed by pattern (see [page 481](#))
 - Pattern too late after edge (see [page 482](#))
- Bus Pattern
 - Pattern (see [page 482](#))
 - Edge and Pattern (see [page 480](#))
 - Pattern present for > "T" time (see [page 483](#))
 - Pattern present for < "T" time (see [page 483](#))
 - Pattern absent for > "T" time (see [page 484](#))
 - Pattern absent for < "T" time (see [page 484](#))
 - Edge followed by pattern (see [page 481](#))
 - Pattern too late after edge (see [page 482](#))
- Other
 - Find anything "N" times (see [page 484](#))
 - Width violation on pattern or pulse (see [page 485](#))
 - Wait "T" seconds (see [page 485](#))
 - Run until user stop (see [page 486](#))
 - Wait for external arm (see [page 486](#))
 - Wait for arm from another module (see [page 487](#))
- Advanced
 - Advanced If/Then (see [page 487](#))
 - Advanced 2-Way Branch (see [page 488](#))
 - Advanced 3-Way Branch (see [page 488](#))
 - Advanced 4-Way Branch (see [page 489](#))
 - Pattern "AND" Pattern (see [page 489](#))
 - Pattern "OR" Pattern (see [page 490](#))

- See Also
- To replace or insert trigger functions into trigger sequence steps (see [page 158](#))
 - State Mode Trigger Functions (see [page 490](#))
 - To store a trigger (see [page 194](#))
 - To recall a trigger (see [page 194](#))

Edge

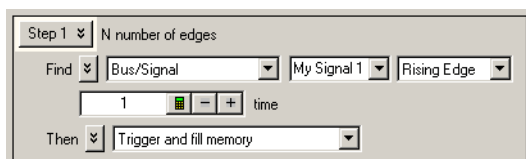


This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a user-defined edge occurs.

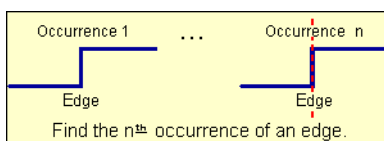


- To edit this function
- [Specifying Advanced Triggers \(see page 152\)](#)

"N" number of edges

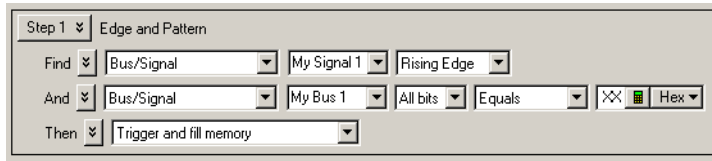


This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when the "Nth" occurrence of a user-defined edge occurs.

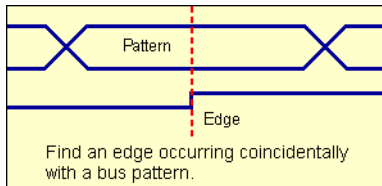


- To edit this function
- [Specifying Advanced Triggers \(see page 152\)](#)

Edge and Pattern



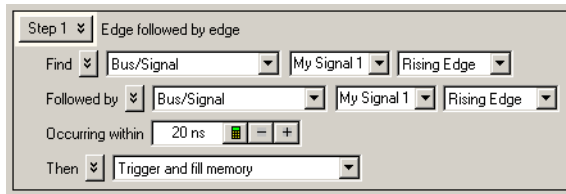
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when both a user-defined edge and bus pattern occur at the same time.



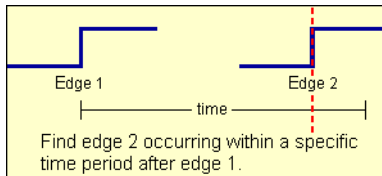
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Edge followed by edge



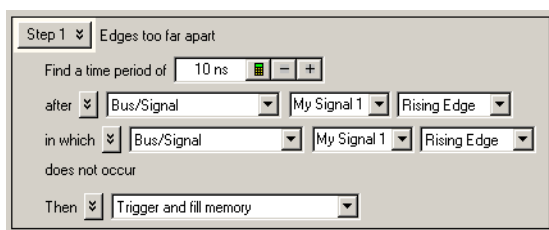
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when edge 2 occurs within a specified time period after edge 1.



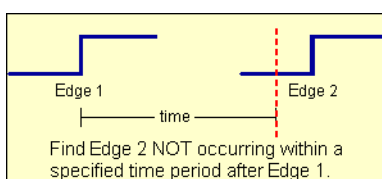
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Edges too far apart



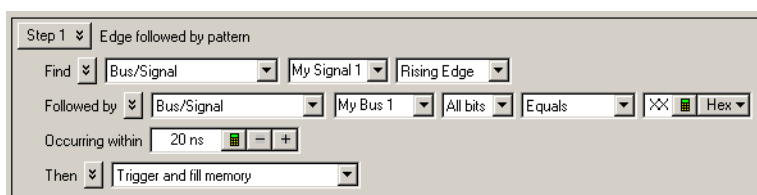
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when edge 2 does not occur within a specified time period after edge 1.



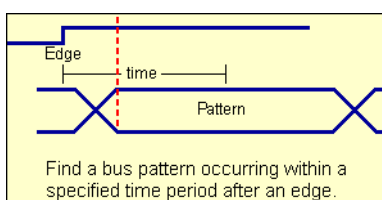
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Edge followed by pattern



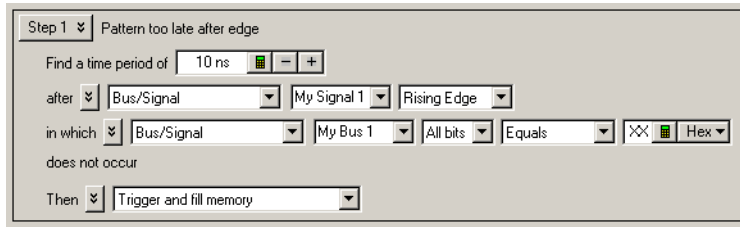
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a bus pattern occurs within a specified time period after an edge.



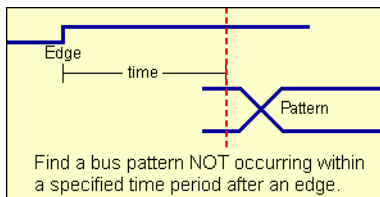
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern too late after edge



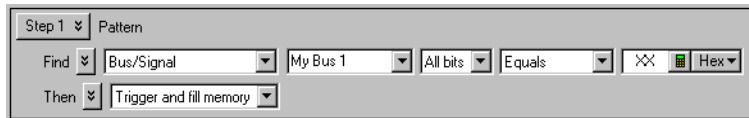
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a specified bus pattern does not occur within a specified time period after an edge.



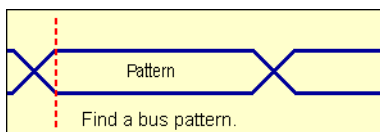
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern



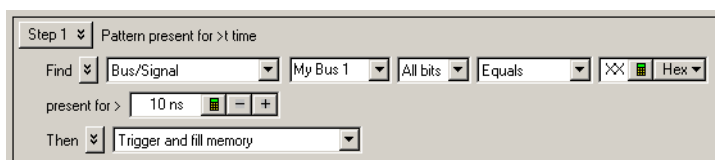
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a designated bus pattern occurs.



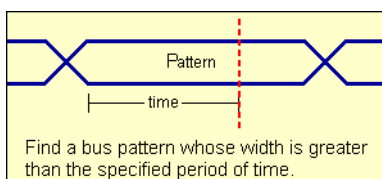
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern present for > "T" time



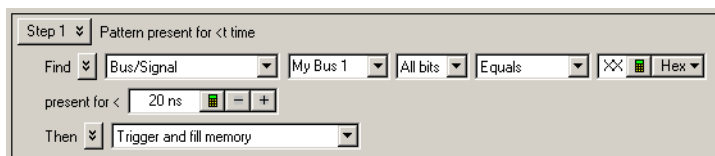
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a user-defined bus pattern is present greater than a specified time period.



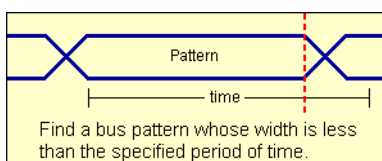
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern present for < "T" time



This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a user-defined bus pattern is present less than a specified time period.



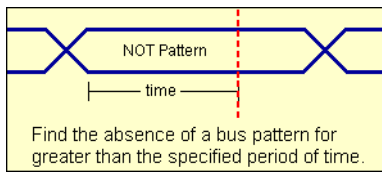
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern absent for > "T" time



This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a user-defined bus pattern is absent greater than a specified time period.

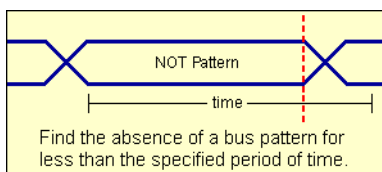


To edit this function · Specifying Advanced Triggers (see [page 152](#))

Pattern absent for < "T" time

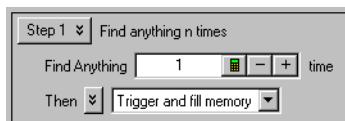


This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a user-defined bus pattern is absent less than a specified time period.

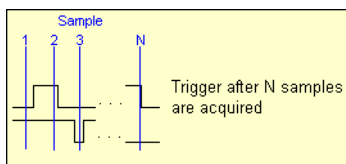


To edit this function · Specifying Advanced Triggers (see [page 152](#))

Find anything "N" times (timing)



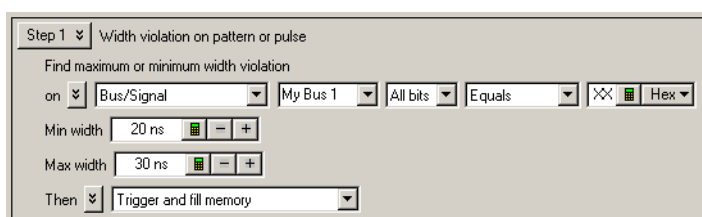
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when it sees any data (Anything) for the Nth time.



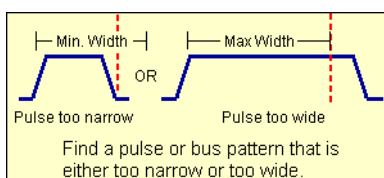
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Width violation on pattern or pulse



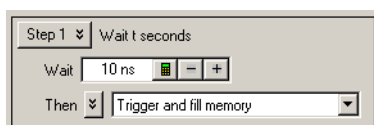
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a pulse or bus pattern is found that is either too narrow or too wide.



To edit this function

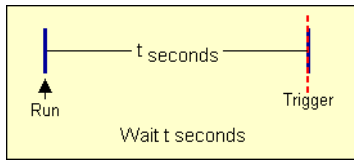
- Specifying Advanced Triggers (see [page 152](#))

Wait "T" seconds



This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers after the specified time period expires.

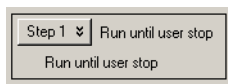
The maximum amount of time you can enter is based on the logic analyzer's sampling period. If you need to wait longer than the maximum time allowed, you can use a timer (see [page 153](#)) instead.



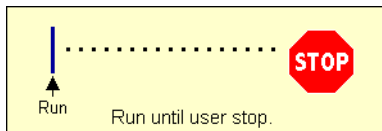
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Run until user stop (timing)



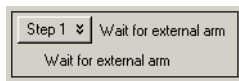
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. This trigger function sets up to never trigger. You must select the stop button to view the captured data.



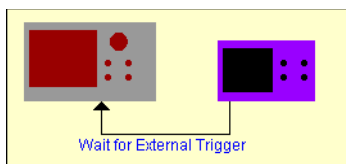
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Wait for external arm (timing)



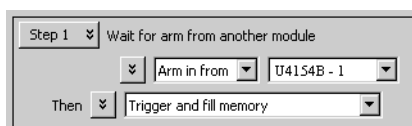
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when an external arming signal appears through the external **Trigger In** port (see [Triggering From, and Sending Triggers To, Other Modules/Instruments \(see page 186\)](#)). The **Trigger In** BNC connector is located on the front panel of the U4154A/B Logic Analyzers.



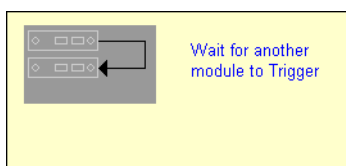
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Wait for arm from another module (timing)



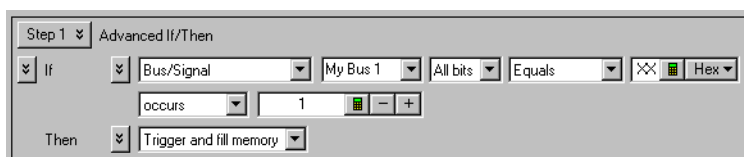
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when an arming signal from another module occurs (see [Triggering From, and Sending Triggers To, Other Modules/Instruments](#) (see [page 186](#))).



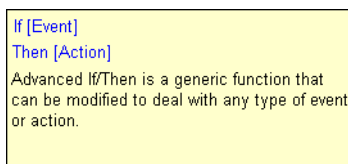
To edit this function

- [Specifying Advanced Triggers](#) (see [page 152](#))

Advanced If/Then (timing)



This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when the "If" clause becomes true.



To edit this function

- [Specifying Advanced Triggers](#) (see [page 152](#))

Advanced 2-Way Branch (timing)

This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The two-way branch is evaluated true when either of two patterns (if or Else if) are found. Depending on which pattern is found true, the appropriate "Then" action is executed.

If [Event] then [Action]
Else if [Event] then [Action]

Advanced 2-Way Branch provides the ability to check for two different events and take a different action for each event.

To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Advanced 3-Way Branch (timing)

This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The three-way branch is evaluated true when either of three patterns (If or Else if) are found. Depending on which pattern is found true, the appropriate "Then" action is executed.

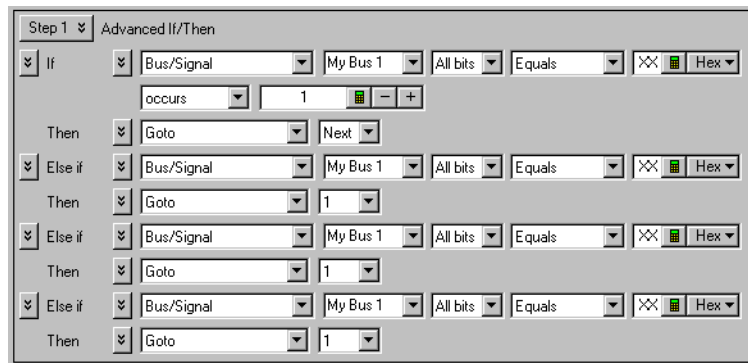
If [Event] then [Action]
Else if [Event] then [Action]
Else if [Event] then [Action]

Advanced 3-Way Branch provides the ability to check for three different events and take a different action for each event.

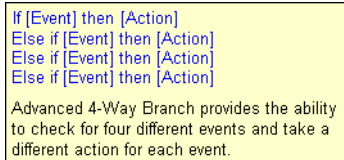
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Advanced 4-Way Branch (timing)



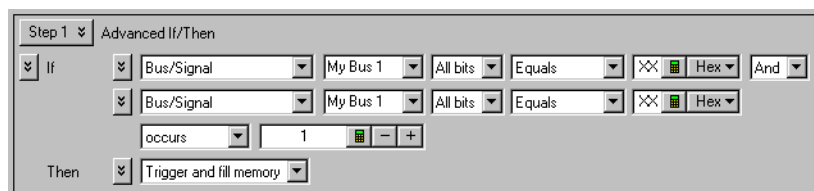
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The four-way branch is evaluated true when either of four patterns (If or Else if) are found. Depending on which pattern is found true, the appropriate "Then" action is executed.



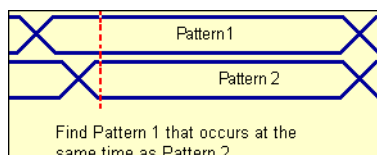
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern "AND" Pattern (timing)



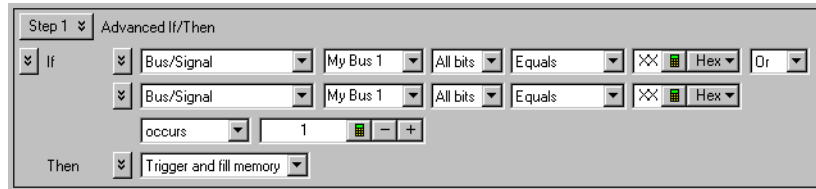
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when both pattern1 "AND" pattern2 occur at the same time, and for the specified numbers of samples (occurs).



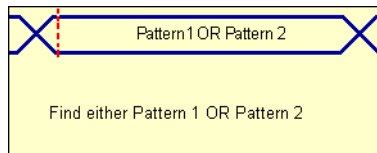
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern "OR" Pattern (timing)



This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when either pattern1 "OR" pattern2 occurs for the specified numbers of samples (occurs).



To edit this function

- Specifying Advanced Triggers (see [page 152](#))

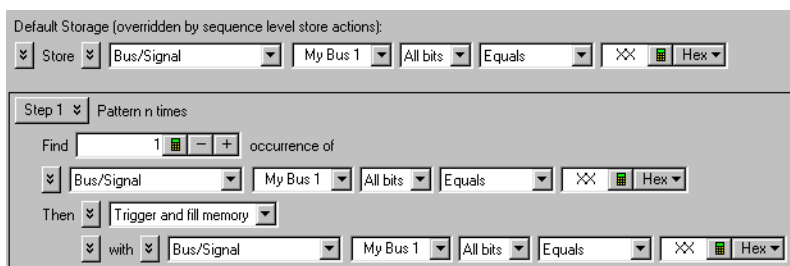
State Mode Trigger Functions

The following trigger setup examples are available as Trigger Functions in the Advanced Trigger dialog when in the state acquisition mode. To see these trigger setups in the context of an example measurement refer to Making a state analyzer measurement (see [page 51](#)).

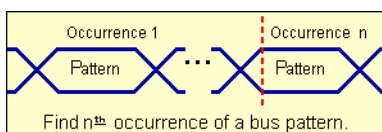
- Patterns
- Pattern "N" times (see [page 491](#))
 - "N" consecutive samples with Pattern1 (see [page 491](#))
 - Pattern1 followed by Pattern2 (see [page 492](#))
 - Pattern1 immediately followed by Pattern2 (see [page 492](#))
 - Pattern1 followed by Pattern2 before Pattern3 (see [page 493](#))
 - Too few states between Pattern1 and Pattern2 (see [page 494](#))
 - Too many states between Pattern1 and Pattern2 (see [page 494](#))
 - Pattern2 occurring too soon after Pattern1 (see [page 495](#))
 - Pattern2 occurring too late after Pattern1 (see [page 496](#))
 - Find a packet (see [page 496](#))
- Other
- Reset and start timer (see [page 497](#))
 - Find anything "N" times (see [page 497](#))
 - Run until user stop (see [page 498](#))
 - Wait for external arm (see [page 498](#))
 - Wait for arm from another module (see [page 499](#))
 - Wait "N" external clock states (see [page 499](#))

- Advanced
- Advanced If/Then (see [page 500](#))
 - Advanced 2-Way Branch (see [page 500](#))
 - Advanced 3-Way Branch (see [page 501](#))
 - Advanced 4-Way Branch (see [page 501](#))
 - Pattern "AND" Pattern (see [page 502](#))
 - Pattern "OR" Pattern (see [page 503](#))
- See Also
- To replace or insert trigger functions into trigger sequence steps (see [page 158](#))
 - Timing Mode Trigger Functions (see [page 478](#))
 - To specify default storage (see [page 167](#))
 - To store a trigger (see [page 194](#))
 - To recall a trigger (see [page 194](#))

Pattern "N" times

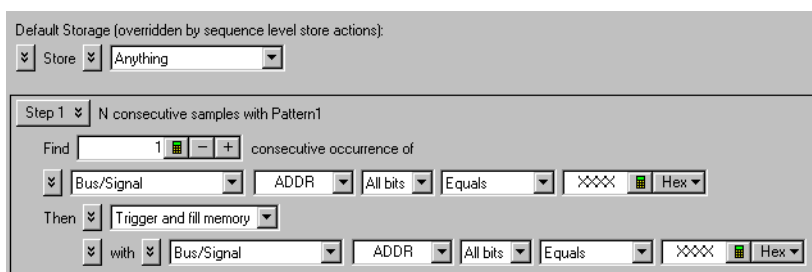


This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when it finds the nth occurrence of a bus pattern as shown below.

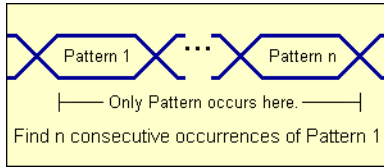


- To edit this function
- Specifying Advanced Triggers (see [page 152](#))

"N" consecutive samples with Pattern1



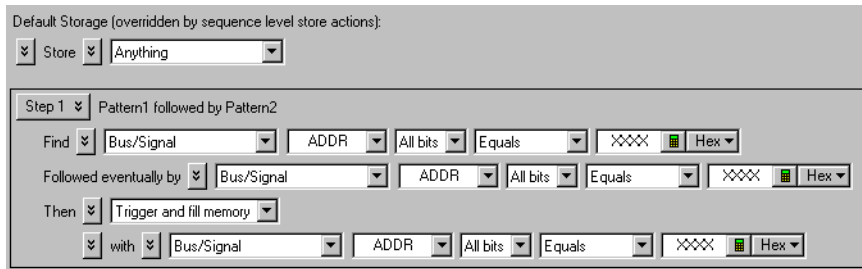
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when a bus pattern occurs a specified number times.



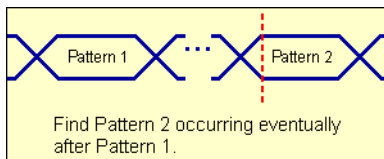
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern1 followed by Pattern2



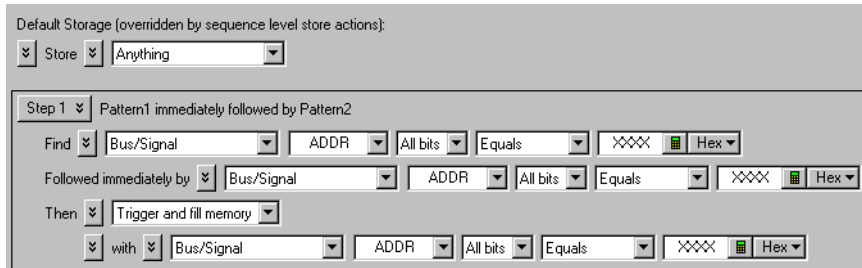
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern2 occurs eventually after pattern 1.



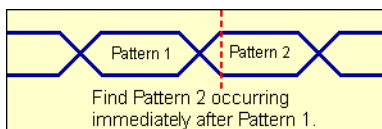
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern1 immediately followed by Pattern2



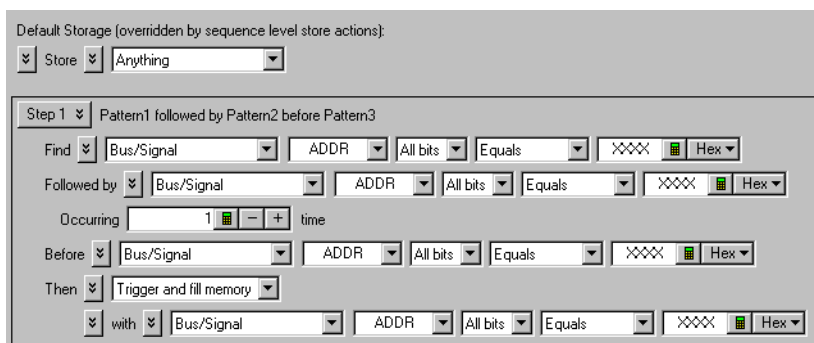
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern 2 is found immediately after exiting pattern 1.



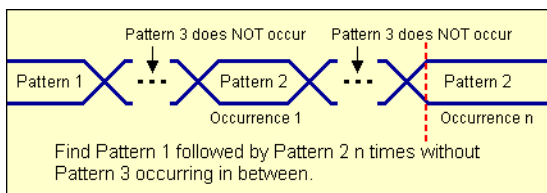
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern1 followed by Pattern2 before Pattern3



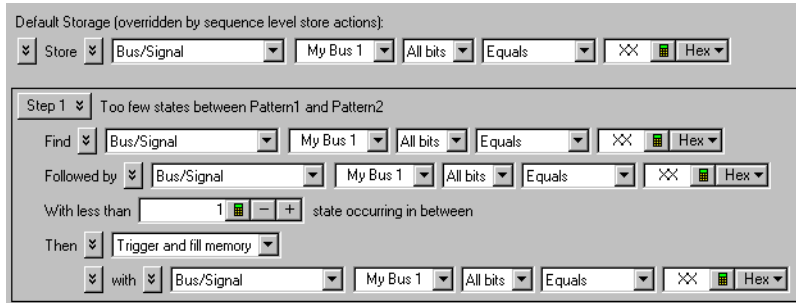
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern2 occurs eventually after pattern1, for a specified number of times, without pattern3 occurring in between.



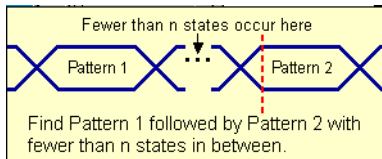
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Too few states between Pattern1 and Pattern2



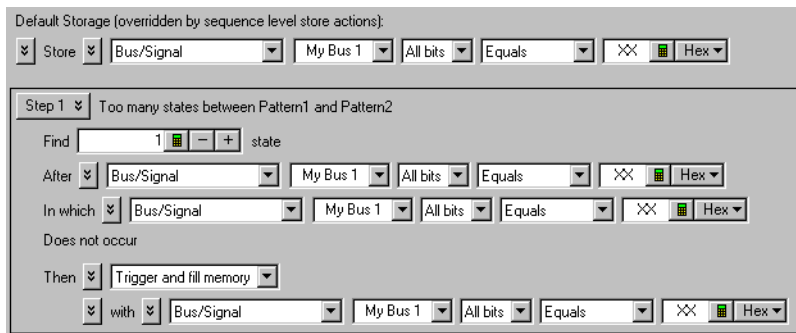
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern1 is followed by pattern2 with fewer than "N" specified states in between.



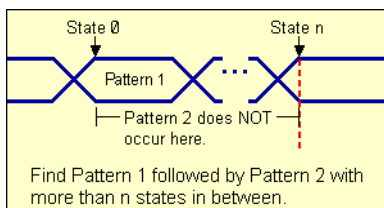
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Too many states between Pattern1 and Pattern2



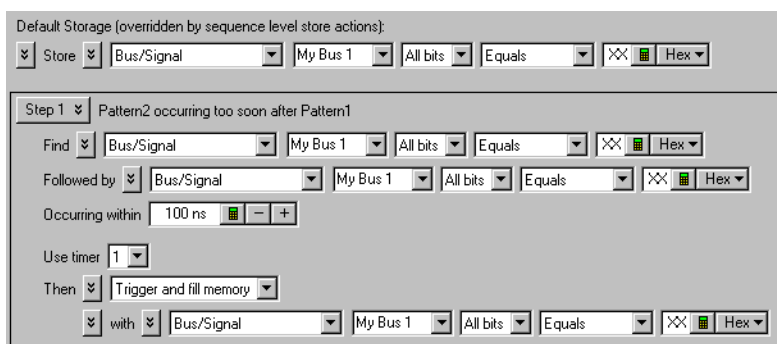
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern1 is followed by pattern2 with more than "N" specified states in between.



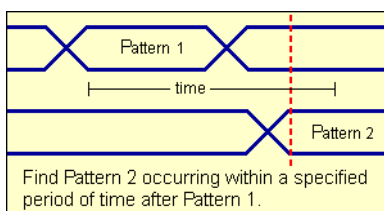
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern2 occurring too soon after Pattern1



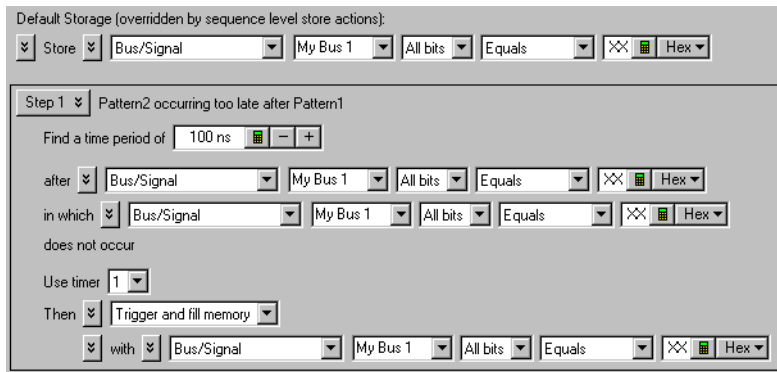
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern2 occurs within a specified time period after pattern1.



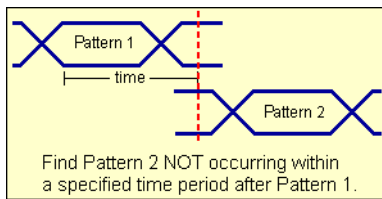
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern2 occurring too late after Pattern1

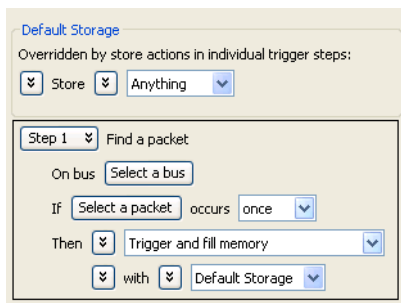


This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern2 does not occur within a specified time period after pattern1.

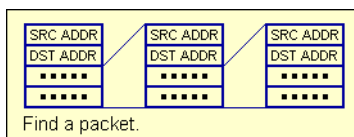


To edit this function · Specifying Advanced Triggers (see [page 152](#))

Find a packet



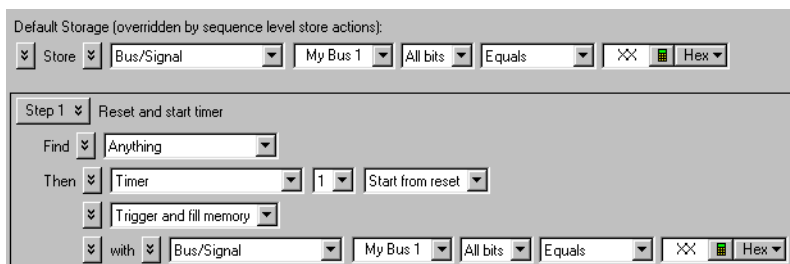
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when the selected packet and occurrence is found.



To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Reset and start timer (state)



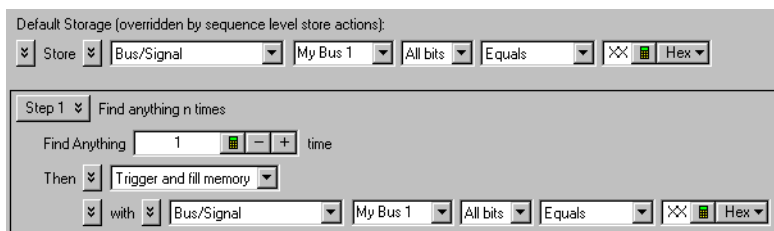
This trigger function is available when the acquisition mode is set to **State - Synchronous**. This trigger function resets a timer, then starts the timer for a specified period of time. This trigger function requires that the timer value be set in either the same trigger step, or another trigger step that follows. When the timer stops, the analyzer triggers. For more information refer to "To configure a timer (see [page 153](#))".



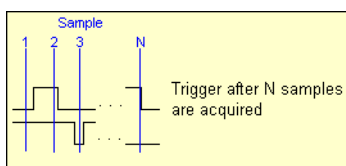
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Find anything "N" times (state)

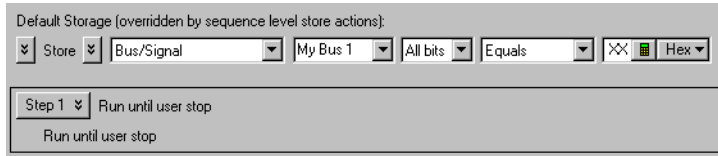


This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when any data (Anything) is seen for the Nth time. It is commonly used to create an immediate trigger, or a trigger after a user-defined delay.

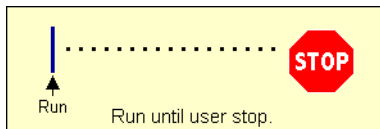


To edit this function · Specifying Advanced Triggers (see [page 152](#))

Run until user stop (state)



This trigger function is available when the acquisition mode is set to **State - Synchronous**. This trigger function sets up to never trigger. You must select the stop button to view the captured data.

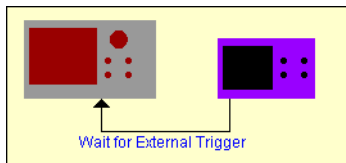


To edit this function · Specifying Advanced Triggers (see [page 152](#))

Wait for external arm (state)

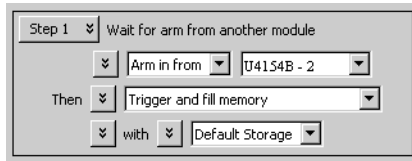


This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers when an external arming signal appears through the external **Trigger In** port (see Triggering From, and Sending Triggers To, Other Modules/Instruments (see [page 186](#))). The **Trigger In** BNC connector is located on the front panel of the U4154A/B Logic Analyzers.

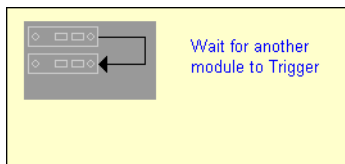


To edit this function · Specifying Advanced Triggers (see [page 152](#))

Wait for arm from another module (state)



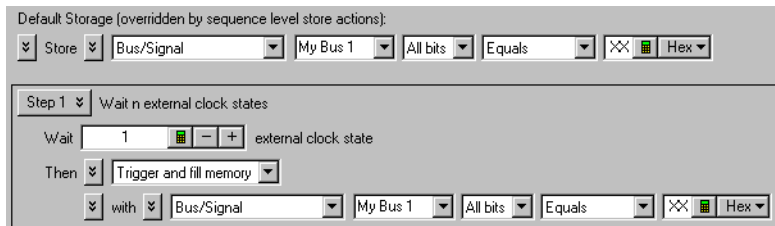
This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers when an arming signal from another module occurs (see [Triggering From, and Sending Triggers To, Other Modules/Instruments](#) (see [page 186](#))).



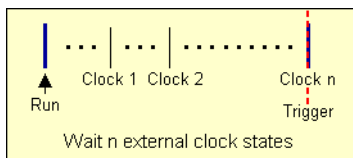
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Wait "N" external clock states



This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers on the "Nth" occurrence of the external clock signal (plus any user-defined clock qualification) from the device under test.



To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Advanced If/Then (state)

Default Storage [overridden by sequence level store actions]:
 Store: Bus/Signal, My Bus 1, All bits, Equals, Hex

Step 1: Advanced If/Then

If: Bus/Signal, My Bus 1, All bits, Equals, Hex, occurs 1

Then: Trigger and fill memory

This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers when the "If" clause becomes true.

If [Event]
 Then [Action]
 Advanced If/Then is a generic function that can be modified to deal with any type of event or action.

To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Advanced 2-Way Branch (state)

Default Storage [overridden by sequence level store actions]:
 Store: Bus/Signal, My Bus 1, All bits, Equals, Hex

Step 1: Advanced If/Then

If: Bus/Signal, My Bus 1, All bits, Equals, Hex, occurs 1

Then: Goto Next

Else if: Bus/Signal, My Bus 1, All bits, Equals, Hex

Then: Goto 1

This trigger function is available when the acquisition mode is set to **State - Synchronous**. The two-way branch is evaluated true when either of two patterns (if or Else if) are found. Depending on which pattern is found true, the appropriate "Then" action is executed.

If [Event] then [Action]
 Else if [Event] then [Action]
 Advanced 2-Way Branch provides the ability to check for two different events and take a different action for each event.

To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Advanced 3-Way Branch (state)

Default Storage (overridden by sequence level store actions):

Store: Bus/Signal My Bus 1 All bits Equals Hex

Step 1 Advanced If/Then

If: Bus/Signal My Bus 1 All bits Equals Hex
occurs: 1

Then: Goto Next

Else if: Bus/Signal My Bus 1 All bits Equals Hex
Then: Goto 1

Else if: Bus/Signal My Bus 1 All bits Equals Hex
Then: Goto 1

This trigger function is available when the acquisition mode is set to **State - Synchronous**. The three-way branch is evaluated true when either of three patterns (If or Else if) are found. Depending on which pattern is found true, the appropriate "Then" action is executed.

If [Event] then [Action]
Else if [Event] then [Action]
Else if [Event] then [Action]

Advanced 3-Way Branch provides the ability to check for three different events and take a different action for each event.

To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Advanced 4-Way Branch (state)

Default Storage (overridden by sequence level store actions):

Store: Bus/Signal My Bus 1 All bits Equals Hex

Step 1 Advanced If/Then

If: Bus/Signal My Bus 1 All bits Equals Hex
occurs: 1

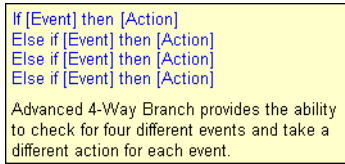
Then: Goto Next

Else if: Bus/Signal My Bus 1 All bits Equals Hex
Then: Goto 1

Else if: Bus/Signal My Bus 1 All bits Equals Hex
Then: Goto 1

Else if: Bus/Signal My Bus 1 All bits Equals Hex
Then: Goto 1

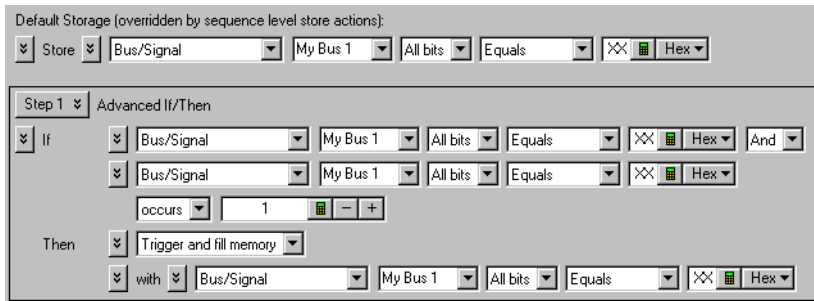
This trigger function is available when the acquisition mode is set to **State - Synchronous**. The four-way branch is evaluated true when either of four patterns (If or Else if) are found. Depending on which pattern is found true, the appropriate "Then" action is executed.



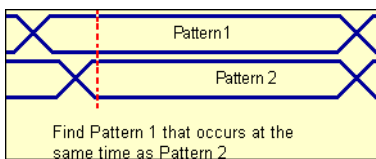
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern "AND" Pattern (state)



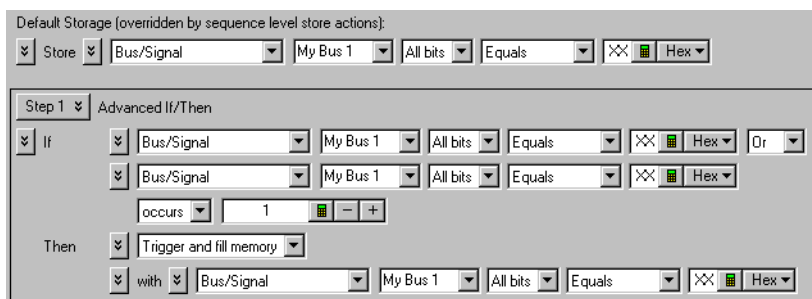
This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers when both pattern1 "AND" pattern2 occur at the same time, and for the specified numbers of samples (occurs).



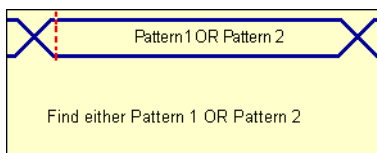
To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Pattern "OR" Pattern (state)



This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers when either pattern1 "OR" pattern2 occurs for the specified numbers of samples (occurs).



To edit this function

- Specifying Advanced Triggers (see [page 152](#))

Data Formats

- Configuration File Formats
 - ALA Format (see [page 504](#))
 - "XML Format" (in the online help)
- Data Export Formats
 - Standard CSV Format (see [page 504](#))
 - Module CSV Format (see [page 504](#))
 - Module Binary (ALB) Format (see [page 511](#))

ALA Format

The Keysight Logic Analyzer (ALA) format is the default format for saving configuration files. The ALA format is proprietary; ALA format configuration files are not intended to be read by programs other than the *Keysight Logic Analyzer* application.

ALA format configuration files contain everything that is needed to restore a session (in other words, the information necessary to reconstruct the display appearance, instrument settings, and optionally, captured data).

Configuration files are saved (see [page 198](#)) and opened (see [page 208](#)) through the File menu (see [page 386](#)).

Standard CSV Format

You can export captured data to CSV (Comma-Separated Values) format files which can then be imported by other applications like Excel.

Output is standard CSV format where the first row is the headings for the columns you have chosen to export (for example, buses/signal names, sample number, or time), and each successive row contains data for those columns, in the range specified, separated by commas (or other specified separation characters). For example:

```
"Sample Number", "My Bus 1", "My Signal 1", "Time"
-10, FE, 1, -13 ns
-9, FE, 1, -10.5 ns
-8, FE, 1, -8 ns
-7, FE, 1, -5.5 ns
-6, FE, 1, -3 ns
-5, FE, 1, -500 ps
-4, FE, 1, 2 ns
-3, FE, 1, 4.5 ns
-2, FE, 1, 7 ns
-1, FE, 1, 9.5 ns
0, FF, 1, 12 ns
1, FF, 1, 14.5 ns
2, FF, 1, 17 ns
3, FF, 1, 19.5 ns
4, FF, 1, 22 ns
5, FF, 1, 24.5 ns
6, FF, 1, 27 ns
7, FF, 1, 29.5 ns
8, FF, 1, 32 ns
9, FF, 1, 34.5 ns
10, FF, 1, 37 ns
```

Module CSV Format

You can export captured data to module CSV (Comma-Separated Values) format files which can then be post-processed.

CAUTION

Do not modify module CSV files with Microsoft Excel.

Module CSV format files contain a header section (see [page 505](#)) followed by comma separated values. For example:

```
KEYSIGHT_CSV_DATA
HEADER_BEGIN
#
# -----
# Created: Dec 5, 2005 15:38:38
# By: Keysight Logic Analyzer
# -----
#
VALUE_SEPARATOR=", "
VALUE_FILL_IN=ZEROS # ZEROS, ONES, PREVIOUS, or ERROR
TRIGGER_CORRELATION_OFFSET=0.0000000000000000e+000
TRIGGER_ROW=524288
NUM_ROWS=1048576
TIME_SOURCE_PERIOD=2.0000000000000000e-009
COLUMN "Sample Number" SAMPLE_NUMBER INTEGER
COLUMN "My Bus 1" VALUE UNSIGNED_INTEGER HEX WIDTH_BITS=8
COLUMN "My Bus 2" VALUE UNSIGNED_INTEGER HEX WIDTH_BITS=16
COLUMN "My Bus 3" VALUE UNSIGNED_INTEGER HEX WIDTH_BITS=32
COLUMN "My Signal 1" VALUE UNSIGNED_INTEGER HEX WIDTH_BITS=1
COLUMN "My Signal 2" VALUE UNSIGNED_INTEGER HEX WIDTH_BITS=1
COLUMN "Time" TIME FLOAT EXPONENT=-12 ABSOLUTE
HEADER_END
-524288,53,FFF0,004103E7,1,1,-1048576000
-524287,53,FFF0,004103E7,1,1,-1048574000
-524286,53,FFF0,004103E7,1,1,-1048572000
-524285,53,FFF0,004103E7,1,1,-1048570000
.
.
.
```

- See Also
- Module CSV and Module Binary File Header Format (see [page 505](#))
 - Module CSV Format File Characteristics (see [page 510](#))

Module CSV and Module Binary File Header Format

The header section comes at the beginning of the module CSV or module binary (ALB) file. The header definition is slightly different in the two cases.

```
KEYSIGHT_CSV_DATA KEYSIGHT_BINARY_DATA
```

REQUIRED: Key word tag to confirm the file type. The first is for module CSV text files. The second is for module binary (ALB) files. This tag must be the first characters 16 (or 19) characters in the file.

```
#
```

OPTIONAL: Comment character. All text following a # is ignored to the end of the line. Any line (except the first) may be a comment, except in module binary (ALB) files, where comments are only allowed in the header.

HEADER_BEGIN

REQUIRED: Key word tag to delimit the beginning of the header.

TABLE_BEGIN "<name>" <TableType>

A table represents data sampled at a specified timebase and sampling depth. A table can only represent a single timebase and sample depth.

OPTIONAL: If only one timebase and sample depth is needed, this keyword is not needed.

REQUIRED: If multiple timebases and sampling depths are needed, this keyword is needed to delimit the beginning of a table. The name may contain most printable characters, including embedded quote marks single or double), provided they are escaped with a backslash. For example, `TABLE_BEGIN "my \"perfect\" table" . . .`

Generally, each analog waveform channel has its own table, and there is only one table for digital channels.

When multiple tables are used, the binary data associated with each table is concatenated after the header.

<TableType> :

- ANALOG
- DIGITAL

An identifier is needed to explicitly determine what the table contains. There are specific fields that are required depending on the type of table. Analog requires the `X/Y_INC/ORG/REF`.

TIME_SOURCE PERIOD = <n.nnne-nn>

TIME_SOURCE FREQUENCY = <n.nnne+nn>

TIME_SOURCE COLUMN = "<columnName>"

OPTIONAL: How to assign a time tag to each sample row. `PERIOD` gives the sampling period; `FREQUENCY` gives the sample rate. `COLUMN` indicates which time column contains the time tags to use (ignored if `PERIOD` or `FREQUENCY` is given).

TRIGGER_ROW = <nnnn>

OPTIONAL: Row number containing the trigger sample, relative to the first data row in the file, which is row 0. The trigger row may be outside the range of rows in a file with a time line defined by a period or frequency, in which case it will be negative (trigger occurred before the first data row) or positive and greater than or equal to the number of data rows in the file (trigger occurred after the last data row in the file).

TRIGGER_CORRELATION_OFFSET = <n.nnne-nn>

OPTIONAL: Time offset in seconds (positive or negative) to apply to trigger (that is, to the T=0 reference) when correlating with data from other modules. If not given, 0.0 is assumed. The offset is rounded to the nearest picosecond.

NUM_ROWS = <nnn>

REQUIRED: If **TABLE_BEGIN** is present. Indicates the total number of data rows that will be read in from the file.

OPTIONAL: The expected number of data rows in the file. A warning is given if the number given does not match the actual number of rows in the file. This is provided as an optional sanity check to detect file truncation.

X_INC = <nnn.nnn>

OPTIONAL: Used for oscilloscope data processing.

X_ORG = <nnn.nnn>

OPTIONAL: Used for oscilloscope data processing.

X_REF = <nnn>

OPTIONAL: Used for oscilloscope data processing.

Y_INC = <nnn.nnn>

OPTIONAL: Used for oscilloscope data processing.

Y_ORG = <nnn.nnn>

OPTIONAL: Used for oscilloscope data processing.

Y_REF = <nnn>

OPTIONAL: Used for oscilloscope data processing.

Y_MIN = <nnn.nnn>

OPTIONAL: Used for oscilloscope data processing--the minimum voltage value represented in the data.

Y_MAX = <nnn.nnn>

OPTIONAL: Used for oscilloscope data processing--the maximum voltage value represented in the data.

VALUE_SEPARATOR = "<ccc>"

OPTIONAL, Module CSV only: One or more characters to be used as the separation character(s) between values in each row. If not given, the comma character is used. If the value for a column is missing (for example, two consecutive commas), its value is determined by the **VALUE_FILL_IN** property. The separation string may contain any printable character, and/or the blank and tab characters, except the following ambiguous characters:

- '#' – the comment character.
- '-', '+' – sign characters (signed decimal, also known as two's complement, and floats).
- '.' – the period, used in floating point.

- a-f, A-F – hex.
- 0-9 – digits.

VALUE_FILL_IN = PREVIOUS | ZEROS | ONES | ERROR

OPTIONAL, Module CSV only: The value to be used for a skipped cell in a row. This attribute is optional. The default is ZEROS. PREVIOUS is consistent with the VCD file format and means "use the value from the cell above". ERROR indicates that any missing value should be treated as an error. This setting applies to all columns except TIME. Any missing time tag is an error.

BYTE_ORDER = LITTLE_ENDIAN | BIG_ENDIAN

OPTIONAL, Module binary (ALB) only: The order of bytes in a multi-byte value. An Intel-compatible system uses little endian as its internal format. This applies to both integer and floating point values. The default is LITTLE_ENDIAN.

LABEL "<name>" CHANNELS = "<ChannelDescription>" [HEX | DECIMAL | OCTAL | BINARY | SYMBOL]

OPTIONAL: As described below, a COLUMN represents a pod. If there are no LABELs defined, a LABEL is created by default which has the same characteristics as the COLUMN. If a LABEL is defined, it represents a subset of the data defined by the COLUMN. Another way to look at it is that a COLUMN defines the physical data that was exported and a LABEL is an interpretation of the data. The "<name>" may contain most printable characters, including embedded quote marks (single or double), provided they are escaped with a backslash. For example, LABEL "my \ "perfect\" label". It is required that LABEL names be unique. If a duplicated name is found, only the first instance of the LABEL definition will be processed. There is an optional base which, if specified, will be the default label base. The default is HEX.

<ChannelDescription>

A channel description consists of the following format: <COLUMN_NAME> [bit position] with multiple channel descriptions separated by a comma. A <COLUMN_NAME> corresponds to the COLUMN name attribute. If there are multiple COLUMNS by the same name, the first one will be used. The bit position list is enclosed in [] and is order dependent. The MSB is on the left-hand side and the LSB is on the right-hand side. For example, [7:0] represents bits 0-7 and [7,4,3,1] represents individual bits, 1, 3, 4, and 7. Bit patterns that are not monotonically increasing from right to left are considered reordered. For example, [1,3,4,7] represents bits 7 as the LSB and bit 1 as the MSB.

SYMBOL "<name>" LABEL = "<label or column name>" VALUE = "<value> | <value..value>" [HEX | DECIMAL | OCTAL | BINARY]

TABLE_END

REQUIRED: If TABLE_BEGIN is used.

HEADER_END

REQUIRED: The explicit end of the header. Information obtained from the header is evaluated at this point to determine if the file makes sense (for example, that a time base and one or more columns were defined).

Each line of a module CSV file (including sample rows following the header) are processed by first stripping the comment character and all characters following it, and then removing all leading whitespace (except blanks or tabs if a blank or tab is part of the delimiter string). If the resulting line is then empty, it is silently skipped. Otherwise, each data row is parsed using the definitions provided by the `COLUMN` definitions in the header, defined below.

COLUMN Syntax

One or more `COLUMN` definitions are required. The order of definitions in the header corresponds to the order of the columns in the data section. At least one of these must define a `VALUE` column. `COLUMN` syntax is more complicated than the other statements, so it is given in pieces:

```
COLUMN "<name>" <CSVcolumnType> | <ALBcolumnType>
```

These are the major parts of a `COLUMN` definition. The details are different for module CSV versus module binary (ALB) files. The name may contain most printable characters, including embedded quote marks (single or double), provided they are escaped with a backslash. For example, `COLUMN "my \"perfect\" data" . . .`

<CSVcolumnType>:

- `VALUE <CSVformat> WIDTH_BITS = <nnn>`
- `TIME <CSVformat> <timeUnit> ABSOLUTE | RELATIVE`
- `LINE_NUMBER <CSVformat>`
- `SAMPLE_NUMBER <CSVformat>`
- `IGNORE`

The column type gives the purpose of the values in the column.

Line numbers begin at 0 on the first row and increase by 1 for each subsequent row.

Sample numbers begin at 0 on the trigger row, decreasing by one for each row above the trigger row and increasing by one for each row after it.

Time columns give the time position for the sample. Absolute time positions are referenced to T=0 on the trigger row. Relative time positions give the amount of time elapsed from the arrival of the previous sample to the arrival of the sample on the row with the time tag.

Value columns contain data samples to be displayed in viewers in the application. The width in bits of a value is the width assigned to the pod which represents this column in the Bus/Signal setup page. Values are truncated (or expanded) to fit the width. Signed values are sign extended. Floating point values are converted to integers of the appropriate size (after applying the scale factor, if given).

<ALBcolumnType>:

- `VALUE NBYTES = <nnn> <ALBformat> WIDTH_BITS = <nnn>`
- `TIME NBYTES = <nnn> <ALBformat> <timeUnit> ABSOLUTE | RELATIVE`
- `LINE_NUMBER NBYTES = <nnn> <ALBformat>`
- `SAMPLE_NUMBER NBYTES = <nnn> <ALBformat>`
- `IGNORE NBYTES = <nnn>`

This is the same as `<CSVcolumnType>`, with the addition of an `NBYTES` attribute that gives the number of bytes in the binary data that are assigned to the column.

<CSVformat>:

- `INTEGER [HEX | DECIMAL | OCTAL | BINARY]`
- `UNSIGNED_INTEGER [HEX | DECIMAL | OCTAL | BINARY]`
- `FLOAT [SCALE = <nnn.nnn>]`

The column format describes the way the value is represented in the file. The base is optional, and defaults to hex if not given. *SCALE* is also optional, and defaults to 1.0. The final value loaded is the value in the column, multiplied by the scale factor, rounded to the nearest integer (except for time values, which are rounded to the nearest 10^{-24} sec). All values are checked for overflow with respect to the field's *WIDTH_BITS*. Line numbers and sample numbers are treated as 32-bit quantities.

<ALBformat>:

- INTEGER
- UNSIGNED_INTEGER
- FLOAT [*SCALE* = <nnn.nnn>]

The same as <CSVformat>, except there is no numeric base. Floating point values must be in IEEE Standard 754, and so must be either four bytes or eight bytes wide.

<timeUnit>:

- *EXPONENT* = <nnn>
- *EXPONENT* = -<nnn>

The unit to apply to time tag values (in addition to the scale factor, if used with floating point values). For example, *EXPONENT* = -12 is equivalent to picosecond time ticks.

Header Example

```
KEYSIGHT_CSV_DATA
#
# ----- #
# Created: 2008 Sep 21 21:18 #
# By: Keysight Logic Analyzer #
# ----- #
#
HEADER BEGIN
TIME COLUMN="tSample"
TRIGGER_ROW=100
TRIGGER_CORRELATION_OFFSET=1.24e-9
COLUMN "Sample" SAMPLE_NUMBER INTEGER DECIMAL
COLUMN "Address" VALUE INTEGER HEX WIDTH_BITS=24
COLUMN "Data" VALUE INTEGER HEX WIDTH_BITS=16
COLUMN "Status" VALUE INTEGER BIN WIDTH_BITS=8
COLUMN "DigRF" VALUE FLOAT SCALE=16.0e+3 WIDTH_BITS=16
COLUMN "tSample" TIME INTEGER DECIMAL EXPONENT=-12 RELATIVE
VALUE_SEPARATOR ","
VALUE_FILL_IN PREVIOUS
HEADER END
# Start data
0afc38,ff98,01000110,0.114,5000
0afc3c,055a,,,2500
# embedded comment
0afc38,ff98,01010001,0.116,2500
```

Module CSV Format File Characteristics

The file must define a time line so that each sample can be associated with a time tag. To do this, a periodic rate, or a time column with absolute or relative time tags, is required (see the *TIME_SOURCE* property in the file header). Time tags in the file are verified to be monotonically increasing. The specification of a periodic rate overrides the presence of time tag columns, if any.

The file must have one or more data value columns. Data value columns have the data to load. Other columns, such as line numbers, sample numbers, and time tags, annotate that data, but are not loaded as data.

Data rows in the file are indexed beginning with row 0 first, then 1, and so on. Row numbers are file-centric. They are not the same as the sample numbers in the module's listing. Sample numbers are always relative to the location of the trigger row, not the first row in the file.

The file header has a property, `TRIGGER_ROW`, giving the row number, in the file, associated with logic analyzer trigger. The trigger row is defined as the time origin, and contains $T=0$.

In a multi-module logic analyzer, only one module's trigger is at $T=0$. All the others are offset by hardware delays associated with the propagation of the trigger signal from the originating module to the receiving modules. The time delay is indicated by the `TRIGGER_CORRELATION_OFFSET` property. This is the amount of time that must be added to the time of each sample in the file to correctly position each sample on a time line with samples from another module in the system.

An interesting case arises if the trigger sample is not included in the export range. This can legitimately occur if the time source is periodic. In this case, the `TRIGGER_ROW` value will indicate a row not in the file. For example, if the first data row in the file is the first sample after the trigger in the module, then `TRIGGER_ROW=-1`. Conversely, if the trigger was the first sample after the last row in the exported data, `TRIGGER_ROW` is equal to the number of rows in the file. If the trigger was two samples after the last row, `TRIGGER_ROW` is the number of rows in the file plus one, and so on. Sample numbers are handled in a similar manner, with sample 0 on the trigger row. If the trigger row is not in the file, sample 0 is not in the file either.

These statements about the contents of a module CSV file will always be true:

- The first data row in the file is row 0. The next is 1, then 2, and so on.
- The `TRIGGER_ROW` is always relative to the first row in the file (row 0). The trigger row may or may not be in the file.
- The time associated with the `TRIGGER_ROW` is always 0. The `TRIGGER_CORRELATION_OFFSET` property allows correct correlation to other data sets.
- The sample number on the `TRIGGER_ROW` (if a sample number column is present) is always 0. Sample numbers increase by one for each row after the trigger row; they decrease by one for each row before the trigger. Sample numbers are not the same as row numbers.
- Line numbers, if exported to the file, always begin at 0 on row 0, then increase by one for each row thereafter: `lineNumber = rowNum`.

Inconsistencies between a $T=0$ time tag, sample numbers and the `TRIGGER_ROW` property are resolved in this order:

- 1 The data row containing an absolute time tag of 0 is the trigger row and sample numbers are re-aligned if necessary to begin at zero on that row.
- 2 If the time line is defined by a period or frequency, and a sample number column is given, then the data row with sample number 0 is the trigger row. (This may or may not be within the span of the file itself).
- 3 Finally, the `TRIGGER_ROW` property is used. If time tags (relative or absolute) are used, the trigger row will be moved if it is not within the file.

See Also • [Module CSV and Module Binary File Header Format \(see page 505\)](#)

Module Binary (ALB) Format

You can export captured data to module binary files.

Module binary format files contain a header section (see [page 505](#)) followed by binary data. For example:

```

KEYSIGHT_BINARY_DATA
HEADER_BEGIN
#
# -----
# Created: Dec 5, 2005 16:14:36
# By: Keysight Logic Analyzer
# -----
#
BYTE_ORDER=LITTLE_ENDIAN
TRIGGER_CORRELATION_OFFSET=0.000000000000000e+000
TRIGGER_ROW=524288
NUM_ROWS=1048576
TIME_SOURCE PERIOD=2.000000000000000e-009
COLUMN "Sample Number" SAMPLE_NUMBER NBYTES=4 INTEGER
COLUMN "My Bus 1" VALUE NBYTES=1 UNSIGNED_INTEGER WIDTH_BITS=8
COLUMN "My Bus 2" VALUE NBYTES=2 UNSIGNED_INTEGER WIDTH_BITS=16
COLUMN "My Bus 3" VALUE NBYTES=4 UNSIGNED_INTEGER WIDTH_BITS=32
COLUMN "My Signal 1" VALUE NBYTES=1 UNSIGNED_INTEGER WIDTH_BITS=1
COLUMN "My Signal 2" VALUE NBYTES=1 UNSIGNED_INTEGER WIDTH_BITS=1
COLUMN "Time" TIME NBYTES=8 FLOAT EXPONENT=-12 ABSOLUTE
HEADER_END
<binary data>

```

The binary data section of the file contains only binary data. Binary data is organized by rows, where there are a fixed number of columns within a row (the number of COLUMN definitions in the header) and a fixed number of bytes per column (1, 2, 4, 8, or 16 as specified by NBYTES in the COLUMN definition). This simplifies reading the data, because these sizes match the native C language sizes of `char`, `short`, `long`, and `long long` (`__int64`). Values of 16 bytes contain 128 bits which is the widest bus supported in the logic analysis system. By restricting exported sizes to these values, an ALB file with no header and only one bus/signal becomes a simple dump of an array of integers.

See Also • [Module CSV and Module Binary File Header Format \(see page 505\)](#)

Object File Formats Supported by the Symbol Reader

The logic analysis system can read symbol files in the following formats:

- OMF96
- OMFx86
- IEEE-695
- ELF/DWARF
- ELF/stabs
- TI COFF

For ELF/DWARF1, ELF/stabs, and ELF/stabs/Mdebug files, C++ symbols are demangled so that they can be displayed in the original C++ notation. To improve performance for these ELF symbol files, type information is not associated with variables. Hence, some variables (typically a few local static variables) may not have the proper size associated with them. They may show a size of 1 byte and not the correct size of 4 bytes or even more. All other information function ranges, line numbers, global variables and filenames will be accurate. These behaviors may be changed by editing the readers.ini (see [page 136](#)) file.

- See Also
- To load symbols from a file (see [page 134](#))
 - To change symbol reader options (see [page 136](#))
 - To create an ASCII symbol file (see [page 135](#))

General-Purpose ASCII (GPA) Symbol File Format

General-purpose ASCII (GPA) format files are loaded into a logic analyzer just like other object files.

If your compiler does not produce object files in a supported format, or if you want to define symbols that are not included in the object file, you can create an ASCII format symbol file.

Typically, ASCII format symbol files are created using text processing tools that convert the symbol table information from a compiler or linker map output file.

Different types of symbols are defined in different records in the GPA file. Record headers are enclosed in square brackets, for example, [USER]. For a summary of GPA file records and associated symbol definition syntax, refer to the General-Purpose ASCII (GPA) Record Format Summary (see [page 514](#)).

Each entry in the symbol file must consist of a symbol name followed by an address or address range.

The address or address range must be a hexadecimal number. It must appear on the same line as the symbol name, and it must be separated from the symbol name by one or more blank spaces or tabs. Address ranges must be in the following format:

```
beginning address..ending address
```

The following example defines two symbols that correspond to address ranges and one symbol that corresponds to a single address.

```
main      00001000..00001009
test      00001010..0000101F
var1      00001E22      #this is a variable
```

For more detailed descriptions of GPA file records and associated symbol definition syntax, refer to the following topics:

- SECTIONS (see [page 515](#))
- FUNCTIONS (see [page 516](#))
- USER (see [page 516](#))
- VARIABLES (see [page 518](#))
- SOURCE LINES (see [page 518](#))
- START ADDRESS (see [page 519](#))
- Comments (see [page 519](#))

General-Purpose ASCII (GPA) Record Format Summary

```
Format  [SECTIONS (see page 515)]
        section_name start..end attribute

        [FUNCTIONS (see page 516)]
        func_name start..end

        [USER (see page 516)]
        sym_name value base
        sym_name start_value [size] base
        sym_name start_value..end_value base

        [VARIABLES (see page 518)]
        var_name start [size]
        var_name start..end
```

```
[SOURCE LINES (see page 518)]
File: file_name
line# address
```

```
[START ADDRESS (see page 519)]
address
```

```
#comment text (see page 519)
```

Lines without a preceding header are assumed to be symbol definitions in one of the [USER] formats.

Example This is an example GPA file that contains several different kinds of records.

```
[SECTIONS]
prog      00001000..0000101F
data      40002000..40009FFF
common    FFFF0000..FFFF1000

[FUNCTIONS]
main      00001000..00001009
test      00001010..0000101F

[USER]
bdontcare 00x1    binary
hvalue    00EF    hex
drange    0..99   decimal
srange    -23 20   signed decimal # The 20 is a decimal value for size.

[VARIABLES]
total     40002000 4
value     40008000 4

[SOURCE LINES]
File: main.c
10        00001000
11        00001002
14        0000100A
22        0000101E

File: test.c
5         00001010
7         00001012
11        0000101A
```

SECTIONS

Use SECTIONS to define symbols for regions of memory, such as sections, segments, or classes.

NOTE

To enable section relocation, section definitions must appear before any other definitions in the file.

NOTE

If you use section definitions in a GPA symbol file, any subsequent function or variable definitions must be within the address ranges of one of the defined sections. Functions and variables that are not within the range are ignored.

Format `[SECTIONS]`
`section_name start..end attribute`

Menu	Description
<code>section_name</code>	A symbol representing the name of the section.
<code>start</code>	The first address of the section, in hexadecimal.
<code>end</code>	The last address of the section, in hexadecimal.
<code>attribute</code>	(optional) Attribute may be one of the following: NORMAL (default) - The section is a normal, relocatable section, such as code or data. NONRELOC - The section contains variables or code that cannot be relocated. In other words, this is an absolute segment. AddReloc <code>reloc_offset</code> - Lets you specify an offset value (up to 32-bit) for a relocated section. SetReloc <code>reloc_value</code> - Lets you specify a new base value (up to 32-bit) for a relocated section.

Example `[SECTIONS]`

```

prog      00001000..00001FFF
data      00002000..00003FFF
display_io 00008000..0000801F  NONRELOC
sect3     00003000..00003FFF  AddReloc   1000
sect4     00005000..00005FFF  SetReloc   0000F000

```

FUNCTIONS

Use FUNCTIONS to define symbols for program functions, procedures or subroutines.

Format `[FUNCTIONS]`
`func_name start..end`

Menu	Description
<code>func_name</code>	A symbol representing the function name.
<code>start</code>	The first address of the function, in hexadecimal.
<code>end</code>	The last address of the function, in hexadecimal.

Example `[FUNCTIONS]`

```

main      00001000..00001009
test      00001010..0000101F

```

USER

Under the [USER] record header, you can create symbols with don't care values, and you can use value number bases other than hex.

USER is the default record type; this means symbols defined without any record header ([USER], [VARIABLE], etc.) are assumed to be USER symbols.

USER symbol definitions can have 128-bit values; symbol definitions in all other record types are limited to 32-bit values.

```
Format [USER]
sym_name value base
sym_name start_value [size] base
sym_name start_value..end_value base
```

* use quotes(") around names or values with spaces.

Menu	Description
sym_name	A symbol name.
value	The value of the symbol.
base	The number base of the value(s); can be: <ul style="list-style-type: none"> · binary · octal · hex (or hexadecimal or blank) · decimal · signed decimal (two's complement)
start_value	The low value of the range.
end_value	The high value of the range.
size	The size of the range of values, in decimal.

```
Example [USER]
bdontcare 00x1 binary
bvalue 0011 binary
brange 0000..0011 binary
brange2 0011 9 binary # The 9 is a decimal value for size
.
odontcare 00x7 octal
ovalue 0077 octal
orange 0000..0077 octal
orange2 0000 99 octal # The 99 is a decimal value for size.
hdontcare 00xF hex
hvalue 00EF hex
hrange 0000..00FF hex
hrange3 0000 99 hex # The 99 is a decimal value for size.
dvalue 1090 decimal
drange 0..99 decimal
drange2 0 100 decimal # The 100 is a decimal value for size.
svalue -23 signed decimal
srange -23..-5 signed decimal
srange -23 20 signed decimal # The 20 is a decimal value for size.
"hvalue with space" "01XX FFFF" hex
```

"hrange with space" "0100 FFFF".. "0400 FFFF" hex # Applies to other bases also.

VARIABLES

You can specify symbols for variables using:

- The address of the variable.
- The address and the size of the variable.
- The range of addresses occupied by the variable.

If you specify only the address of a variable, the size is assumed to be 1 byte.

Format [VARIABLES]
 var_name start [size]
 var_name start..end

Menu	Description
var_name	A symbol representing the variable name.
start	The first address of the variable, in hexadecimal.
end	The last address of the variable, in hexadecimal.
size	(optional) The size of the variable, in bytes, in decimal.

Example [VARIABLES]
 subtotal 40002000 4
 total 40002004 4
 data_array 40003000..4000302F
 status_char 40002345

SOURCE LINES

Use SOURCE LINES to associate addresses with lines in your source files.

Format [SOURCE LINES]
 File: file_name
 line# address

Menu	Description
file_name	The name of a file.
line#	The number of a line in the file, in decimal.
address	The address of the source line, in hexadecimal.

Example [SOURCE LINES]
 File: main.c
 10 00001000
 11 00001002
 14 0000100A
 22 0000101E

See Also · [Viewing Source Code Associated with Captured Data \(see page 277\)](#)

START ADDRESS

Format `[START ADDRESS]`
`address`

Menu	Description
<code>address</code>	The address of the program entry point, in hexadecimal.

Example `[START ADDRESS]`
`00001000`

Comments

Use the # character to include comments in a file. Any text following the # character is ignored. You can put comments on a line alone or on the same line following a symbol entry.

Format `#comment text`

Example `#This is a comment`

Product Overviews

- U4164A Logic Analyzer Product Overview (see [page 520](#))
- U4154A Logic Analyzer Product Overview (see [page 522](#))
- U4154B Logic Analyzer Product Overview (see [page 523](#))
- 16850-Series Logic Analyzer Product Overview (see [page 524](#))
- 16860-Series Logic Analyzer Product Overview (see [page 528](#))
- *Keysight Logic and Protocol Analyzer Application* Product Overview (see [page 532](#))

See Also • Tutorial - Getting to know your logic analyzer (see [page 40](#))

U4164A Logic Analyzer Product Overview

The Keysight U4164A logic analyzer is a 136 channel AXIe based high speed state and timing logic analyzer with new features such as:

- DDR4 and LPDDR4 probing for data rates over 2.5 Gb/s using the new Quad sampling state mode and single touch probing
- deeper memory
- ¼ channel 10GHz Timing mode
- deskew of timing traces by individual channels

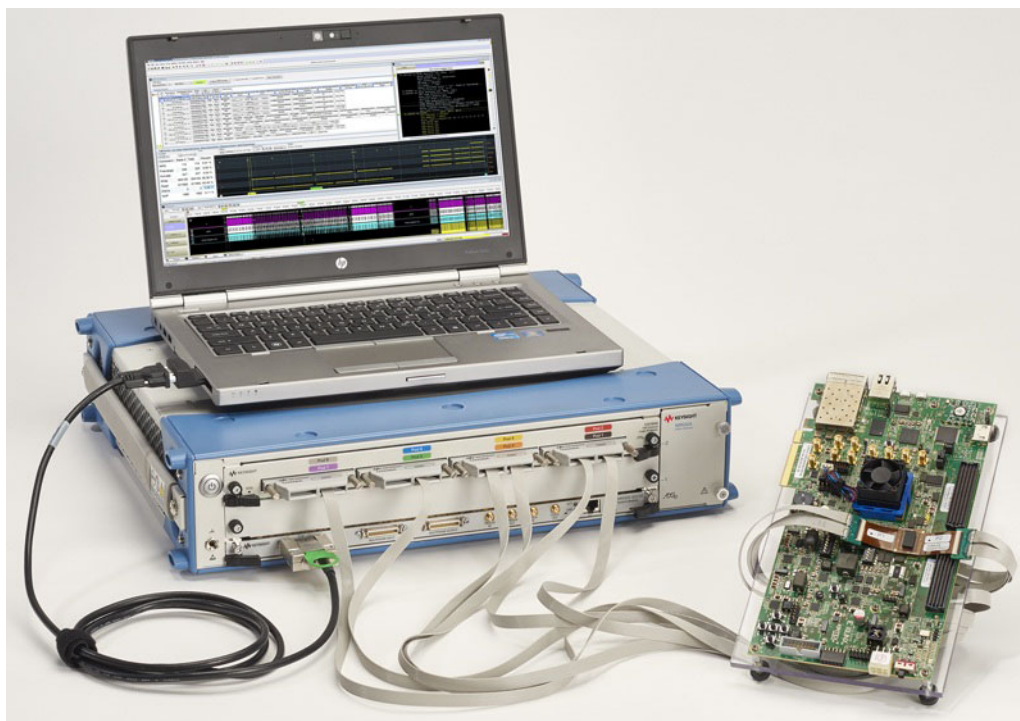
The module can be used as a general purpose logic analysis system as well as for debug, validation and analysis of DDR and LPDDR memory systems. It allows maximum DQ visibility with simultaneous Read and Write traffic capture for DDR4 and LPDDR4 without overloading the system under test. It supports up to 4 Gb/s State Mode and 10 GHz Timing Mode.

U4164A comes as an instrument module that you can install in one of the slots of the Keysight AXIe chassis. This chassis is a modular instrument chassis that supports complex and high density testing. The chassis provides slots for installing multiple instrument modules such as the U4164A Logic Analyzer module.

You can install a single U4164A module or multiple U4164A modules in slots of an AXIe chassis to increase the channel count. You can connect upto three U4164A modules in an AXIe chassis to form a three-card set with 408 channels in Full Channel mode.

You can configure, control, and use the U4164A Logic Analyzer module through the Keysight Logic and Protocol Analyzer application (version 6.2 and above). You can install this application on a host PC (a laptop or a desktop with a PCIe interface or the Keysight M9536A Embedded Controller Module). The laptop or desktop host PC connects to the U4164A module through the PCIe x8 interface of the AXIe chassis.

The following figure displays a U4164A Logic Analyzer module installed in a 2-slot AXIe chassis and connected to a DUT via W4641A BGA interposer and U4208A and U4209A probe cables.



CAUTION

You must power down the AXIe chassis before inserting, replacing, or removing the U4164A Logic Analyzer module. The enclosure surface of the U4164A module may become hot during use. If you need to remove the module, first power down the AXIe chassis, wait for at least five minutes to allow the module to cool, and then pull the module out of the chassis.

To know more about the hardware components of the U4164A Logic Analyzer module and how to set up this module in AXIe chassis, refer to the [AXIe Based Logic Analysis and Protocol Test Modules Installation Guide](#).

Supplied Accessories

- Flex cables for connecting multiple U4164A modules in different slots of an AXIe chassis
- Accessory pouch
- Pod connector cables for connecting Logic Analyzer pods to probes.

Optional Accessories

- Interposers, Probes and cables

Refer to the U4164A Data Sheet (5992-1057EN) on www.keysight.com to get a list of supported interposers, probes, and cables for U4164A. Details of these interposers, probes, and cables are in their respective user guides available on www.keysight.com.

See Also [Tutorial - Getting to know your logic analyzer \(see page 40\)](#)

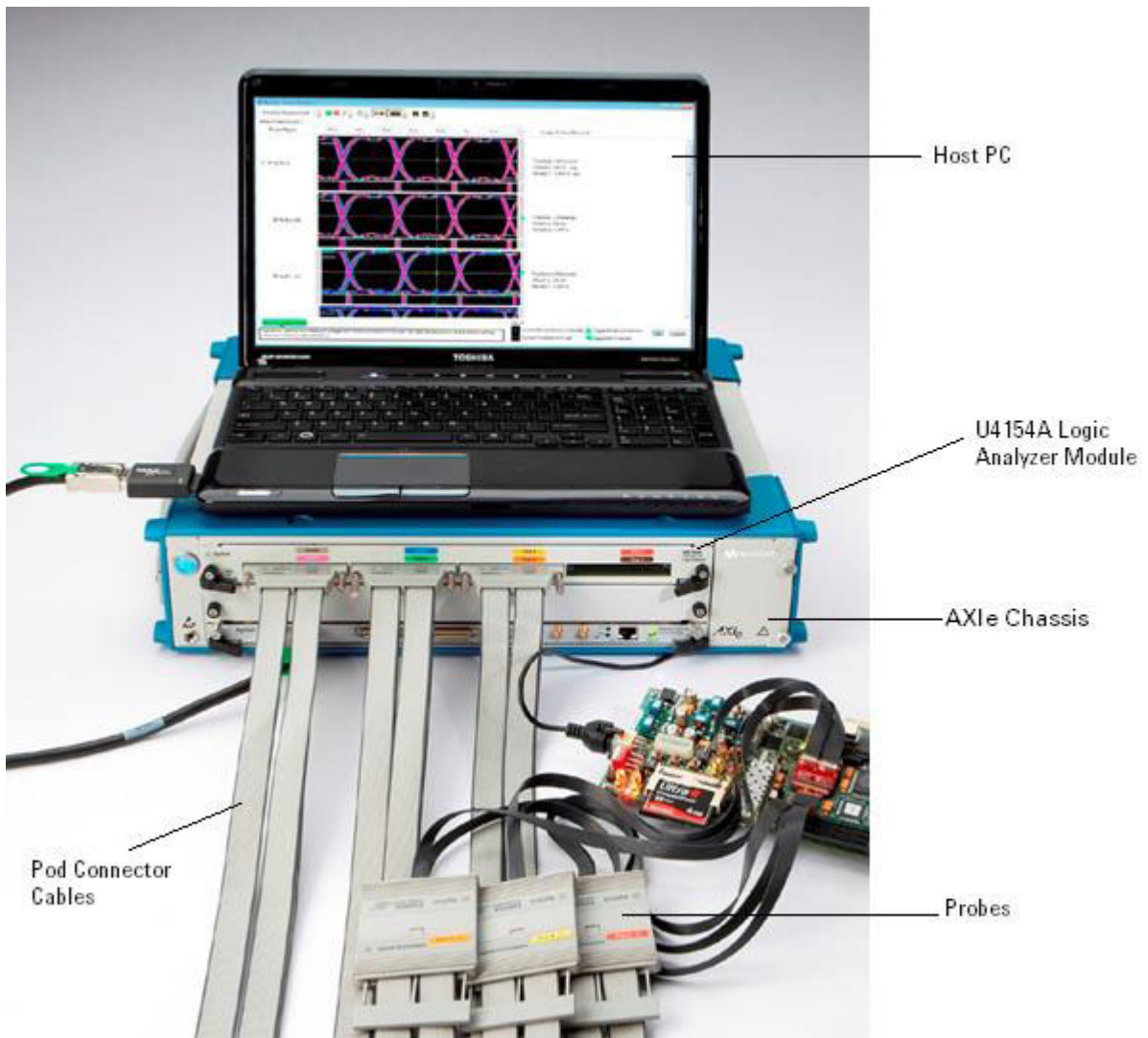
U4154A Logic Analyzer Product Overview

The Keysight U4154A logic analyzer is a 136 channel AXIe based high speed state and timing logic analyzer. It provides reliable data capture with powerful analysis and validation tools for high data rate memory based products. It helps you to perform functional validation and characterization of your memory based products.

U4154A comes as an instrument module that you can install in one of the slots of the Keysight AXIe chassis. The Keysight AXIe chassis is a modular instrument chassis that supports complex and high density testing. The chassis provides slots for installing multiple instrument modules such as the U4154A Logic Analyzer module.

You can configure, control, and use the U4154A Logic Analyzer module through the Keysight Logic Analyzer application (version 5.0 and above). You can install this application on a host PC (a laptop or a desktop with a PCIe interface or the Keysight M9536A Embedded Controller Module). The laptop or desktop host PC connects to the U4154A module through the PCIe x8 interface of the AXIe chassis.

The following figure displays a sample setup of the U4154A Logic Analyzer module in one of the slots of the Keysight AXIe chassis.



You can install a single U4154A module or multiple U4154A modules in slots of an AXIe chassis to increase the channel count. You can connect two U4154A modules in an AXIe chassis to form a two-card set with 272 channels.

CAUTION

You must power down the AXIe chassis before inserting, replacing, or removing the U4154A Logic Analyzer module. The enclosure surface of the U4154A module may become hot during use. If you need to remove the module, first power down the AXIe chassis, wait for at least five minutes to allow the module to cool, and then pull the module out of the chassis.

To know more about the hardware components of the U4154A Logic Analyzer module and how to set up this module in AXIe chassis, refer to the  *AXIe Based Logic Analysis and Protocol Test Modules Installation Guide*.

- Supplied Accessories
 - Flex cables for connecting multiple U4154A modules in different slots of an AXIe chassis
 - Accessory pouch
 - Pod connector cables for connecting Logic Analyzer pods to probes.
- Optional Accessories
 - Probes.
- See Also
 - Tutorial - Getting to know your logic analyzer (see [page 40](#))
 - U4154A Logic Analyzer Specifications and Characteristics (see [page 592](#))

U4154B Logic Analyzer Product Overview

The Keysight U4154B logic analyzer is a 136 channel AXIe based high speed state and timing logic analyzer. It provides reliable data capture with powerful analysis and validation tools for high data rate memory based products. It helps you to perform functional validation and characterization of your memory based products.

U4154B comes as an instrument module that you can install in one of the slots of the Keysight AXIe chassis. This chassis is a modular instrument chassis that supports complex and high density testing. The chassis provides slots for installing multiple instrument modules such as the U4154B Logic Analyzer module.

You can install a single U4154B module or multiple U4154B modules in slots of an AXIe chassis to increase the channel count. You can connect upto three U4154B modules in an AXIe chassis to form a three-card set with 408 channels in Full Channel mode and 204 channels in Half Channel mode.

You can configure, control, and use the U4154B Logic Analyzer module through the Keysight Logic and Protocol Analyzer application (version 6.0 and above). You can install this application on a host PC (a laptop or a desktop with a PCIe interface or the Keysight M9536A Embedded Controller Module). The laptop or desktop host PC connects to the U4154B module through the PCIe x8 interface of the AXIe chassis.

The following figure displays a U4154B Logic Analyzer module.



CAUTION

You must power down the AXle chassis before inserting, replacing, or removing the U4154B Logic Analyzer module. The enclosure surface of the U4154B module may become hot during use. If you need to remove the module, first power down the AXle chassis, wait for at least five minutes to allow the module to cool, and then pull the module out of the chassis.

To know more about the hardware components of the U4154B Logic Analyzer module and how to set up this module in AXle chassis, refer to the [AXle Based Logic Analysis and Protocol Test Modules Installation Guide](#).

- Supplied Accessories
 - Flex cables for connecting multiple U4154B modules in different slots of an AXle chassis
 - Accessory pouch
 - Pod connector cables for connecting Logic Analyzer pods to probes.
- Optional Accessories
 - Probes.
- See Also
 - Tutorial - Getting to know your logic analyzer (see [page 40](#))
 - U4154B Logic Analyzer Specifications and Characteristics (see [page 596](#))

16850-Series Logic Analyzer Product Overview

The Keysight Technologies 16850-series logic analyzers are standalone benchtop logic analyzers that range from 34 to 136 logic acquisition channels, depending on the model.

Model Comparison

Keysight model number:	16851A	16852A	16853A	16854A
Logic acquisition channels:	34	68	102	136

- Features, Logic Acquisition
 - 2 M to 128 M memory depth per channel (depending on memory option), software upgradeable.
 - 350 MHz or 700 MHz maximum state data rate (depending on state speed option), software upgradeable.
 - Full Channel Timing Mode at 2.5 GHz sampling with 12.5 GHz Timing Zoom.
 - Half Channel Timing Mode at 5.0 GHz sampling with 12.5 GHz Timing Zoom.

Features,
Mainframe

- Automated threshold and sample position setup.
- 12.5 GHz timing zoom with 256 K samples.
- Built-in 15 inch TFT color LCD display, 1,024 x 768 (XGA) resolution. Touch screen with Option 103. See Tips for Using the Touch Screen (see [page 525](#)).
- Front panel knob and buttons. See 16850 Series Front Panel Operation (see [page 527](#)).
- 500 GB hard disk drive (or external hard drive option).
- 10Base-T, 100Base-T, 1000Base-T LAN port.
- USB 2.0 ports (six total, two on front, four on back).
- One PCI expansion slot.
- One PCI Express x1 expansion slot.
- Windows 7 operating system.
- *Keysight Logic and Protocol Analyzer* application which takes the complexity out of making logic analyzer measurements. You can perform all operations directly from one window.

CAUTION

When powering off the 16850-series logic analyzer, wait until the fans stop turning (about 15 seconds) before turning the logic analyzer back on. This ensures that internal circuitry restarts in a known state. (For more information on powering off the logic analyzer, see the ["16850-Series Logic Analyzer Installation/Quick Start Guide"](#).)

Supplied
Accessories

- PS/2 mouse
- PS/2 mini keyboard
- Accessory pouch
- Power cord

Optional
Accessories

- Probes

See Also

- Tutorial - Getting to know your logic analyzer (see [page 40](#))
- 16850-Series Logic Analyzer Specifications and Characteristics (see [page 600](#))

Tips for Using the Touch Screen

The 16850-series logic analyzer frames with Option 103 have a touch screen. Here are some tips for using the touch screen:

Use firm, even pressure on the touch screen.

You may prefer to use a stylus.

Use the front panel marker knobs to place markers. Placing markers is hard to do accurately using the touch screen. (Any marker can be selected using the Marker Choose button.)

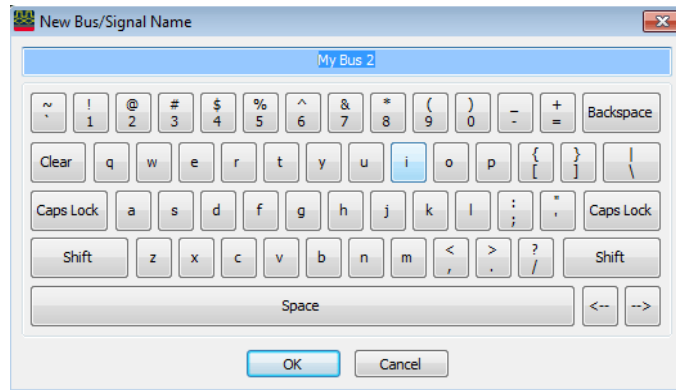
For trees, the touchable area around the +/- buttons is expanded.

For option selections, both the option and the caption are active.

To open a keyboard
dialog

Inside the *Keysight Logic Analyzer* application:

- Press the  keyboard button in any edit field. This opens a dialog for entering field values.



Outside the *Keysight Logic Analyzer* application:

- Press the front panel **Keyboard** button. This opens the Microsoft On-Screen Keyboard and the touch screen Event Selector.



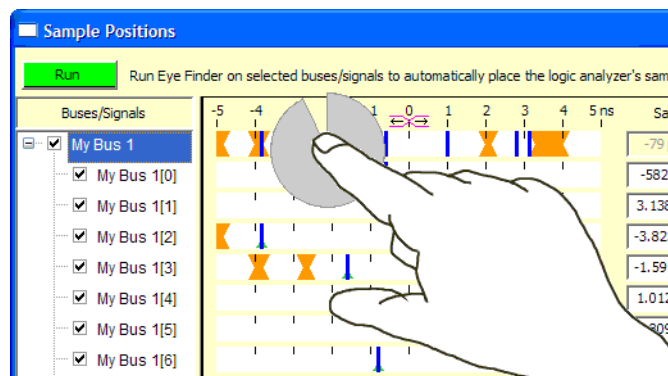
(You can also choose **Start>All Programs>Accessories>Accessibility>On-Screen Keyboard**.)

To access right mouse button behavior

Inside the *Keysight Logic Analyzer* application, most right mouse button behavior is accessible just by touching the screen.

Outside the *Keysight Logic Analyzer* application:

- Press down on the touch screen until a full circle is drawn around your finger; then, a right-click occurs.



- Or, you can choose **Start>All Programs>Keysight Logic Analyzer>Utilities>Touch Screen>Event Selector** to open the window:



Touching inside this window causes your next touch to act as a right-click.

To recalibrate the touch screen

- If the touch screen needs to be recalibrated, choose **Start>All Programs>Keysight Logic Analyzer>Utilities>Touch Screen>Calibrate**.

16850-Series Front Panel Operation

The front panel interface consists of a knob and buttons that you use to set up and run measurements. There are also shortcut buttons that quickly access commonly used dialogs in the interface. When a front panel action is not valid, an audible "beep" will sound.

NOTE

When multiple instances of the *Keysight Logic and Protocol Analyzer* application are running on the logic analyzer, the front panel knobs and buttons only work for the application that is connected to the *local* acquisition hardware.

All functions available with the front panel knobs and buttons can also be performed in the graphical user interface (GUI).

- Run/Stop Buttons
- General Purpose Knob
- Touch Off Button

Run/Stop Buttons



Item	Description
Run Single	Runs a single acquisition (see page 196). The Run Single button turns green indicating when a Run action is valid. While the analyzer is running, the light goes out.
Run Rep. (Repetitive)	Runs a repetitive acquisition (see page 196). The Run Repetitive button turns green indicating when a Run Repetitive action is valid. While the analyzer is running, the light goes out.
Stop	Stops (see page 196) the current acquisition. The Stop button turns red during a Run cycle indicating when the Stop action is valid.

General Purpose Knob



The general purpose knob acts on the field that has the current focus. Fields that have the current focus have the blue background. The general purpose knob is typically used to increase/decrease numeric values such as waveform scale and delay.

Touch Off Button



Item	Description
Touch Off	Turns off the touch screen so that accidental touches don't affect the instrument.

See Also • [Tips for Using the Touch Screen \(see page 525\)](#)

16860-Series Logic Analyzer Product Overview

The 16860-series logic analyzers are standalone benchtop state and timing logic analyzers that range from 34 to 136 logic acquisition channels, depending on the model.

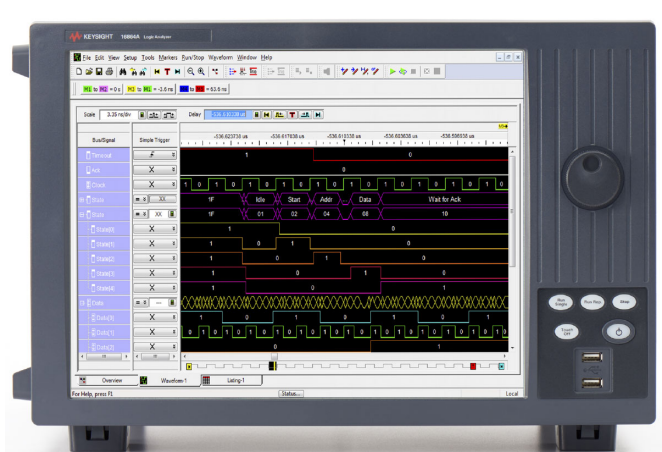
The 16860-series models provide features such as:

- deeper memory
- ¼ channel 10GHz Timing mode
- deskew of timing traces by individual channels
- single as well as multiple clocks support
- advanced clocking capabilities
- clock hysteresis
- various clock modes such as Master, Dual Sample, Master/Slave, and Demultiplex.

This series of logic analyzers can be used:

- at a higher speed (700MHz to 12.5 MSps) for debug, validation, and analysis of DDR and LPDDR memory systems.
- at a lower speed (350MHz to 0 MSps) as a medium to high performance general purpose logic analyzer.

These logic analyzers allow the maximum DQ visibility with simultaneous Read and Write traffic capture for DDR3 and LPDDR3 without overloading the system under test.



You can configure, control, and use the 16860-series Logic Analyzers through the Keysight Logic and Protocol Analyzer application (version 6.3 and above).

To know more about how to set up, use, and update your 16860 series logic analyzer, refer to the 16860 Series Portable Logic Analyzers Installation/Quick Start Guide shipped with your logic analyzer.

- Features, Logic Acquisition
- 2 M to 400 M memory depth per channel (depending on memory option), software upgradeable.
 - 350 MHz or 700 MHz maximum state data rate (depending on state speed option), software upgradeable.
 - Full Channel Timing Mode at 2.5 GHz sampling with 12.5 GHz Timing Zoom.
 - Half Channel Timing Mode at 5.0 GHz sampling with 12.5 GHz Timing Zoom.
 - Automated threshold and sample position setup.
 - 12.5 GHz timing zoom with 256 K samples.

- Features, Mainframe
- Built-in 15 inch TFT color LCD display with 1,024 x 768 (XGA) resolution. Touch screen with Option 103. See Tips for Using the Touch Screen.
 - Front panel knob and buttons. See 16860 Series Front Panel Operation.
 - 500 GB hard disk drive (or external hard drive option).
 - 10Base-T, 100Base-T, 1000Base-T LAN port.
 - USB 2.0 ports (six total, two on front, four on back).
 - One PCI expansion slot.
 - One PCI Express x1 expansion slot.
 - Windows 7 operating system.
 - Keysight Logic and Protocol Analyzer application which takes the complexity out of making logic analyzer measurements. You can perform all operations directly from one window.

- Supplied Accessories
- PS/2 mouse
 - PS/2 mini keyboard
 - Accessory pouch
 - Power cord

- Optional Accessories
- Interposers, Probes, and Cables

Refer to the *16860-Series Logic Analyzer - Probing Selection Card (16860-97002)* which is shipped with your logic analyzer or refer to its electronic version on www.keysight.com to get a list of supported interposers, probes, and cables for 16860-series. In addition, details of these interposers, probes, and cables are in their respective user guides available on www.keysight.com.

- See Also
- Tutorial - Getting to know your logic analyzer (see [page 40](#))
 - 16860-Series Logic Analyzer Specifications and Characteristics (see [page 600](#))

Tips for Using the Touch Screen

The 16860-series logic analyzer frames have a touch screen. Here are some tips for using the touch screen:

Use firm, even pressure on the touch screen.

You may prefer to use a stylus.

Use the front panel marker knobs to place markers. Placing markers is hard to do accurately using the touch screen. (Any marker can be selected using the Marker Choose button.)

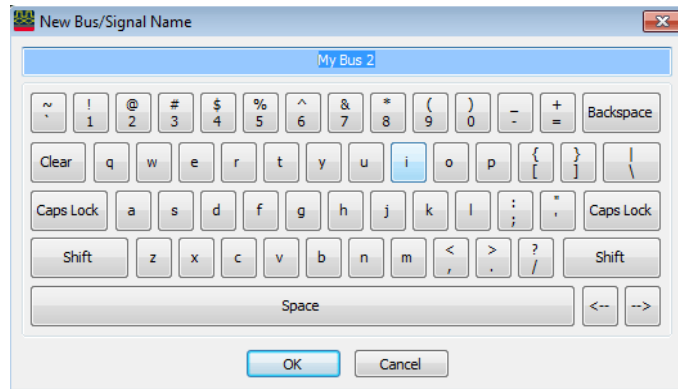
For trees, the touchable area around the +/- buttons is expanded.

For option selections, both the option and the caption are active.

To open a keyboard dialog

Inside the *Keysight Logic Analyzer* application:

- Press the  keyboard button in any edit field. This opens a dialog for entering field values.



Outside the *Keysight Logic Analyzer* application:

- Press the front panel **Keyboard** button. This opens the Microsoft On-Screen Keyboard and the touch screen Event Selector.



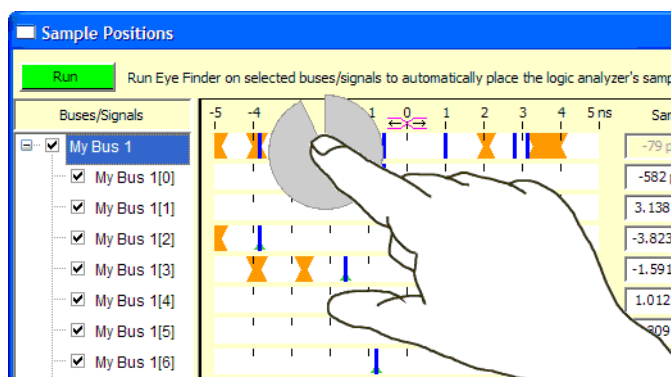
(You can also choose **Start>All Programs>Accessories>Accessibility>On-Screen Keyboard**.)

To access right mouse button behavior

Inside the *Keysight Logic Analyzer* application, most right mouse button behavior is accessible just by touching the screen.

Outside the *Keysight Logic Analyzer* application:

- Press down on the touch screen until a full circle is drawn around your finger; then, a right-click occurs.



- Or, you can choose **Start>All Programs>Keysight Logic Analyzer>Utilities>Touch Screen>Event Selector** to open the window:



Touching inside this window causes your next touch to act as a right-click.

To recalibrate the touch screen

- If the touch screen needs to be recalibrated, choose **Start>All Programs>Keysight Logic Analyzer>Utilities>Touch Screen>Calibrate**.

16860-Series Front Panel Operation

The front panel interface consists of a knob and buttons that you use to set up and run measurements. There are also shortcut buttons that quickly access commonly used dialogs in the interface. When a front panel action is not valid, an audible "beep" will sound.

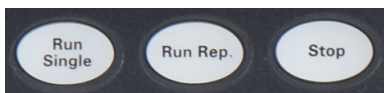
NOTE

When multiple instances of the *Keysight Logic and Protocol Analyzer* application are running on the logic analyzer, the front panel knobs and buttons only work for the application that is connected to the *local* acquisition hardware.

All functions available with the front panel knobs and buttons can also be performed in the graphical user interface (GUI).

- Run/Stop Buttons
- General Purpose Knob
- Touch Off Button

Run/Stop Buttons



Item	Description
Run Single	Runs a single acquisition (see page 196). The Run Single button turns green indicating when a Run action is valid. While the analyzer is running, the light goes out.
Run Rep. (Repetitive)	Runs a repetitive acquisition (see page 196). The Run Repetitive button turns green indicating when a Run Repetitive action is valid. While the analyzer is running, the light goes out.
Stop	Stops (see page 196) the current acquisition. The Stop button turns red during a Run cycle indicating when the Stop action is valid.

General Purpose Knob



The general purpose knob acts on the field that has the current focus. Fields that have the current focus have the blue background. The general purpose knob is typically used to increase/decrease numeric values such as waveform scale and delay.

Touch Off Button



Item	Description
Touch Off	Turns off the touch screen so that accidental touches don't affect the instrument.

See Also · [Tips for Using the Touch Screen \(see page 525\)](#)

Keysight Logic and Protocol Analyzer Application Product Overview

The *Keysight Logic Analyzer* application is a familiar Windows-based user interface which takes the complexity out of making logic analyzer measurements. You can perform all operations directly from one window.

Keysight's Simple, Quick and Advanced Trigger functions take the complexity out of triggering. Use Simple Trigger's pull down menus to define events in terms of edges and patterns. With Quick Trigger you can see if a suspect event ever reoccurs by just drawing a box around the event in the display. Quick trigger will do the rest! Use Advanced Trigger's drag and drop graphical icons with sentence-like structures to customize complex trigger scenarios.

- [Keyboard Commands \(see page 532\)](#)

Keyboard Commands

- [Access Menus \(see page 533\)](#)
- [File Operations \(see page 533\)](#)
- [Edit Operations \(see page 533\)](#)
- [Search Operations \(see page 533\)](#)
- [View operations \(see page 534\)](#)
- [Run/Stop Operations \(see page 534\)](#)

- Compare Operations (see [page 534](#))
- Listing Operations (see [page 534](#))
- Waveform Operations (see [page 535](#))
- Window Operations (see [page 535](#))
- Help Operations (see [page 535](#))
- Miscellaneous (see [page 535](#))

Access Menus

Menu	Description
Alt+F	Access to File menu
Alt+E	Access to Edit menu
Alt+V	Access to View menu
Alt+S	Access to Setup menu
Alt+T	Access to Tools menu
Alt+M	Access to Markers menu
Alt+R	Access to Run/Stop menu
Alt+W	Access to Window menu
Alt+H	Access to Help menu

File Operations

The following operations are located under **File** in the menu bar.

Menu	Description
Ctrl+N	File - New
Ctrl+O	File - Open
Ctrl+F4	File - Close
Ctrl+S	File - Save
Shift+E	File - Export
Ctrl+P	File - Print

Edit Operations

The following operations are located under **Edit** in the menu bar.

Menu	Description
Ctrl+Z	Edit - Undo
Ctrl+X	Edit - Cut
Ctrl+C	Edit - Copy
Ctrl+V	Edit - Paste
Alt+I	Edit - Insert Bus/Signal into Window
Alt+P	Edit - Current Window Properties

Search Operations

The following operations are located under **Edit** in the menu bar.

Menu	Description
Ctrl+F	Edit - Find
Shift+F3	Edit - Find Previous
F3	Edit - Find Next
Ctrl+B	Edit - Go To Beginning
Ctrl+T	Edit - Go To Trigger
Ctrl+E	Edit - Go To End
Ctrl+G	Edit - Go To

View Operations The following operations are located under **View** in the menu bar.

Menu	Description
Shift+F	View - Zoom Out Full
Shift+O	View - Zoom Out
Shift+I	View - Zoom In
F9	View - Full Screen
F11	View - Toggle Tabbed Windows
F12	View - Toggle Status Bar

Run/Stop Operations The following operations are located under **Run/Stop** in the menu bar.

Menu	Description
F5	Run/Stop - Run
Ctrl+F5	Run/Stop - Run Repetitive
F8	Run/Stop - Stop
Shift+F8	Run/Stop - Cancel
Shift+Ctrl+F8	Run/Stop - Resume

Compare Operations The following operations are located under **Compare** in the menu bar.

Menu	Description
Alt+P	Compare - Properties

Listing Operations The following operations are located under **Listing** in the menu bar.

Menu	Description
Alt+P	Listing - Properties

Waveform
Operations

The following operations are located under **Waveform** in the menu bar.

Menu	Description
Alt+P	Waveform - Properties

Window
Operations

The following operations are located under **Window** in the menu bar.

Menu	Description
F6	Window - Toggle to Next
Shift+F6	Window - Toggle to Previous

Help Operations

The following operations are located under **Help** in the menu bar.

Menu	Description
F1	Help - Help Topics

Miscellaneous

The following operations are located throughout the interface.

Menu	Description
Ctrl+esc	Shows Windows Start bar

Logic Analyzer Notes

- Channels and Memory Depth (see [page 536](#))
 - Timing Mode Sampling Options (see [page 536](#))
 - State Mode Sampling Options (see [page 536](#))
 - Timing Zoom (see [page 536](#))
- Channels and Memory Depth
- U4164A Logic Analyzer Notes, Channels and Memory Depth (see [page 537](#))
 - U4154A Logic Analyzer Notes, Channels and Memory Depth (see [page 545](#))
 - U4154B Logic Analyzer Notes, Channels and Memory Depth (see [page 549](#))
 - 16850-Series Logic Analyzer Notes, Channels and Memory Depth (see [page 553](#))
 - 16860-Series Logic Analyzer Notes, Channels and Memory Depth (see [page 555](#))
- Timing Mode Sampling Options/Period
- The timing mode sampling options let you choose between *Full Channel Timing Mode* (default), *Half Channel Timing Mode* (faster sampling), or *Transitional / Store Qualified Timing Mode* (greater measurement length). For notes on these modes in a particular logic analyzer, see:
- U4164A Logic Analyzer Notes, Timing Mode Sampling Options/Period (see [page 538](#))
 - U4154A Logic Analyzer Notes, Timing Mode Sampling Options/Period (see [page 545](#))
 - U4154B Logic Analyzer Notes, Timing Mode Sampling Options/Period (see [page 545](#))
 - 16850-Series Logic Analyzer Notes, Timing Mode Sampling Options/Period (see [page 553](#))
 - 16860-Series Logic Analyzer Notes, Timing Mode Sampling Options/Period (see [page 555](#))
- State Mode Sampling Options
- The state mode sampling options let you choose between Single Clock or Multiple Clocks options. For notes on these options in a particular logic analyzer, see:
- U4164A Logic Analyzer Notes, State Mode Sampling Options (see [page 540](#))
 - U4154A Logic Analyzer Notes, State Mode Sampling Options (see [page 546](#))
 - U4154B Logic Analyzer Notes, State Mode Sampling Options (see [page 546](#))
 - 16850-Series Logic Analyzer Notes, State Mode Sampling Options (see [page 553](#))
 - 16860-Series Logic Analyzer Notes, State Mode Sampling Options (see [page 555](#))
- Timing Zoom
- U4164A Logic Analyzer Notes, Timing Zoom (see [page 542](#))
 - U4154A Logic Analyzer Notes, Timing Zoom (see [page 546](#))
 - U4154B Logic Analyzer Notes, Timing Zoom (see [page 546](#))
 - 16850-Series Logic Analyzer Notes, Timing Zoom (see [page 553](#))
 - 16860-Series Logic Analyzer Notes, Timing Zoom (see [page 555](#))
- U4164A Logic Analyzer Notes
- Differences from Other Logic Analyzers (see [page 536](#))
 - Channels and Memory Depth (see [page 537](#))
 - Timing Mode Sampling Options/Period (see [page 538](#))
 - State Mode Sampling (see [page 540](#))
 - Timing Zoom (see [page 542](#))
- Differences from Other Logic Analyzers
- Compared to other Keysight logic analyzers supported in earlier releases, the U4164A logic analyzer differs in the following ways:
- Supports two state sampling options - Single Clock and Multiple Clocks.

- The Single Clock option provides a single master clock and supports the state mode sampling speed ranging from 12.5 Mega samples per second to 2.5 GHz (with "02G" state speed license option).
- The Multiple Clocks mode provides up to four clocks and is a low speed option more suitable for general-purpose logic analysis with state mode sampling speed ranging from 0 Mega samples per second to 700 MHz.

To know more about these two state sampling options, refer to the topic [Selecting the State Sampling Option for a U4164A Logic Analyzer](#).

- You can connect three U4164A modules to form a three-module set in an AXIe chassis. U4164A modules cannot merge into module sets with U4154A or U4154B modules.
- In a U4164A setup, you can load a .xml Logic Analyzer configuration file that you created/previously saved in a U4154A or U4154B setup. You cannot, however load a Standard configuration (.ala) file that you created/previously saved in a U4154A or U4154B setup.
- U4164A supports four state speeds. The maximum state speed supported is 4 Gb/s. The "02G" license sets it to 4 Gb/s (2.5 GHz). These license options do not however control and set the timing speed of the U4164A module.
- Supports Master, Master/Slave/Demux, Dual Sample, and Quad Sample clock modes. The availability of these clock modes depend on the state sampling option (Single Clock or Multiple clocks) that you have selected for your logic analyzer.
- Supports high speed data capture with a new State sampling clock mode - Quad Sample that allows four samples per clock edge. It allows you to capture separate rising and falling edge samples of both read and write DDR/LPDDR DQ signals for data rates up to 4 Gb/s from a single probe point. 02G license of U4164A is required for this clock mode. Refer to the topic ["To set up the quad sample sampling clock mode"](#) on page 119 to know more.
- Supports the Dual Sample clock mode with two thresholds that can be assigned as separate offsets to capture two samples per clock edge with different thresholds. Refer to the topic ["To set up the dual sample sampling clock mode"](#) on page 116mode to know more about how dual sampling functions in the U4164A module.
- Supports a new Timing (Asynchronous) sampling option - Quarter Channel Timing Mode. 01G or 02G license of U4164A is required for this timing mode.
- Allows you to set the sampling positions for all Timing (Asynchronous) sampling options using a new tool named the Signal Deskew tool.
- U4164A provides a new setting called Clock hysteresis for the state sampling clock on Pod1 of the U4164A module. Using this setting, you can set a value (in milliVolts) between 0 and 1 volts around the clock threshold. This allows you to avoid false sampling on noisy clock inputs where a differential clock input turns off and floats to zero volts. Refer to the topic [Setting Clock Hysteresis for the U4164A State Sampling Clock](#) to know more.

Channels and Memory Depth

136 channels per U4164A module (This includes 128 data channels and 8 clock channels)

You can install and connect a maximum of three U4164A modules in an AXIe chassis to form a 3-card set with 408 channels. Refer to the table below for the number of channels available for each of the supported sampling modes.

U4164A Sampling Modes	For State (Synchronous) Sampling Modes			For Timing (Asynchronous) Sampling Modes		
	Conventional State Sampling	Dual Sample State Sampling	Quad Sample State Sampling	Full Channel	Half Channel	Quarter Channel
Number of Channels Available in one U4164A module	136	68	34	136	68	34
Number of Channels Available in a two U4164A modules set	272	136	68	272	136	68
Number of Channels Available in a three U4164A modules set	408	204	102	408	204	102

U4164A Memory depth	For State (Synchronous) Sampling Modes	For Timing (Asynchronous) Sampling Modes		
		Full Channel	Half Channel	Quarter Channel
Standard memory	2 M	2 M	4 M	8 M
Option 004	4 M	4 M	8 M	16 M
Option 008	8 M	8 M	16 M	32 M
Option 016	16 M	16 M	32 M	64 M
Option 032	32 M	32 M	64 M	128 M
Option 064	64 M	64 M	128 M	200 M
Option 128	128 M	128 M	200 M	400 M
Option 200	200 M	200 M	400 M	800 M
Option 400	400 M	400 M	800 M	1.6 G

Timing Mode
Sampling
Options/Period

Timing Mode Sampling Option	Description																																																																																																																																																																																								
Full channel, 2.5 GHz (Full Channel Timing Mode)	<p>Maximum acquisition rate supported in this option - 2.5 GHz.</p> <p>With this timing mode sampling option:</p> <ul style="list-style-type: none"> You can use the full memory depth of your U4164A logic analyzer module. Data sampling period ranging from 400 ps to 10 ns. You can set the sample period using the Sample Period field. All the eight pods of U4164A are available. Eight clock channels are available per U4164A module. 																																																																																																																																																																																								
Half channel, 5.0 GHz (Half Channel Timing Mode)	<p>Maximum acquisition rate supported in this option - 5.0 GHz.</p> <p>With this timing mode sampling option:</p> <ul style="list-style-type: none"> The memory depth is doubled the available depth (refer to the table above). Data is sampled and stored every 200 ps; this rate cannot be changed. Only odd pod of each pod pair is available that is, Pod 1, 3, 5, and 7. Channels assigned to the unavailable even pods are ignored. Four clock channels are available per U4164A module. 																																																																																																																																																																																								
	<table border="1"> <thead> <tr> <th colspan="8">Slot A Pod 7</th> <th colspan="8">Slot A Pod 5</th> <th colspan="8">Slot A Pod 3</th> <th colspan="8">Slot A Pod 1</th> </tr> <tr> <th colspan="8">Threshold: HSTL</th> <th colspan="8">Threshold: HSTL</th> <th colspan="8">Threshold: HSTL</th> <th colspan="8">Threshold: HSTL</th> </tr> </thead> <tbody> <tr> <td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td> <td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td> <td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td> <td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td> </tr> <tr> <td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td> <td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td> <td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td> <td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td> </tr> </tbody> </table>	Slot A Pod 7								Slot A Pod 5								Slot A Pod 3								Slot A Pod 1								Threshold: HSTL								Threshold: HSTL								Threshold: HSTL								Threshold: HSTL								1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0
Slot A Pod 7								Slot A Pod 5								Slot A Pod 3								Slot A Pod 1																																																																																																																																																																	
Threshold: HSTL								Threshold: HSTL								Threshold: HSTL								Threshold: HSTL																																																																																																																																																																	
1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0																																																																																																																														
1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0	1	1	1	1	1	9	8	7	6	5	4	3	2	1	0																																																																																																																														
Quarter channel, 10 GHz (Quarter Channel Timing Mode)	<p>Maximum acquisition rate supported in this option - 10.0 GHz.</p> <p>This timing mode sampling option is available only if you have installed the 01G or 02G Speed license option of the U4164A module.</p> <p>With this timing sampling option:</p> <ul style="list-style-type: none"> The memory depth is four times the available depth (refer to the table above). Data is sampled and stored every 100 ps; this rate cannot be changed. Only odd pod of each pod pair is available that is, Pod 1, 3, 5, and 7. Only even channels (0, 2, 4, 6, 8, 10, 12, 14) for each odd pod are available. Two clock channels are available per U4164A module. 																																																																																																																																																																																								
	<table border="1"> <thead> <tr> <th>Clo</th> <th colspan="8">Slot A Pod 7</th> <th colspan="8">Slot A Pod 5</th> <th colspan="8">Slot A Pod 3</th> <th colspan="8">Slot A Pod 1</th> </tr> <tr> <th>Clo</th> <th colspan="8">Threshold: HSTL</th> <th colspan="8">Threshold: HSTL</th> <th colspan="8">Threshold: HSTL</th> <th colspan="8">Threshold: HSTL</th> </tr> </thead> <tbody> <tr> <td>16</td> <td>14</td><td>12</td><td>10</td><td>8</td><td>6</td><td>4</td><td>2</td><td>0</td> <td>14</td><td>12</td><td>10</td><td>8</td><td>6</td><td>4</td><td>2</td><td>0</td> <td>14</td><td>12</td><td>10</td><td>8</td><td>6</td><td>4</td><td>2</td><td>0</td> <td>14</td><td>12</td><td>10</td><td>8</td><td>6</td><td>4</td><td>2</td><td>0</td> </tr> <tr> <td>15</td> <td>14</td><td>12</td><td>10</td><td>8</td><td>6</td><td>4</td><td>2</td><td>0</td> <td>14</td><td>12</td><td>10</td><td>8</td><td>6</td><td>4</td><td>2</td><td>0</td> <td>14</td><td>12</td><td>10</td><td>8</td><td>6</td><td>4</td><td>2</td><td>0</td> <td>14</td><td>12</td><td>10</td><td>8</td><td>6</td><td>4</td><td>2</td><td>0</td> </tr> </tbody> </table>	Clo	Slot A Pod 7								Slot A Pod 5								Slot A Pod 3								Slot A Pod 1								Clo	Threshold: HSTL								Threshold: HSTL								Threshold: HSTL								Threshold: HSTL								16	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0	15	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0																																																				
Clo	Slot A Pod 7								Slot A Pod 5								Slot A Pod 3								Slot A Pod 1																																																																																																																																																																
Clo	Threshold: HSTL								Threshold: HSTL								Threshold: HSTL								Threshold: HSTL																																																																																																																																																																
16	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0																																																																																																																																																									
15	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0	14	12	10	8	6	4	2	0																																																																																																																																																									
Transitional / Store Qualified Timing Modes	<p>This timing mode sampling option provides maximum duration of acquisition because data is only stored when a change from the last value is detected. Data is sampled at regular intervals but stored only when there is a signal transition across the threshold voltage level. See transitional timing. Three sampling options are available in this mode:</p> <ul style="list-style-type: none"> Transitional / Store qualified, (Full channel mode), 2.5 GHz Transitional / Store qualified, (Half channel mode), 5.0 GHz Transitional / Store qualified, (Quarter channel mode), 10.0 GHz - Available only when you have installed 01G or 02G license of the U4164A module. 																																																																																																																																																																																								

NOTE

For the U4164A module, you can set the sampling positions for the Timing mode sampling options as well. You use the Signal Deskew Tool to accomplish this. Refer to the online help available with this tool's GUI to know more.

State Mode Sampling Option

State Sampling Option	Description
Single Clock	Faster state mode sampling speeds are supported, but some triggering features, multiple clocks, and advanced clocking are not available. Refer to the table below to know about the maximum acquisition rate available in the Single Clock sampling option.
Multiple Clocks	Advanced clocking capabilities and multiple clocks are supported, but at a slower state sampling speed. Refer to the table below to know about the maximum acquisition rate available in the Multiple Clocks sampling option.

Refer to the topic [Selecting the State Sampling Option for a U4164A Logic Analyzer](#) to get a comparison between these two sampling options and to know how these options are used in the context of a U4164A Logic Analyzer.

The U4164A logic analyzer supports the following four state speeds. Based on the state speed license option that you have purchased for your logic analyzer, the maximum available state speed of your logic analyzer is automatically set.

U4164A State Speed License Option	Maximum Acquisition Rate	
	In Single Clock State Sampling Option	In Multiple Clock State Sampling Option
Standard (Default) State Speed 350 MHz (700 Mb/s)	700 Mb/s on 136 channels per U4164A, clocking on both edges of the clock 350 Mb/s on 136 channels per U4164A, clocking on either edge of the clock 700 Mb/s on 68 channels per U4164A, clocking on either or both edges of the clock	Captures a 350MHz signal on rising, falling, or both edges of the clock up to 350 MSP/s
Option 700 700 MHz (1.4 Gb/s)	1.4 Gb/s on 136 channels per U4164A, clocking on both edges of the clock 700 Mb/s on 136 channels per U4164A, clocking on either edge of the clock 1.4 Gb/s on 68 channels per U4164A, clocking on either or both edges of the clock	Captures a 700MHz signal on rising, falling, or both edges of the clock up to 700 MSP/s
Option 01G 2.8 Gb/s (1.4 GHz)	2.5 Gb/s on 136 channels per U4164A, clocking on both edges of clock 1.4 Gb/s on 136 channels per U4164A, clocking on either edge of the clock 2.8 Gb/s on 68 channels per U4164A, clocking on either or both edges of the clock	Not allowed
Option 02G 4 Gb/s (2.5 GHz)	2.5 Gb/s on 136 channels per U4164A, clocking on either or both edges of clock 4.0 Gb/s on 68 channels per U4164A, clocking on either edge of the clock	Not allowed

State Sampling Clock Modes

The state sampling clock mode specifies how the clock inputs are used for sampling. The clock modes available for a U4164A logic analyzer depends on the state sampling option (Single clock or Multiple Clocks) that you selected.

Clock Modes Available for the Single Clock Sampling Option	Clock Modes Available for the Multiple Clocks Sampling Option
Master Dual Sample Quad Sample	Master Master/Slave/Demux
For a detailed description of these clock modes, refer to the topics: <ul style="list-style-type: none"> · Master · Dual Sample · Quad Sample · Master/Slave · Demux 	
For information on how to set up these clock modes, see: <ul style="list-style-type: none"> · To set up the master only sampling clock mode · To set up the dual sample sampling clock mode · To set up the quad sample sampling clock mode · To set up the master/slave sampling clock mode · To set up the demultiplex sampling clock mode 	

State Sampling Clock(s)

The U4164A logic analyzers support both Single and Multiple clocks sampling options. Therefore, you can set up a single or multiple state sampling clocks.

- In case of a single clock, the state sampling clock input is always from Pod1 of the logic analyzer. In a multi-module set, the clock is allowed only on Pod1 of the master card (clocking module). The master card is the middle U4164A module in a three-module set and the bottom U4164A module in a two-module set.
- In case of multiple clocks, the state sampling clock input can be from multiple Pods (Pod 1, 2, 3, or 4) of the logic analyzer in an Ored combination.

To know how to set up a single or multiple clocks, refer to the topic [To set up the state sampling clock](#).

When setting up multiple clocks, you can also use the Advanced Clocking feature to define complex clock descriptions.

State Sampling Clock Qualifiers

For a Single Clock

The clock qualifiers can take clock signal inputs from the following pods of logic analyzer. You cannot use the signals from the other pods clock inputs as the sampling clock qualifiers.

Clock qualifier Pod input	Usage
Pod 2	Can be used as AND or OR clock qualifier
Pod 3	Can be used as AND or OR clock qualifier
Pod 4	Can be used as AND or OR clock qualifier
Pod 5	Can be used as AND or OR clock qualifier
Pod 7	The RESET clock qualifier input on pod 7 is available as an AND input only when other clock qualifiers are set up as OR inputs. One situation in which this clock qualifier is particularly useful is to capture RESET when the other clock qualifiers are looking for CKE. This qualifier allows you to create OR qualification of upto 4 DDR memory CKE clock qualifiers along with an AND qualified with RESET.

The following are some points to remember when using the clock qualifier on Pod 7:

- This qualifier is only available when other clock qualifier(s) are used as OR qualifiers.

- This qualifier is only an AND qualifier. The Clock Description equation therefore, always adds the Ck7 as an AND condition. For example: $Ck1 \wedge (Ck4=0 \vee Ck5=1) \wedge Ck7=1$
- Setup and hold of Pod 7 qualifier will be larger than other clock qualifiers.

For multiple clocks

- The clock qualifiers can take clock signal inputs from the following pods of logic analyzer. You cannot use the signals from the other pods clock inputs as the sampling clock qualifiers.

Clock Qualifiers in Multiple Clocks

- Up to two clock qualifiers (Pod 1, 2, 3, 4)
- Though three clock qualifiers are displayed for use, you can set up to 2 clock qualifiers for a clock.

Determining Optimal Sample Positions and Thresholds

In the state sampling acquisition mode, you need to adjust the sampling positions on U4164A channels relative to the sampling clock to make sure data is sampled when it is valid.

You can use the eye scan feature of the U4164A Logic Analyzer to automatically determine and set the optimal sample positions and thresholds for the individual signals. Clicking the Eye scan: Sample Positions and Threshold button in the Sampling tab displays the Eye Scan - Sample Positions and Threshold Settings dialog box where you can set up and run the eye scan measurement for U4164A channels. Refer to the topic Setting up and Running Eyescans in Logic Analyzer to learn more.

Timing Zoom

Timing zoom collects additional high-speed timing data around the trigger of the logic analyzer. It uses a 256+K-depth, 12.5 GHz timing analyzer to sample data every 80 ps on all channels. Timing zoom is available for all acquisition modes.

Setting up the State Sampling Options for a U4164A Logic Analyzer

Pod:	Pod 1.7	Pod 1.5	Pod 1.4	Pod 1.3	Pod 1.2	Pod 1.1	Clock Description
Clock:	Clk7	Clk5	Clk4	Clk3	Clk2	Clk1	
Activity:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Master:	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Clk1↑

The state sampling option that you select specifies the speed up to which the state mode sampling clock will match input clock edges from the device under test.

The following table describes and compares the state sampling options that you can set for a U4164A logic analyzer.

State Sampling Options	Single Clock, Full Channel, 2.5GHz to 12.5 MSps	Multiple Clocks, Full channel, 700 MSps to 0 MSps
Usage	<p>Default Option</p> <p>Faster state mode sampling speeds are supported, but some triggering and advanced clocking features are not available. More suitable for DDR/LPDDR data captures with high data rates.</p>	<p>Supports slower state sampling speeds but allows advanced clocking capabilities and multiple clocks.</p> <p>More suitable for general-purpose logic analysis.</p>
State Sampling Speeds Supported	<p>2.5 GHz to 12.5 MHz (with state speed option 02G)</p> <p>1.4 GHz to 12.5 MHz (with state speed option 01G)</p> <p>700 MHz to 12.5 MHz (with state speed option 700)</p> <p>350 MHz to 12.5 MHz (with standard state speed)</p> <p>Note: An error message is displayed if the input clock edges from the DUT are outside the lowest and highest speed range allowed in this option. In such a situation, measurements are not collected.</p>	<p>700 MSps to 0 MSps</p> <p>(This sampling speed will not change with state speed license options of U4164A)</p> <p>Note: A warning message is displayed if the input clock edges from the DUT are faster than the highest speed allowed in this option. In such a situation, measurements are still available but there is a risk of some missing data. You can use the Single Clock option in such a situation.</p>
Maximum Acquisition Rate	<p>At the 2.5 GHz state sampling speed:</p> <p>2.5 Gb/s on 136 channels per U4164A, clocking on either or both edges of clock</p> <p>4.0 Gb/s on 68 channels per U4164A, clocking on either edge of the clock</p> <p>At the 1.4 GHz state sampling speed:</p> <p>2.5 Gb/s on 136 channels per U4164A, clocking on both edges of clock</p> <p>1.4 Gb/s on 136 channels per U4164A, clocking on either edge of the clock</p> <p>2.8 Gb/s on 68 channels per U4164A, clocking on either or both edges of the clock</p> <p>At the 700 MHz state sampling speed:</p> <p>1.4 Gb/s on 136 channels per U4164A, clocking on both edges of the clock</p> <p>700 Mb/s on 136 channels per U4164A, clocking on either edge of the clock</p> <p>1.4 Gb/s on 68 channels per U4164A, clocking on either or both edges of the clock</p> <p>At the 350 MHz state sampling speed:</p> <p>700 Mb/s on 136 channels per U4164A, clocking on both edges of the clock</p> <p>350 Mb/s on 136 channels per U4164A, clocking on either edge of the clock</p> <p>700 Mb/s on 68 channels per U4164A, clocking on either or both edges of the clock</p>	<p>At the 350MHz state sampling speed:</p> <p>Captures a 350MHz signal on rising, falling, or both edges of the clock up to 350 MSp/s</p> <p>At the 700MHz state sampling speed:</p> <p>Captures a 700MHz signal on rising, falling, or both edges of the clock up to 700 MSp/s</p>
Number of Clocks Supported	Single clock (Always on Pod1 of the logic analyzer)	<p>Up to 4 clocks (in an Ored combination).</p> <p>Master and slave clocks are counted separately.</p> <p>Refer to the topic To set up the state sampling clock to know about setting multiple clocks.</p>
State Sampling Clock Modes Supported	<p>Master</p> <p>Dual Sample</p>	<p>Master</p> <p>Master/Slave</p> <p>Demux</p>
Advanced Clocking Capabilities	<p>Not supported</p> <p>A clock channel can be used either as an edge or as a qualifier but not both. The clock channel on Pod 1 can only be used as an edge.</p>	<p>Supported</p> <p>You can specify complex clock setups with the available clock channels used both as an edge and a qualifier in the same clock description.</p> <p>Refer to the topic Advanced Clocking Setup dialog to know more.</p>

State Sampling Options	Single Clock, Full Channel, 2.5GHz to 12.5 MSps	Multiple Clocks, Full channel, 700 MSps to 0 MSps
Clock Qualifiers	Maximum of five clock qualifiers (Pod 2, 3, 4, 5, 7)	Maximum of two clock qualifiers (Pod1, 2, 3, 4). Note: Though there are three clock qualifiers displayed, you can use a maximum of only two qualifiers for a clock. The third qualifier is automatically set to "Don't Care" and can be used only as an Ored clock in the clock description.
Triggering Capabilities Differences	Number of trigger sequences - 8 Burst patterns supported - Yes Number of timers - 1 Global counters supported - No Event counters supported - Yes	Number of trigger sequences - 16 Burst patterns supported - No Number of timers - 3 Global counters supported - 2 Event counters supported - No

The logic analysis features that are not included in this table work the same in Single clock and Multiple Clocks options.

- See Also
- To set up the state sampling clock
 - U4164A Logic Analyzer Notes - State Mode Sampling

U4154A Logic Analyzer Notes

- Differences from Other Logic Analyzers (see [page 544](#))
- Channels and Memory Depth (see [page 545](#))
- Timing Mode Sampling Options/Period (see [page 545](#))
- State Mode Sampling (see [page 546](#))
- Timing Zoom (see [page 546](#))

Differences from Other Logic Analyzers

Compared to other Keysight logic analyzers supported in earlier releases, the U4154A logic analyzer differs in the following ways:

- There is no quarter channel timing mode.
- U4154A supports two license based state speeds. A "01G" license sets the maximum state speed of U4154A to 1.4Gbs. A "02G" license sets it to 2.5Gbs. These license options do not however control and set the timing speed of the U4154A module.
- There is a single state sampling clock with four qualifiers. The clock qualifiers allow you to sample only when qualifying signal is active so that you can view more system activity. The state sampling clock input is always from Pod1 on the master card in a multi-card set. (The master card is the lower-middle card of a multi-card set). You cannot use the signals from the other pods clock inputs as the sampling clock inputs. For more information, refer to the topic "[Setting up the State Sampling Options in U4154A Logic Analyzer](#)" on page 547.
- Advanced probe settings are enabled for U4154A logic analyzers. You can enable or disable the peaking at the channel/pod/module level for the probing system used for these logic analyzers. For more information, refer to "[Changing Advanced Probe Settings for Logic Analyzers](#)" on page 563.
- Memory depth controls and licensing is different in U4154A Logic Analyzer compared to previous Keysight logic analyzers. Unlike previous logic analyzers, the density of data samples stored in U4154A memory varies with the acquisition mode. The amount of U4154A memory available is fixed based on the memory license that you have purchased. However, there is a variation in the maximum number of samples that you can store in the fixed memory based on the selected acquisition mode.
- You cannot logically split a U4154A logic analyzer module.
- The U4154A module is installed in one of the slots of the Keysight AXIe chassis. It connects to the host PC through the PCIe interface of the AXIe chassis. You can connect two U4154A modules to form a two-card set in an AXIe chassis.

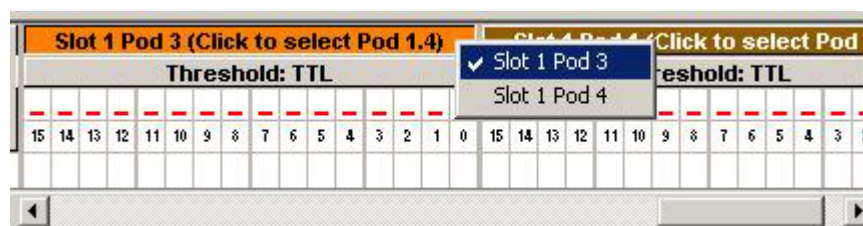
- Colorized Eyescan on all signals is supported to quickly set accurate sample positions and threshold voltages as per the optimal values suggested by the eye scan run. You can also export the eyescan data to a specified .csv file. For more information, refer to ["Setting up and Running Eyescans in Logic Analyzers"](#) on page 567.
- Trigger expansions with more levels and deeper bursts. Burst levels increased from 4 to 8. Faster trigger sequencer rate (with triggers on sequential events upto 2.5 Gb/s). New event counter added for specifying the trigger action.
- The DDR Setup Assistant tool has been enhanced to support the set up of the U4154A Logic Analyzer module using this tool. The automated setup steps now also include automatically determining and setting the optimal acquisition sample position at the center of the eye on individual channels of U4154A. You can now use this tool to automatically set the sampling positions for command, address, read and write data signals separately while setting up DDR measurement setup with U4154A logic analyzer. The eye scan feature of the U4154A Logic Analyzer is used to accomplish this. To learn more, refer to the DDR Setup Assistant online help integrated with this help.

Channels and Memory Depth

U4154A Memory depth	Option 002	Option 004	Option 008	Option 016	Option 032	Option 064	Option 128	Option 200
	2 M	4 M	8 M	16 M	32 M	64 M	128 M	200 M
Channels	136 channels per U4154A module (This includes 128 data channels and 8 clock channels) In a multi-card set: 136 channels * number of U4154A modules installed in slots of an Keysight AXIe chassis You can install and connect two U4154A modules in an AXIe chassis to form a 2-card set with 272 channels.							

Timing Mode Sampling Options/Period

- **Full channel 2.5 GHz = Full Channel Timing Mode**
With this timing mode sampling option, you can use the full memory depth of your U4154A logic analyzer module, with data sampling period ranging from 400 ps to 10 ns. You can set the sample period using the Sample Period field.
- **Half channel 5.0 GHz = Half Channel Timing Mode**
With this timing mode sampling option, only one pod of each pod pair is available, and the memory depth is doubled. Channels assigned to unavailable pods are ignored. You can specify which pod to use in the Buses/Signals tab of the Analyzer Setup dialog by clicking the Pod button and selecting the desired pod.

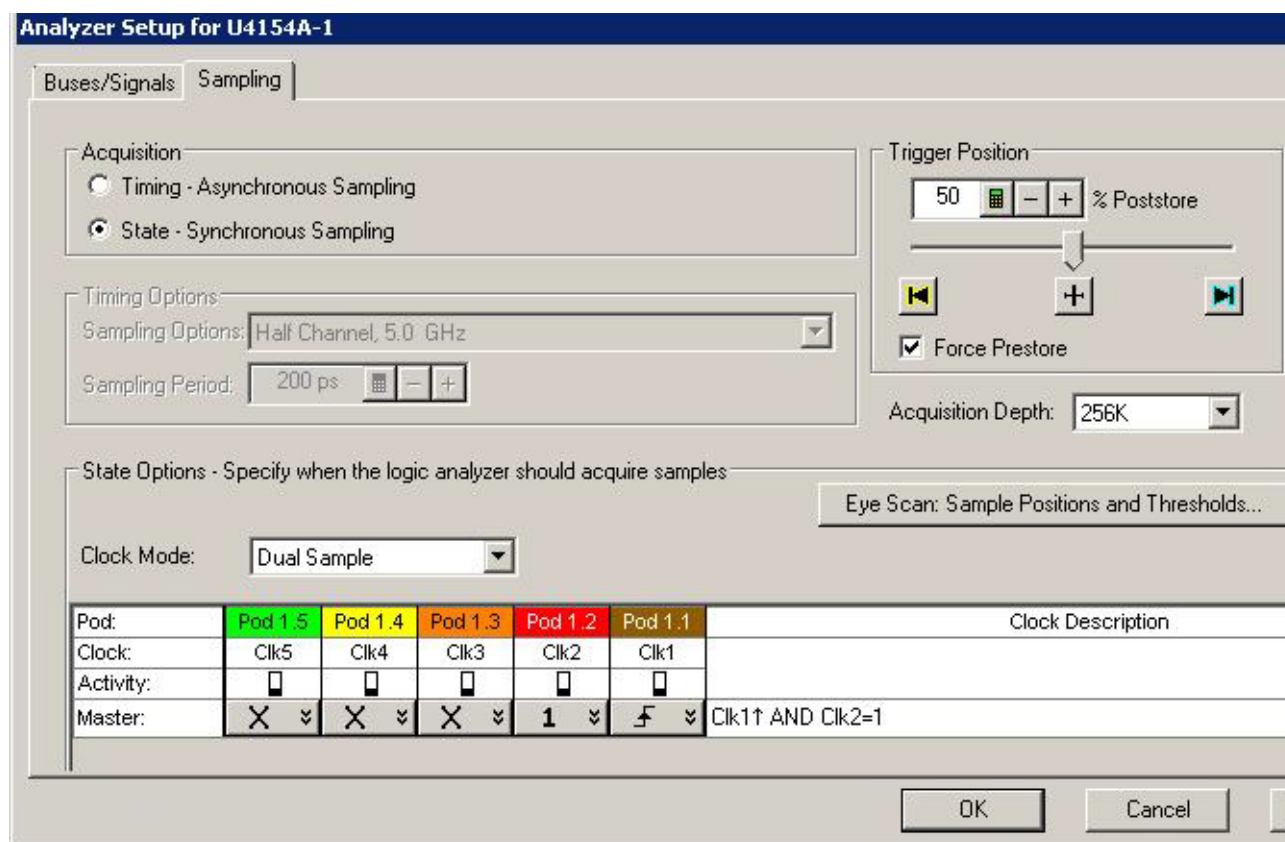


Data is sampled and stored every 200 ps; this rate cannot be changed.

- The Quarter channel timing mode is not supported.
- **Transitional / Store Qualified Timing Mode**
This timing mode sampling option provides maximum duration of acquisition because data is only stored when a change from the last value is detected. See transitional timing on [page 365](#). Two sampling options are available in this mode:
 - **Transitional / Store qualified, (Full channel mode), 2.5 GHz** - With this sampling option, you can use the full memory depth of your U4154A logic analyzer module, with data being sampled at regular intervals but stored only when there is a signal transition across the threshold voltage level. You can set the sampling period from 400 ps to 10 ns for this mode.

- **Transitional / Store qualified, (Half channel mode), 5.0 GHz** - With this sampling option, only one pod of each pod pair is available, and the memory depth is doubled. Channels assigned to unavailable pods are ignored. You can specify which pod to use in the Buses/Signals tab of the Analyzer Setup dialog by clicking the Pod button and selecting the desired pod. Data is sampled at regular intervals but stored only when there is a signal transition across the threshold voltage level. The sampling period is 200 ps for this mode and cannot be changed.
- State Mode Sampling
- U4154A has two license based state speeds. A "01G" license sets the maximum state speed of U4154A to 1.4Gbs. A "02G" license sets it to 2.5Gbs. Based on the license option that you have purchased for U4154A, the maximum available state speed of U4154A is automatically set. If the licensed speed is 1.4Gbs (01G option) and while acquiring data, the clock frequency is found to exceed the maximum state speed available as per the 01G license, an error message is displayed and the data is discarded.
 - There is a single state sampling clock with four qualifiers. In a multi-card set, the sampling clock signals can come only from Pod 1 of the master card in the set. The qualifiers can take clock signal inputs only from the pods 2 to 5 on the master card in the set. The clock can be used as a rising, falling, or both edge clock.
 - U4154A supports Full channel (Master) and Dual sample clock modes. In both these modes, the sampling option is as per the state speed license acquired for U4154A.
- Timing Zoom
- Timing zoom collects additional high-speed timing data around the trigger of the logic analyzer. It uses a 256K-depth, 12.5 GHz timing analyzer to sample data every 80 ps on all channels. Timing zoom is available for all acquisition modes.
- See Also
- ["Setting up the State Sampling Options in U4154A Logic Analyzer"](#)
 - ["Changing Advanced Probe Settings for Logic Analyzers"](#)
 - ["Setting up and Running Eyescans in Logic Analyzers"](#)
 - ["U4154A Logic Analyzer Specifications and Characteristics"](#)
 - ["Pod and Channel Naming Conventions in Logic Analyzer"](#)

Setting up the State Sampling Options in U4154A Logic Analyzer



The U4154A state sampling speed matches your device under test's clock rate, from 100 Mb/s up to 2.5 Gb/s.

The state sampling clock inputs let signals from the device under test specify when data should be captured.

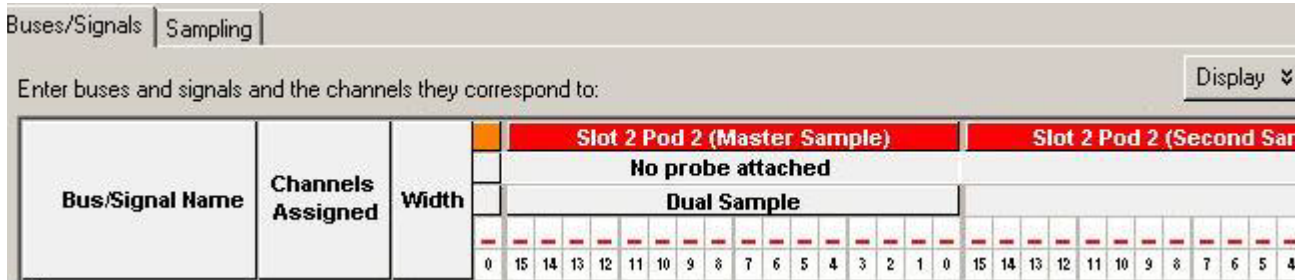
U4154A provides the following two license options for state mode speeds (2.5Gb/s and 1.4Gb/s) at which the state sampling clock matches input clock edges from the device under test.

- **2.5 Gb/s speed** - If you have the 02G license for U4154A module, then the maximum available state speed of U4154A is automatically set to 2.5Gb/s.
- **1.4 Gb/s speed** - If you have the 01G license for U4154A module, then the maximum available state speed of U4154A is automatically set to 1.4Gb/s. If, while acquiring data, the clock frequency is found to exceed the maximum state speed available as per the 01G license, an error message is displayed and the data is discarded.

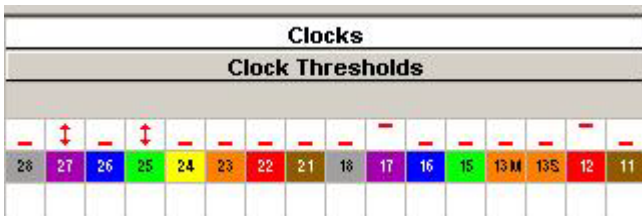
State Sampling Clock Mode

The state sampling clock mode specifies how the clock inputs are used for sampling. There are two state sampling clock modes to choose from:

- **Master** – all pods sampled on one master clock.
- **Dual Sample** – one pod in the pod pair sampled on one master clock but with different delays. When you enable this clock mode, the next step is to set up the pod pairs that you want to be dual sampled in the Buses/Signals tab of the Analyzer Setup dialog box (see ["To set up the dual sample sampling clock mode"](#) on page 116). Data on the active pod in a dual sampled pod pair is sampled twice (as master sample and second sample).



The pods for which you select the dual sample mode have their associated clock bit also dual sampled. The active clock bit of the pods in a dual sampled pod pair is the clock bit of the selected active pod in that pod pair. notice that in the following screen, the clock channels of the dual sample pod pair (pod 1.3 and pod 1.4) are also dual sampled. These dual sampled clocks are represented as 13 M or C3M and 13 S or C3S (C3 Master Sample and C3 Second Sample).



For instructions on setting up these state sampling clock modes, see:

- “To set up the master only sampling clock mode” on page 114
- “To set up the dual sample sampling clock mode” on page 116

State Sampling Clock and Qualifiers

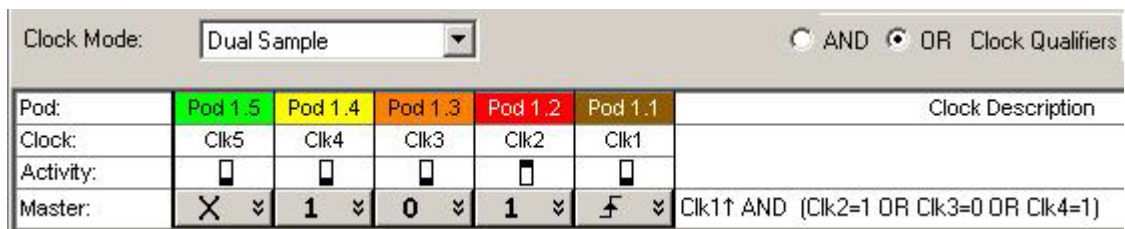
In U4154A, only one state sampling clock is provided. The state sampling clock is the clock of Pod 1 of the master card (the lower middle level card in the set). To know which card is the master card in the set, click **System Summary** button in the Buses/Signals tab of the Analyzer Setup dialog box.

You can choose the sampling to occur on rising, falling, or both edges of the input clock signal.

There are four qualifiers available for the state clock. The clock inputs of pods 2 to 5 are used as the state clock qualifiers. In a multi card set, the clock inputs of pods 2 to 5 of the master card in the set are used as the state clock qualifiers.

You cannot use the signals from the other pods clock inputs as the sampling clock inputs or its qualifiers.

In the following screen, the clock channel (Clk1) of pod 1 of the master card (first card from bottom in the set of two cards) is used as the sampling clock. Clk2, Clk3, Clk4, and Clk5 are the state clock qualifiers.



You can use the "AND"/ "OR" conditional operators with the state clock qualifiers to add conditions and completely describe the clock as a combination of edges and highs or lows.

Determining Optimal Sample Positions and Thresholds

In the state sampling acquisition mode, you need to adjust the sampling positions on each U4154A channel relative to the sampling clock to make sure data is sampled when it is valid.

You can use the eye scan feature of the U4154A Logic Analyzer to automatically determine and set the optimal sample positions and thresholds for the individual signals. Clicking the **Eye scan: Sample Positions and Threshold** button in the **Sampling** tab displays the Eye Scan - Sample Positions and Threshold Settings dialog box where you can set up and run the eye scan measurement for U4154A channels. Refer to the topic Setting up and Running Eyescans in U4154A Logic Analyzer to learn more.

U4154B Logic Analyzer Notes

- Differences from Other Logic Analyzers (see [page 549](#))
- Channels and Memory Depth (see [page 549](#))
- Timing Mode Sampling Options/Period (see [page 550](#))
- State Mode Sampling (see [page 550](#))
- Timing Zoom (see [page 551](#))

Differences from Other Logic Analyzers

Compared to other Keysight logic analyzers supported in earlier releases, the U4154B logic analyzer differs in the following ways:

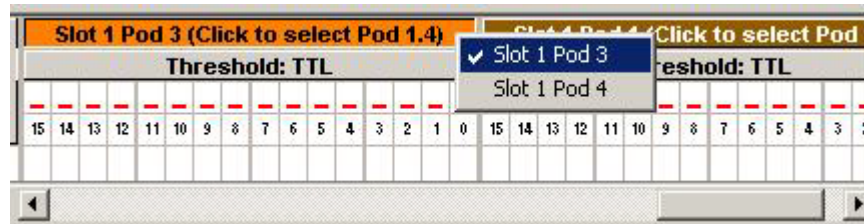
- U4154B supports two state speeds. By default, it supports the maximum state speed of 2.8 Gb/s. A "02G" license sets it to 4 Gb/s (2.5 GHz). These license options do not however control and set the timing speed of the U4154B module.
- The state sampling clock input is always from Pod1 of the U4154B module. In a multi-card set, the clock is allowed only on Pod1 of the master card. The master card is the middle U4154B module in a three-card set and the bottom U4154B module in a two-card set. You cannot use the signals from the other pods clock inputs as the sampling clock inputs. For more information, refer to the topic "[Setting up the State Sampling Options in U4154B Logic Analyzer](#)" on page 551.
- U4154B has improved clock qualifiers to capture traces out of reset. U4154B has Pod 7, Pod 5, Pod 4, Pod 3 and Pod 2 clock qualifiers on the clocking module. U4154B provides an additional new clock enable on Pod 7 of the clocking module to allow 'OR' qualification of upto 4 DDR memory CKE clock qualifiers AND qualified with RESET.
- You cannot logically split a U4154B logic analyzer module.
- The U4154B module is installed in one of the slots of the Keysight AXIe chassis. It connects to the host PC through the PCIe or USB interface of the AXIe chassis. You can connect three U4154B modules to form a three-card set in an AXIe chassis.
- The three U4154B modules support in a multi-card set in the AXIe chassis enables DDR4 interposers to perform full data capture at speeds >2.5 Gb/s.
- U4154B modules cannot merge into module sets with U4154A/B modules.
- There is no quarter channel timing mode.

Channels and Memory Depth

U4154B Memory depth	Option 004	Option 008	Option 016	Option 032	Option 064	Option 128	Option 200
	4 M	8 M	16 M	32 M	64 M	128 M	200 M
Channels	136 channels per U4154A module (This includes 128 data channels and 8 clock channels) In a multi-card set: 136 channels * number of U4154A modules installed in slots of an Keysight AXIe chassis You can install and connect two U4154A modules in an AXIe chassis to form a 2-card set with 272 channels.						

Timing Mode Sampling Options/Period

- **Full channel 2.5 GHz = Full Channel Timing Mode**
With this timing mode sampling option, you can use the full memory depth of your U4154B logic analyzer module, with data sampling period ranging from 400 ps to 10 ns. You can set the sample period using the Sample Period field.
- **Half channel 5.0 GHz = Half Channel Timing Mode**
With this timing mode sampling option, only one pod of each pod pair is available, and the memory depth is doubled. Channels assigned to unavailable pods are ignored. You can specify which pod to use in the Buses/Signals tab of the Analyzer Setup dialog by clicking the Pod button and selecting the desired pod.



Data is sampled and stored every 200 ps; this rate cannot be changed.

- The Quarter channel timing mode is not supported.
- **Transitional / Store Qualified Timing Mode**
This timing mode sampling option provides maximum duration of acquisition because data is only stored when a change from the last value is detected. See transitional timing on [page 365](#). Two sampling options are available in this mode:
 - **Transitional / Store qualified, (Full channel mode), 2.5 GHz** - With this sampling option, you can use the full memory depth of your U4154B logic analyzer module, with data being sampled at regular intervals but stored only when there is a signal transition across the threshold voltage level. You can set the sampling period from 400 ps to 10 ns for this mode.
 - **Transitional / Store qualified, (Half channel mode), 5.0 GHz** - With this sampling option, only one pod of each pod pair is available, and the memory depth is doubled. Channels assigned to unavailable pods are ignored. You can specify which pod to use in the Buses/Signals tab of the Analyzer Setup dialog by clicking the Pod button and selecting the desired pod. Data is sampled at regular intervals but stored only when there is a signal transition across the threshold voltage level. The sampling period is 200 ps for this mode and cannot be changed.

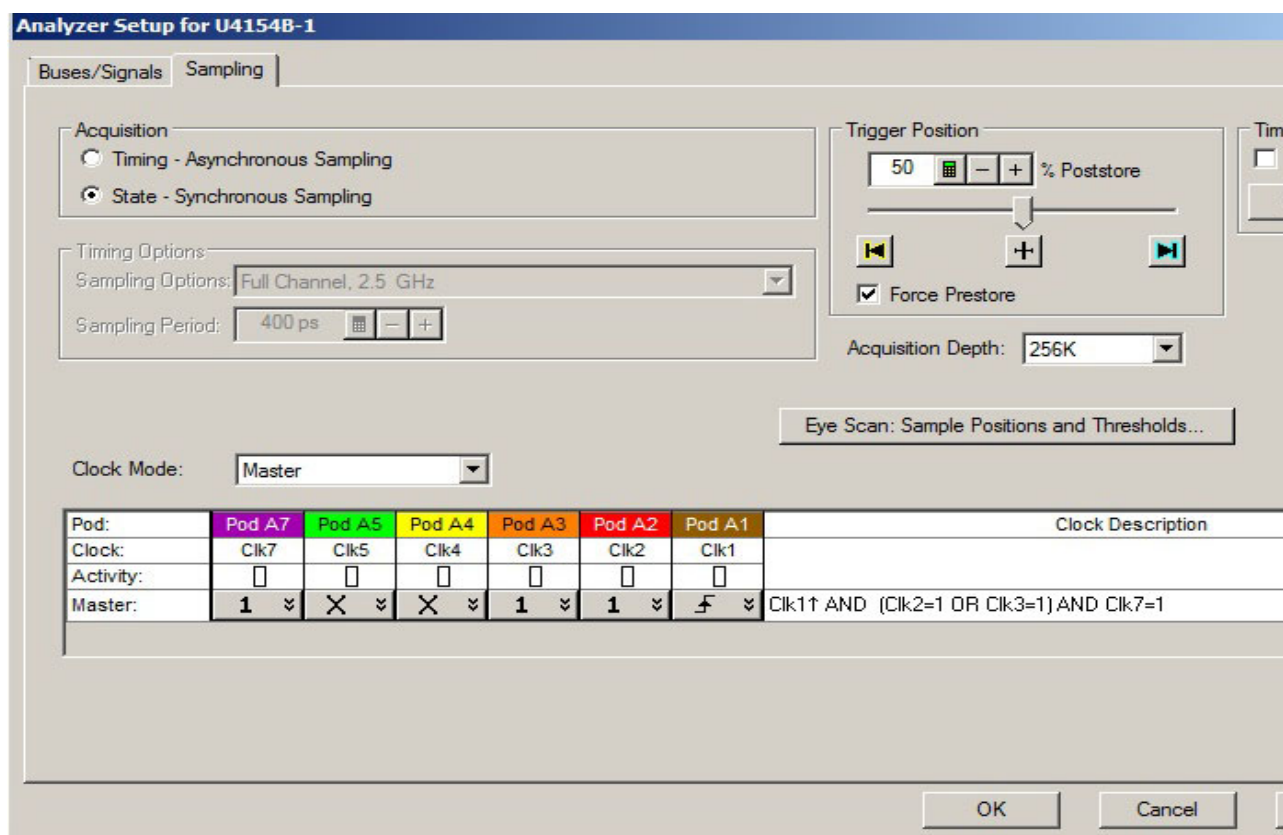
State Mode Sampling

- U4154B supports two state speeds. By default, it supports the maximum state speed of 2.8 Gb/s. A "02G" license sets it to 4 Gb/s (2.5 GHz). These license options do not however control and set the timing speed of the U4154B module.
- Based on the license option that you have purchased for U4154B, the maximum available state speed of U4154B is automatically set. If the licensed speed is 2.8 Gb/s (default) and while acquiring data, the clock frequency is found to exceed the maximum state speed available, an error message is displayed and the data is discarded.
- The state sampling clock input is always from Pod1 of the U4154B module. In a multi-card set, the clock is allowed only on Pod1 of the master card. The master card is the middle U4154B module in a three-card set and the bottom U4154B module in a two-card set. You cannot use the signals from other pods clock inputs as the sampling clock inputs. The qualifiers can take clock signal inputs from the Pod 7, Pod 5, Pod 4, Pod 3 and Pod 2 on the clocking module. The clock can be used as a rising, falling, or both edge clock.
- U4154B supports Full channel (Master) and Dual sample clock modes. In both these modes, the sampling option is as per the state speed license acquired for U4154B.

Timing Zoom Timing zoom collects additional high-speed timing data around the trigger of the logic analyzer. It uses a 256K-depth, 12.5 GHz timing analyzer to sample data every 80 ps on all channels. Timing zoom is available for all acquisition modes.

- See Also
- [“Setting up the State Sampling Options in U4154B Logic Analyzer](#)
 - [“Changing Advanced Probe Settings for Logic Analyzers](#)
 - [“Setting up and Running Eyescans in Logic Analyzers](#)
 - [“U4154B Logic Analyzer Specifications and Characteristics](#)
 - [“Pod and Channel Naming Conventions in Logic Analyzer](#)

Setting up the State Sampling Options in U4154B Logic Analyzer



The U4154B state sampling speed matches your device under test's clock rate, from 100 Mb/s up to 4 Gb/s.

The state sampling clock inputs let signals from the device under test specify when data should be captured.

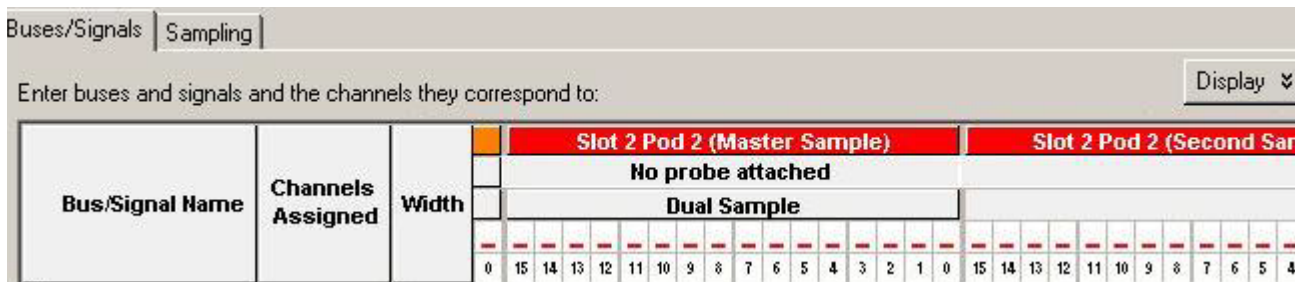
U4154B provides the following two options for state mode speeds (2.8 Gb/s and 4 Gb/s) at which the state sampling clock matches input clock edges from the device under test.

- **2.8 Gb/s speed** - This is the default state speed of U4154B. If, while acquiring data, the clock frequency is found to exceed this maximum available state speed, an error message is displayed and the data is discarded.
- **4 Gb/s speed** - If you have the O2G license for U4154B module, then the maximum available state speed of U4154B is automatically set to 4 Gb/s.

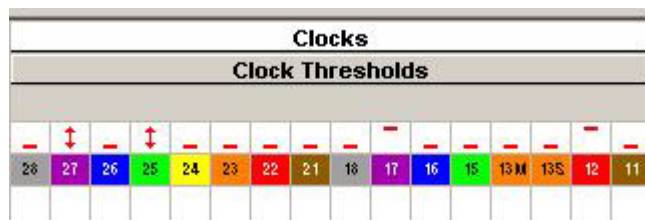
State Sampling Clock Mode

The state sampling clock mode specifies how the clock inputs are used for sampling. There are two state sampling clock modes to choose from:

- **Master** – all pods sampled on one master clock.
- **Dual Sample** – one pod in the pod pair sampled on one master clock but with different delays. When you enable this clock mode, the next step is to set up the pod pairs that you want to be dual sampled in the **Buses/Signals** tab of the **Analyzer Setup** dialog box (see [“To set up the dual sample sampling clock mode”](#) on page 116). Data on the active pod in a dual sampled pod pair is sampled twice (as master sample and second sample).



The pods for which you select the dual sample mode have their associated clock bit also dual sampled. The active clock bit of the pods in a dual sampled pod pair is the clock bit of the selected active pod in that pod pair. Notice that in the following screen, the clock channels of the dual sample pod pair (pod 1.3 and pod 1.4) are also dual sampled. These dual sampled clocks are represented as 13 M or C3M and 13 S or C3S (C3 Master Sample and C3 Second Sample).



For instructions on setting up these state sampling clock modes, see:

- [“To set up the master only sampling clock mode”](#) on page 114
- [“To set up the dual sample sampling clock mode”](#) on page 116

State Sampling Clock and Qualifiers

In U4154B, only one state sampling clock is provided. The state sampling clock is the clock of Pod 1 of the U4154B module. In a multi-card set, the clock is allowed only on Pod1 of the master card. The master card is the middle U4154B module in a three-card set and the bottom U4154B module in a two-card set. You cannot use the signals from the other pods clock inputs as the sampling clock inputs. To know which card is the master card in the set, click System Summary button in the Buses/Signals tab of the Analyzer Setup dialog box.

You can choose the sampling to occur on rising, falling, or both edges of the input clock signal.

There are following five qualifiers available for the state clock. You cannot use the signals from the other pods clock inputs as the sampling clock qualifiers. You can use the "AND"/ "OR" conditional operators with these state clock qualifiers to add conditions and completely describe the clock as a combination of edges and highs or lows.

Clock qualifier Pod input	Description
Pod 2	Can be used as "AND" or "OR" clock qualifiers
Pod 3	Can be used as "AND" or "OR" clock qualifiers
Pod 4	Can be used as "AND" or "OR" clock qualifiers
Pod 5	Can be used as "AND" or "OR" clock qualifiers
Pod 7	The "RESET" clock qualifier input on pod 7 is available as an "AND" input only when other clock qualifiers are set up as "OR" inputs. One situation in which this clock qualifier is particularly useful is to capture "RESET" when the other clock qualifiers are looking for CKE. This qualifier allows you to create OR qualification of upto 4 DDR memory CKE clock qualifiers along with an AND qualified with RESET.

The following are some points to remember when using the clock qualifier to Pod 7:

- This qualifier is only available when other clock qualifier(s) are used as "OR" qualifiers.
- This qualifier is only an "AND" qualifier. The Clock Description equation therefore, always adds the Ck7 as an AND condition. For example: $Ck1 \wedge (Ck4=0 \text{ OR } Ck5=1) \wedge Ck7=1$.
- Setup and hold of Pod 7 qualifier will be larger than other clock qualifiers.

In the following screen, the clock channel (Clk1) of pod 1 of the master card is used as the sampling clock. Clk2 and Clk3 are used as the OR'ed qualifiers along with Clk7 as the AND qualifier.

The screenshot shows the 'Eye Scan: Sample Positions and Thresholds...' dialog box. At the top, 'Clock Mode' is set to 'Master'. Below is a table with columns for Pod A7, Pod A5, Pod A4, Pod A3, Pod A2, and Pod A1. The 'Clock' row lists Clk7, Clk5, Clk4, Clk3, Clk2, and Clk1. The 'Activity' row shows checkboxes for each. The 'Master' row shows activity levels (1 or X) and a 'Clock Description' field containing the equation: $Ck1 \wedge (Ck2=1 \text{ OR } Ck3=1) \wedge Ck7=1$.

Pod:	Pod A7	Pod A5	Pod A4	Pod A3	Pod A2	Pod A1	Clock Description
Clock:	Clk7	Clk5	Clk4	Clk3	Clk2	Clk1	
Activity:	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Master:	1	X	X	1	1	1	$Ck1 \wedge (Ck2=1 \text{ OR } Ck3=1) \wedge Ck7=1$

Determining Optimal Sample Positions and Thresholds

In the state sampling acquisition mode, you need to adjust the sampling positions on each U4154B channel relative to the sampling clock to make sure data is sampled when it is valid.

You can use the eye scan feature of the U4154B Logic Analyzer to automatically determine and set the optimal sample positions and thresholds for the individual signals. Clicking the Eye scan: Sample Positions and Threshold Settings dialog box where you can set up and run the eye scan measurement for U4154B channels. Refer to the topic "[Setting up and Running Eyescans in Logic Analyzers](#)" on page 567 to learn more.

- See Also
- To set up the state sampling clock
 - "[Pod and Channel Naming Conventions in Logic Analyzer](#)" on page 561

16850-Series Logic Analyzer Notes

- Channels and Memory Depth
- Maximum State Sampling Speed
- Timing Mode Sampling Options/Period
- State Mode Sampling Options
- Timing Zoom

Channels and
Memory Depth

	Default	Option 004	Option 008	Option 016	Option 032	Option 064	Option 128
Memory depth	2 M	4 M	8 M	16 M	32 M	64 M	128 M

Channels *Memory depth upgrades for the 16850 Series logic analyzers can be ordered by appending the relevant memory option number to the following upgrade model numbers:
16851AU, 16852AU, 16853AU, 16854AU
For instance, 16851AU-008 to upgrade to 8M samples.
(See also Installing Licensed Hardware Upgrades.)

	16851A	16852A	16853A	16854A
Channels	34	68	102	136

See also Memory Depth and Channel Count Trade-offs (see [page 363](#)).

Maximum State
Sampling Speed

	Default	Option 700*
Max. state clock rate	350 MHz	700 MHz
Max. state data rate	700 Mb/s	1400 Mb/s

*State sampling speed upgrade for the 16850 Series logic analyzers can be ordered by appending the -700 option number to the following upgrade model numbers:
16851AU, 16852AU, 16853AU, 16854AU
(See also Installing Licensed Hardware Upgrades (see [page 139](#)).)

Timing Mode
Sampling
Options/Period

- **Full channel, 2.5 GHz** = Full Channel Timing Mode
With this sampling option, you can use the full memory depth of your logic analyzer, with data sampling period on all channels as 400 ps to 10 ns. You can set the sample rate to go slower with the Sample Period field.
- **Half channel, 5 GHz** = *Half Channel Timing Mode*
With this sampling option, only one pod of each pod pair is available, and the memory depth is doubled. Channels assigned to unavailable pods are ignored. You can specify which pod to use in the Buses/Signals tab of the Analyzer Setup dialog by clicking the Pod button and selecting the desired pod.
Data sampling period is 200 ps and is fixed.
- **Transitional / Store qualified, Full channel 2.5 GHz / Half channel 5.0 GHz** = Transitional / Store Qualified Timing Modes
At the 2.5 GHz sample rate, one pod pair (34 channels) must be reserved for time tag storage. At slower sample rates, you can get full channels by using 1/2 (or less) of a module's acquisition memory depth (for more information, see Memory Depth and Channel Count Trade-offs (see [page 363](#))).
Transitional / Store Qualified Timing mode provides maximum duration of acquisition because data is only stored when a change from the last value is detected. The sampling period ranges from 10 ns to 400 ps for full channel mode and is fixed at 200 ps for half channel mode. (see [page 365](#)).

Timing Zoom

Timing zoom collects additional high-speed timing data around the trigger of the logic analyzer. It uses a 256 K-sample, 12.5 GHz timing analysis sample rate to sample data at a sampling period of 80 ps.

See Also

- 16850 Series Logic Analyzer Specifications and Characteristics (see [page 600](#))

16860-Series Logic Analyzer Notes

- Differences from Other Logic Analyzers
- Channels and Memory Depth
- Timing Mode Sampling Options/Period
- State Mode Sampling Options
- State Sampling Clock Modes
- State Sampling Clock(s)
- State Sampling Clock Qualifiers
- Timing Zoom

Differences from
Other Logic
Analyzers

Compared to other Keysight logic analyzers supported in earlier releases, the 16860-series logic analyzers differ in the following ways:

- Supports two state sampling options - Single Clock and Multiple Clocks.
 - The **Single Clock** option provides a single master clock and supports the state mode sampling speed ranging from 12.5 Mega samples per second to 700 MHz (with "700" state speed license option).
 - The **Multiple Clocks** mode provides up to four clocks and is a low speed option more suitable for general-purpose logic analysis with state mode sampling speed ranging from 0 Mega samples per second to 350 MHz.

To know more about these two state sampling options, refer to the topic [Selecting the State Sampling Option for a 16860-Series Logic Analyzer](#).

- Supports two state speeds. The default state speed is 350 MHz. The maximum state speed supported is 700 MHz (1.4 GSa/s). The "700" license sets it to 700 MHz. This license option does not however control and set the timing speed of the 16860-series logic analyzer.
- Supports the Master, Master/Slave/Demux, and Dual Sample clock modes. The availability of these clock modes depend on the state sampling option (Single Clock or Multiple clocks) that you have selected for your logic analyzer.
- Supports a new 10 GHz Timing (Asynchronous) sampling option - **Quarter Channel Timing Mode**. This option is only available on 16862A (68 channels) and 16864A (136 channels) models in the 16860-series of logic analyzers. Also, the "T10" license of 16860-series logic analyzer is required for this timing mode.
- Allows you to set the sampling positions for all Timing (Asynchronous) sampling options using the Signal Deskew tool.
- Supports **Clock hysteresis** for the state sampling clock on Pod1 of the 16860-series logic analyzers. Using this feature, you can set a value (in milliVolts) between 0 and 1 volts around the clock threshold. This allows you to avoid false sampling on noisy clock inputs where a differential clock input turns off and floats to zero volts. Refer to the topic [Setting Clock Hysteresis for the State Sampling Clock](#) to know more.
- Does not support multiframe capabilities.
- In a 16860 setup, you can load a .xml Logic Analyzer configuration file that you created/previously saved in a 16850-series, U4154A, or U4154B setup. You cannot, however load a Standard configuration (.ala) file that you created/previously saved in a 16850-series, U4154A, or U4154B setup.

Channels and
Memory Depth

Four models of the 16860-series logic analyzers are available. The number of channels available in each of these models is given below.

	16861A	16862A	16863A	16864A
Channels	34 (32 data channels and 2 clock channels)	68 (64 data channels and 4 clock channels)	102 (96 data channels and 6 clock channels)	136 (128 data channels and 8 clock channels)

The number of channels and pods available are also impacted by the Timing mode sampling option that you select or on selecting the Dual Sample clock mode.

16860-Series Memory Depth License Options	For State (Synchronous) Sampling Modes	For Timing (Asynchronous) Sampling Modes		
		Full Channel	Half Channel	Quarter Channel
Standard memory	2 M	2 M	4 M	8 M
Option 004*	4 M	4 M	8 M	16 M
Option 008*	8 M	8 M	16 M	32 M
Option 016*	16 M	16 M	32 M	64 M
Option 032*	32 M	32 M	64 M	128 M
Option 064*	64 M	64 M	128 M	256 M
Option 128*	128 M	128 M	256 M	512 M
Option 200* (Available only in the Single Clock State Sampling option)	200 M	200 M	400 M	800 M
Option 400* (Available only in the Single Clock State Sampling option)	400 M	400 M	800 M	1.6 G

* You can order Memory depth upgrades for the 16860-series logic analyzers by appending the relevant memory option number to the following upgrade model numbers:
16851AU, 16852AU, 16853AU, 16854AU
For instance, 16851AU-008 to upgrade to 8M samples.
(See also Installing Licensed Hardware Upgrades.)

Timing Mode
Sampling
Options/Period

Timing Mode Sampling Option	Description																									
Full channel, 2.5 GHz (Full Channel Timing Mode)	<p>Maximum acquisition rate supported in this option - 2.5 GHz.</p> <p>With this timing mode sampling option:</p> <ul style="list-style-type: none"> You can use the full memory depth of your logic analyzer. Data sampling period ranging from 400 ps to 10 ns. You can set the sample period using the Sample Period field. All the pods of the logic analyzer are available. All the clock channels are available. 																									
Half channel, 5.0 GHz (Half Channel Timing Mode)	<p>Maximum acquisition rate supported in this option - 5.0 GHz.</p> <p>With this timing mode sampling option:</p> <ul style="list-style-type: none"> The memory depth is double the available depth (refer to the Memory Depth table above). Data is sampled and stored every 200 ps; this rate cannot be changed. Only odd pod of a pod pair is available that is, Pod 1, 3, 5, and 7. Channels assigned to the unavailable even pods are ignored. Clock channels are reduced to half of the total available clock channels and are for only odd pod of each pod pair. 																									
	<table border="1"> <thead> <tr> <th>Clks</th> <th>Slot 1 Pod 7</th> <th>Slot 1 Pod 5</th> <th>Slot 1 Pod 3</th> <th>Slot 1 Pod 1</th> </tr> </thead> <tbody> <tr> <td>Clock Th</td> <td>Threshold: TTL +/- offset</td> <td>Threshold: TTL</td> <td>Threshold: TTL</td> <td>Threshold: GTLPlus</td> </tr> <tr> <td></td> <td>↑ ↓</td> <td>↑ ↓</td> <td>↑ ↓</td> <td>↑ ↓</td> </tr> <tr> <td></td> <td>15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0</td> <td>15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0</td> <td>15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0</td> <td>15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0</td> </tr> <tr> <td></td> <td>✓</td> <td></td> <td></td> <td>✓ ✓ ✓ ✓ ✓</td> </tr> </tbody> </table>	Clks	Slot 1 Pod 7	Slot 1 Pod 5	Slot 1 Pod 3	Slot 1 Pod 1	Clock Th	Threshold: TTL +/- offset	Threshold: TTL	Threshold: TTL	Threshold: GTLPlus		↑ ↓	↑ ↓	↑ ↓	↑ ↓		15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0	15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0	15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0	15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0		✓			✓ ✓ ✓ ✓ ✓
Clks	Slot 1 Pod 7	Slot 1 Pod 5	Slot 1 Pod 3	Slot 1 Pod 1																						
Clock Th	Threshold: TTL +/- offset	Threshold: TTL	Threshold: TTL	Threshold: GTLPlus																						
	↑ ↓	↑ ↓	↑ ↓	↑ ↓																						
	15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0	15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0	15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0	15 14 13 12 11 10 9 8 7 6 5 4 3 2 1 0																						
	✓			✓ ✓ ✓ ✓ ✓																						
Quarter channel, 10 GHz (Quarter Channel Timing Mode)	<p>Maximum acquisition rate supported in this option - 10.0 GHz.</p> <p>This timing mode sampling option is only available:</p> <ul style="list-style-type: none"> If you have the 16862A or 16864A model of logic analyzer. If you have installed the T10 Speed license option of the 16860-series logic analyzer. <p>With this timing sampling option:</p> <ul style="list-style-type: none"> The memory depth is four times the available depth (refer to the Memory Depth table above). Data is sampled and stored every 100 ps; this rate cannot be changed. Only odd pod of a pod pair is available that is, Pod 1, 3, 5, and 7. Only even channels (0, 2, 4, 6, 8, 10, 12, 14) for each odd pod are available. Clock channels are reduced to one-fourth of the total available clock channels and are for only odd pod of each pod pair. 																									
	<table border="1"> <thead> <tr> <th>Clk</th> <th>Slot 1 Pod 7</th> <th>Slot 1 Pod 5</th> <th>Slot 1 Pod 3</th> <th>Slot 1 Pod 1</th> </tr> </thead> <tbody> <tr> <td>Clk</td> <td>TTL +/- offset</td> <td>Threshold: TTL</td> <td>Threshold: TTL</td> <td>GTLPlus</td> </tr> <tr> <td></td> <td>↑ ↓</td> <td>↑ ↓</td> <td>↑ ↓</td> <td>↑ ↓</td> </tr> <tr> <td></td> <td>14 12 10 8 6 4 2 0</td> <td>14 12 10 8 6 4 2 0</td> <td>14 12 10 8 6 4 2 0</td> <td>14 12 10 8 6 4 2 0</td> </tr> <tr> <td></td> <td>✓</td> <td></td> <td></td> <td>✓ ✓ ✓ ✓</td> </tr> </tbody> </table>	Clk	Slot 1 Pod 7	Slot 1 Pod 5	Slot 1 Pod 3	Slot 1 Pod 1	Clk	TTL +/- offset	Threshold: TTL	Threshold: TTL	GTLPlus		↑ ↓	↑ ↓	↑ ↓	↑ ↓		14 12 10 8 6 4 2 0	14 12 10 8 6 4 2 0	14 12 10 8 6 4 2 0	14 12 10 8 6 4 2 0		✓			✓ ✓ ✓ ✓
Clk	Slot 1 Pod 7	Slot 1 Pod 5	Slot 1 Pod 3	Slot 1 Pod 1																						
Clk	TTL +/- offset	Threshold: TTL	Threshold: TTL	GTLPlus																						
	↑ ↓	↑ ↓	↑ ↓	↑ ↓																						
	14 12 10 8 6 4 2 0	14 12 10 8 6 4 2 0	14 12 10 8 6 4 2 0	14 12 10 8 6 4 2 0																						
	✓			✓ ✓ ✓ ✓																						
Transitional / Store Qualified Timing Mode	<p>This timing mode sampling option provides the maximum duration of acquisition because data is only stored when a change from the last value is detected. Data is sampled at regular intervals but stored only when there is a signal transition across the threshold voltage level. See transitional timing. Three sampling options are available in this mode:</p> <ul style="list-style-type: none"> Transitional / Store qualified, (Full channel mode), 2.5 GHz Transitional / Store qualified, (Half channel mode), 5.0 GHz Transitional / Store qualified, (Quarter channel mode), 10.0 GHz - Available only when you have installed the T10 timing speed license and the logic analyzer model is either 16862A or 16864A. 																									

NOTE For the 16860-series logic analyzers, you can set the sampling positions for the Timing mode sampling options as well. You use the Signal Deskew Tool to accomplish this. Refer to the online help available with this tool's GUI to know more.

State Mode Sampling Option

State Sampling Option	Description
Single Clock	Faster state mode sampling speeds are supported, but some triggering and advanced clocking features are not available. Refer to the table below to know about the maximum acquisition rate available in the Single Clock sampling option.
Multiple Clocks	Advanced clocking capabilities and multiple clocks are supported, but at a slower state sampling speed. Refer to the table below to know about the maximum acquisition rate available in the Multiple Clocks sampling option.

Refer to the topic [Selecting the State Sampling Option for a 16860-Series Logic Analyzer](#) to get a comparison between these two sampling options and to know how these options are used in the context of a 16860-Series Logic Analyzer.

16860-series logic analyzers support the following two state speeds. Based on the state speed license option that you have purchased for your logic analyzer, the maximum available state speed of your logic analyzer is automatically set.

State Speed License Option	Maximum Acquisition Rate	
	In Single Clock State Sampling Option	In Multiple Clock State Sampling Option
Standard (Default) State Speed 350 MHz (700 Mb/s)	Captures a 350MHz signal on either edge of the clock up to 350 MSP/s Captures a 350MHz signal on both edges of the clock up to 700 MSP/s	Captures a 350MHz signal on either edge of the clock up to 350 MSP/s Captures a 350MHz signal on both edges of the clock up to 700 MSP/s
Option 700* 700 MHz (1.4 Gb/s)	Captures a 700MHz signal on either edge of the clock up to 700 MSP/s Captures a 700MHz signal on both edges of the clock up to 1400 MSP/s	Captures a 350MHz signal on either edge of the clock up to 350 MSP/s Captures a 350MHz signal on both edges of the clock up to 700 MSP/s

*State sampling speed upgrade for the 16860 Series logic analyzers can be ordered by appending the -700 option number to the following upgrade model numbers: 16861AU, 16862AU, 16863AU, 16864AU
(See also [Installing Licensed Hardware Upgrades](#).)

State Sampling Clock Modes

The state sampling clock mode specifies how the clock inputs are used for sampling. The clock modes available for a 16860-series logic analyzer depends on the state sampling option (Single clock or Multiple Clocks) that you selected.

Clock Modes Available for the Single Clock Sampling Option	Clock Modes Available for the Multiple Clocks Sampling Option
Master Dual Sample	Master Master/Slave/Demux

For a detailed description of these clock modes, refer to the topics:

- Master
- Dual Sample
- Master/Slave
- Demux

For information on how to set up these clock modes, see:

- To set up the master only sampling clock mode
- To set up the dual sample sampling clock mode
- To set up the master/slave sampling clock mode
- To set up the demultiplex sampling clock mode

- State Sampling Clock(s) The 16860-series logic analyzers support both Single and Multiple clocks sampling options. Therefore, you can set up a single or multiple state sampling clocks.
- In case of a single clock, the state sampling clock input is always from Pod1 of the logic analyzer.
 - In case of multiple clocks, the state sampling clock input can be from multiple Pods (Pod1, 2, 3, and 4) of the logic analyzer in an Ored combination.

To know how to set up a single or multiple clocks, refer to the topic To set up the state sampling clock.

When setting up multiple clocks, you can also use the Advanced Clocking feature to define complex clock descriptions.

- State Sampling Clock Qualifiers For a Single Clock
- The clock qualifiers can take clock signal inputs from the following pods of logic analyzer.

16861A	16862A/16863A	16864A
One clock qualifier (Pod 2)	Three clock qualifiers (Pod 2, 3, 4)	Five clock qualifiers (Pods 2, 3, 4, 5, and 7)
You cannot use the signals from the other pods clock inputs as the sampling clock qualifiers.		

Clock qualifier Pod input	Usage
Pod 2	Can be used as AND or OR clock qualifier
Pod 3	Can be used as AND or OR clock qualifier
Pod 4	Can be used as AND or OR clock qualifier
Pod 5	Can be used as AND or OR clock qualifier
Pod 7	The RESET clock qualifier input on pod 7 is available as an AND input only when other clock qualifiers are set up as OR inputs. One situation in which this clock qualifier is particularly useful is to capture RESET when the other clock qualifiers are looking for CKE. This qualifier allows you to create OR qualification of upto 4 DDR memory CKE clock qualifiers along with an AND qualified with RESET.

The following are some points to remember when using the clock qualifier on Pod 7:

- This qualifier is only available when other clock qualifier(s) are used as OR qualifiers.
- This qualifier is only an AND qualifier. The Clock Description equation therefore, always adds the Ck7 as an AND condition. For example: $Ck1 \wedge (Ck4=0 \vee Ck5=1) \wedge Ck7=1$
- Setup and hold of Pod 7 qualifier will be larger than other clock qualifiers.

For multiple clocks

The clock qualifiers can take clock signal inputs from the following pods of logic analyzer.

16861A	16862A,16863A/16864A
One clock qualifier (Pod 1 or 2)	Up to two clock qualifiers (Pod 1, 2, 3, 4) Though three clock qualifiers are displayed for use, you can set up to 2 clock qualifiers for a clock.
You cannot use the signals from the other pods clock inputs as the sampling clock qualifiers.	

Determining Optimal Sample Positions and Thresholds

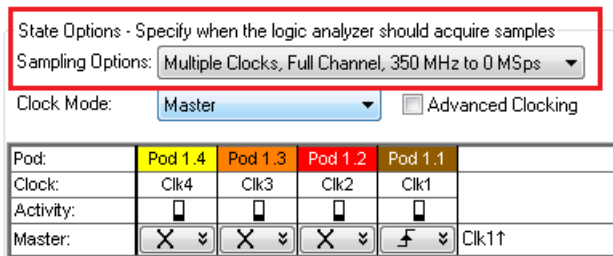
In the state sampling acquisition mode, you need to adjust the sampling positions on 16860 channels relative to the sampling clock to make sure data is sampled when it is valid.

You can use the eye scan feature of the 16860 Logic Analyzer to automatically determine and set the optimal sample positions and thresholds for the individual signals. Clicking the Eye scan: Sample Positions and Threshold button in the Sampling tab displays the Eye Scan - Sample Positions and Threshold Settings dialog box where you can set up and run the eye scan measurement for 16860 channels. Refer to the topic Setting up and Running Eyescans in Logic Analyzer to learn more.

Timing Zoom

Timing zoom collects additional high-speed timing data around the trigger of the logic analyzer. It uses a 256 K-sample, 12.5 GHz timing analysis sample rate to sample data at a sampling period of 80 ps.

Selecting the State Sampling Option for a 16860-Series Logic Analyzer



The state sampling option that you select specifies the speed up to which the state mode sampling clock will match input clock edges from the device under test.

The following table describes and compares the state sampling options that you can set for a 16860-series logic analyzer.

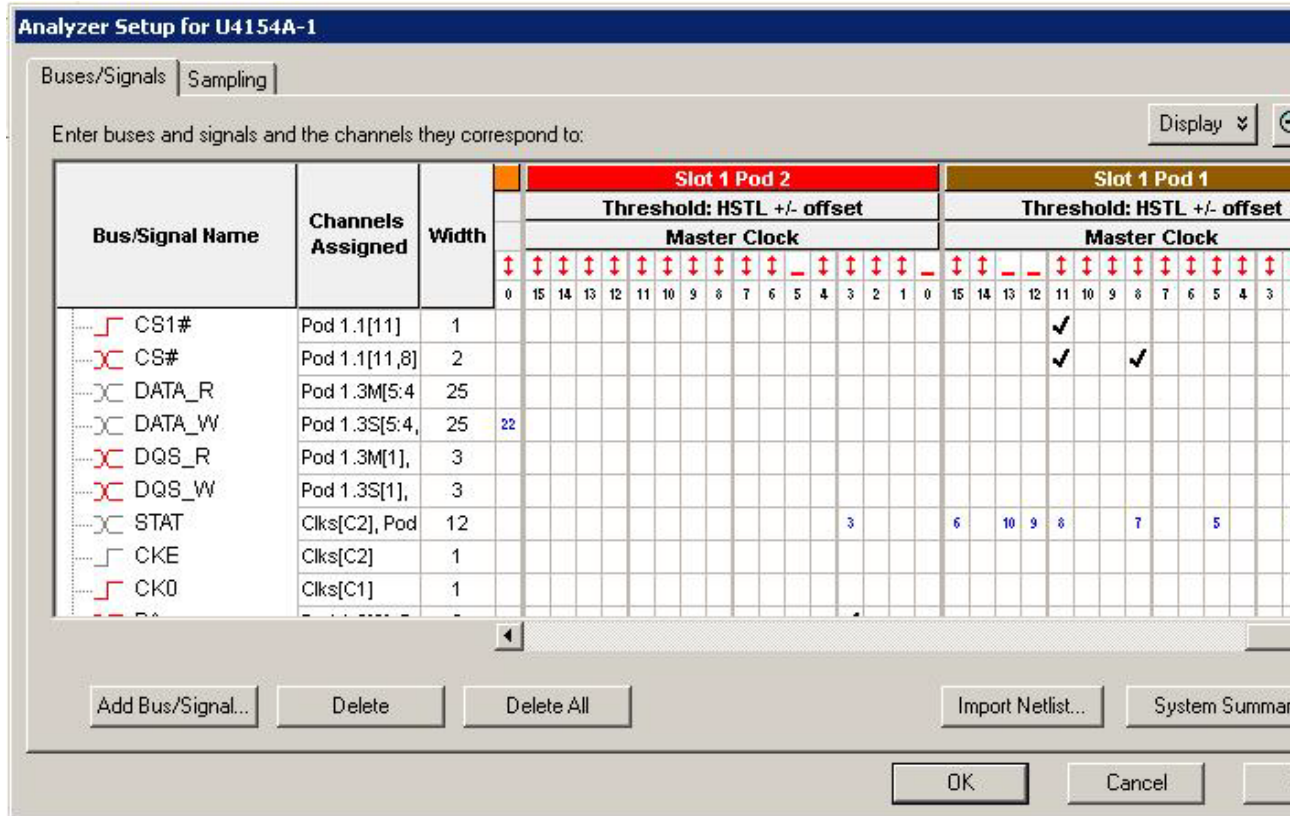
State Sampling Options	Single Clock, Full Channel, 700MHz to 12.5 MSps	Multiple Clocks, Full channel, 350MHz to 0 MSps
Usage	Faster state mode sampling speeds are supported, but some triggering and advanced clocking features are not available. More suitable for DDR/LPDDR data captures with high data rates.	Default option Supports slower state sampling speeds but allows advanced clocking capabilities and multiple clocks. More suitable for general-purpose logic analysis.
State Sampling Speeds Supported	700MHz to 12.5 MSps (if state speed option 700 is installed) 350MHz to 12.5 MSps (if state speed option 700 is NOT installed) Note: An error message is displayed if the input clock edges from the DUT are outside the lowest and highest speed range allowed in this option. In such a situation, measurements are not collected.	350 MHz to 0 MSps (with or without the state speed option 700 installed) Note: A warning message is displayed if the input clock edges from the DUT are faster than the highest speed allowed in this option. In such a situation, measurements are still available but there is a risk of some missing data. You can use the Single Clock option in such a situation.
Maximum Acquisition Rate	At the 350MHz state sampling speed: Captures a 350MHz signal on either edge of the clock up to 350 MSp/s Captures a 350MHz signal on both edges of the clock up to 700 MSp/s At the 700MHz state sampling speed: Captures a 700MHz signal on either edge of the clock up to 700 MSp/s Captures a 700MHz signal on both edges of the clock up to 1400 MSp/s	At the 350MHz state sampling speed: Captures a 350MHz signal on either edge of the clock up to 350 MSp/s Captures a 350MHz signal on both edges of the clock up to 700 MSp/s

State Sampling Options	Single Clock, Full Channel, 700MHz to 12.5 MSps	Multiple Clocks, Full channel, 350MHz to 0 MSps
Number of Clocks Supported	Single clock (Always on Pod1 of the logic analyzer)	Up to 2 clocks (in an Ored combination) for a 16861A logic analyzer. Up to 4 clocks (in an Ored combination) for a 16862A/3A/4A logic analyzer. Master and slave clocks are counted separately. Refer to the topic To set up the state sampling clock to know about setting multiple clocks.
State Sampling Clock Modes Supported	Master Dual Sample	Master Master/Slave Demux
Advanced Clocking Capabilities	Not supported	Supported You can specify complex clock setups with the available clock channels used both as an edge and a qualifier in the same clock description. Refer to the topic Advanced Clocking Setup dialog to know more.
Clock Qualifiers	Maximum of five clock qualifiers (Pod 2, 3, 4, 5, 7) for the 16864A model. Maximum of three clock qualifiers (Pod 2, 3, 4) for the 16862A/3A models. Maximum of one clock qualifier (Pod 2) for the 16861A model.	Maximum of two clock qualifiers (Pod 1, 2, 3, 4) for the 16862A/3A/4A models. Note: Though there are three clock qualifiers displayed, you can use a maximum of only two qualifiers for a clock. The third qualifier is automatically set to "Don't Care" and can be used only as an Ored clock in the clock description. Maximum of one clock qualifier for the 16861A model.
Triggering Capabilities Differences	Number of trigger sequences - 8 Burst patterns supported - Yes Number of timers - 1 Global counters supported - No Event counters supported - Yes	Number of trigger sequences - 16 Burst patterns supported - No Number of timers - 3 Global counters supported - 2 Event counters supported - No
The logic analysis features that are not included in this table work the same in Single clock and Multiple Clocks options.		

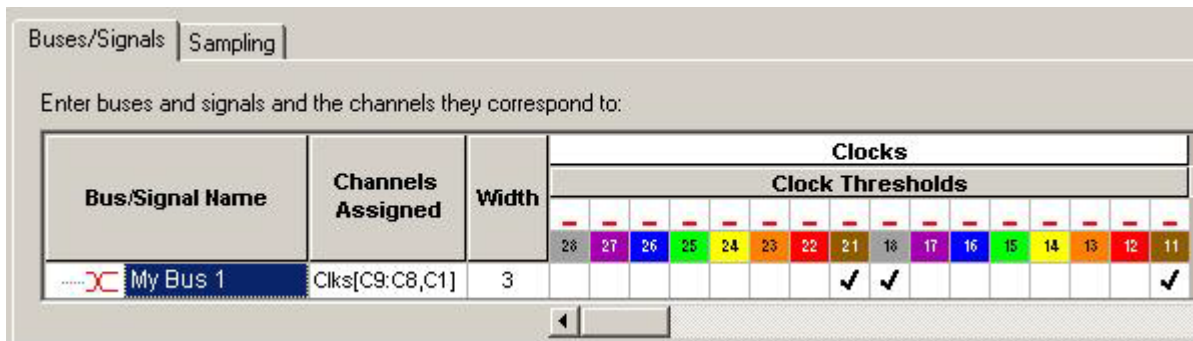
Pod and Channel Naming Conventions in Logic Analyzer

In U4154A/B Logic Analyzer module:

- Slots are named "1" through "5" in the Keysight AXIe chassis starting with the bottom slot.
- A U4154A/B module installed in a slot has eight pods. These pods are named "1" to "8".
- A U4154A/B pod is represented as <Slot_number_of_module><Pod_number>. For instance, Slot 1 Pod 2 represents the Pod 2 of the U4154A/B module that you installed in slot 1 of chassis. This pod is also referred to as Pod 1.2 in Logic Analyzer GUI.
- Each pod has 16 data channels named "0" through "15". Pod 2.8[11] represents the channel 11 of the Pod 8 of the module installed in slot 2.



- Each pod has one clock channel making it a total of eight clock channels for a U4154A/B module. These clock channels are named C1 through C8. If you have installed multiple U4154A/B module in an AXle chassis, then the clock channels of the next U4154A module are named C9 through C16 and so on for the rest of the modules in the set. The following is an example of the clock channels naming convention in U4154A/B.



The clock channel of Pod 8 of the U4154A/B module installed in slot 1 is represented as:

- 18
- Clk8
- C8

Similarly, the clock channel of Pod 4 of the U4154A/B module installed in slot 2 is represented as:

- 24

- Clk12
- C12

The input from the clock channel of Pod 1 is used as the state sampling clock input. If you have installed multiple U4154A/B cards in chassis, then the clock of Pod 1 of the master card in the set is used as the state sampling clock. For instance, if there are five cards in the set, then the third card from the bottom is the master card. The clock of Pod 1 of this card is the state sampling clock.

Changing Advanced Probe Settings for Logic Analyzers

You can change the settings for the probes used with the logic analyzers. You can enable peaking for channels of these logic analyzers to compensate for the additional high frequency attenuation that some probing solutions provide on target signals. Enabling the peaking can improve the capture window of the input signals by peaking the edges of the input signals. This is particularly useful in situations when the additional attenuation on target signals becomes significant at high frequencies and DDR3 edge rates.

When you enable peaking with the DDR3 Eyefinder tool, it can increase the capture window for the data signals therefore providing increased accuracy for capturing the data. (The DDR3 Eyefinder tool helps you set the appropriate sampling positions for read and write data signals to capture DDR data. it is specifically designed to find DDR3 data signal eyes.)

It is recommended that you run the DDR3 Eyefinder tool on the high-speed target systems with the peaking enabled and then disabled to determine the setting that best suits the target system.

CAUTION

Changing the peaking settings can improve or reduce the performance of the probing system. For instance, if the DDR BGA Interposer probe is used at speeds of DDR3-1600 or higher, enabling the peaking results in improving the capture window.

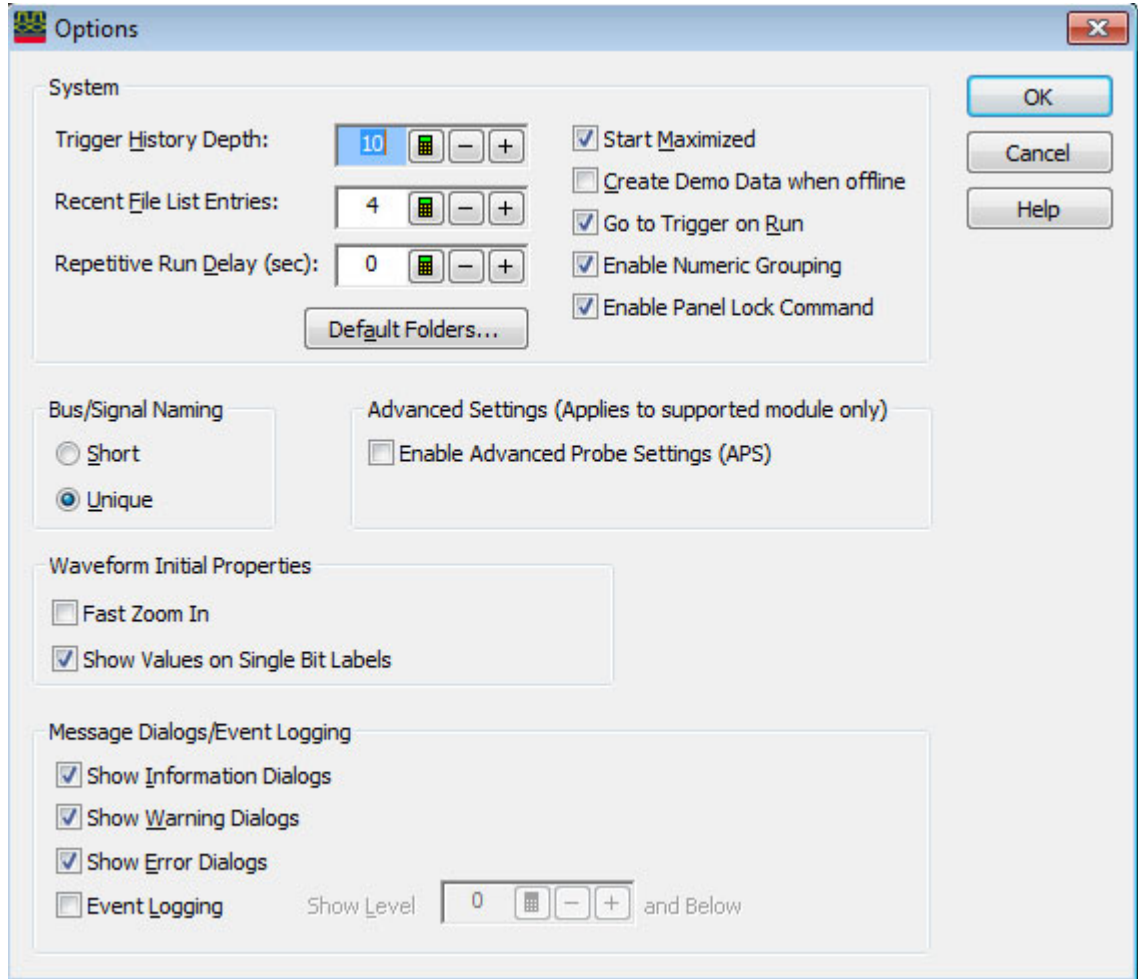
NOTE

The advanced probe settings feature is supported for U4154A/B, U4164A, and 16850-series logic analyzers.

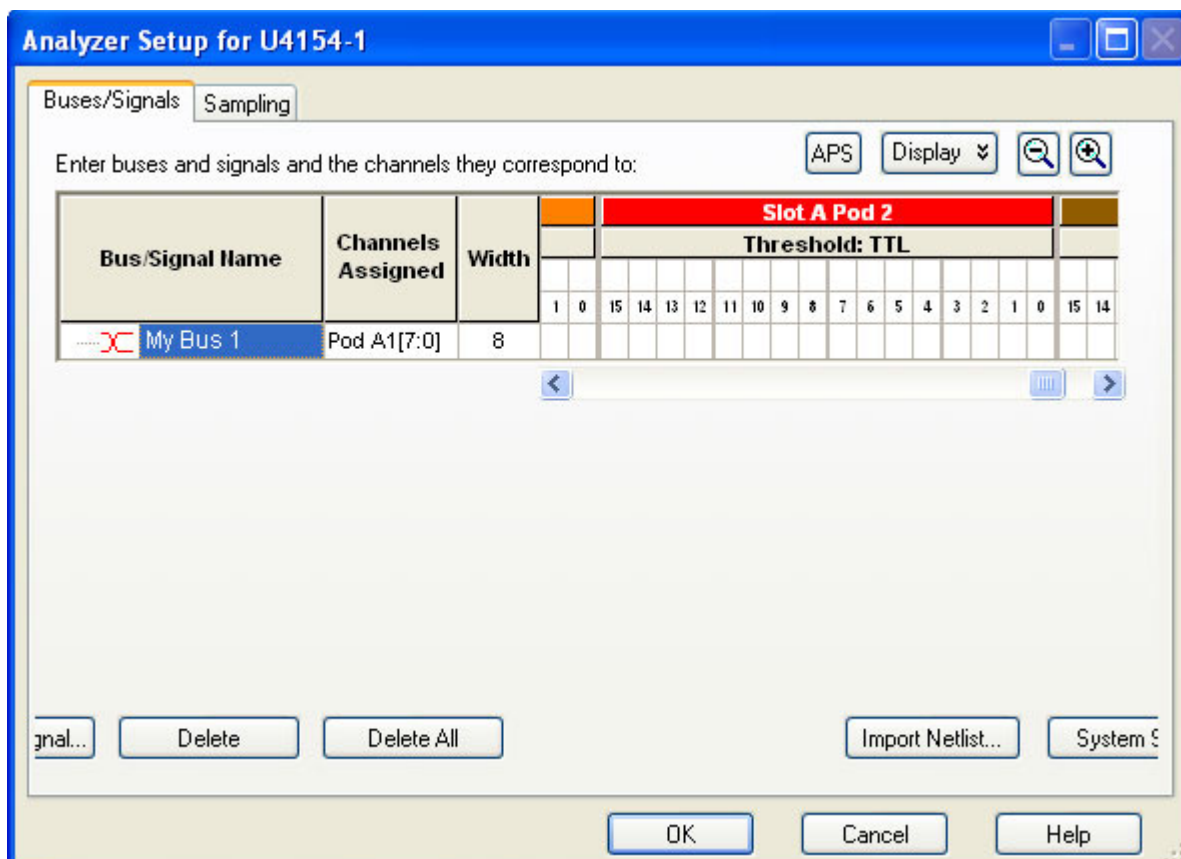
To change probe settings, you must first enable the probe settings and then adjust these settings. These procedures are described below.

To enable Advanced Probe Settings (APS) for the Logic Analyzer Module

- 1 From the menu bar, select **Edit > Options**.
- 2 In the **Options** dialog, select the **Enable Advanced Probe Settings (APS)** checkbox.
- 3 Click **OK**.



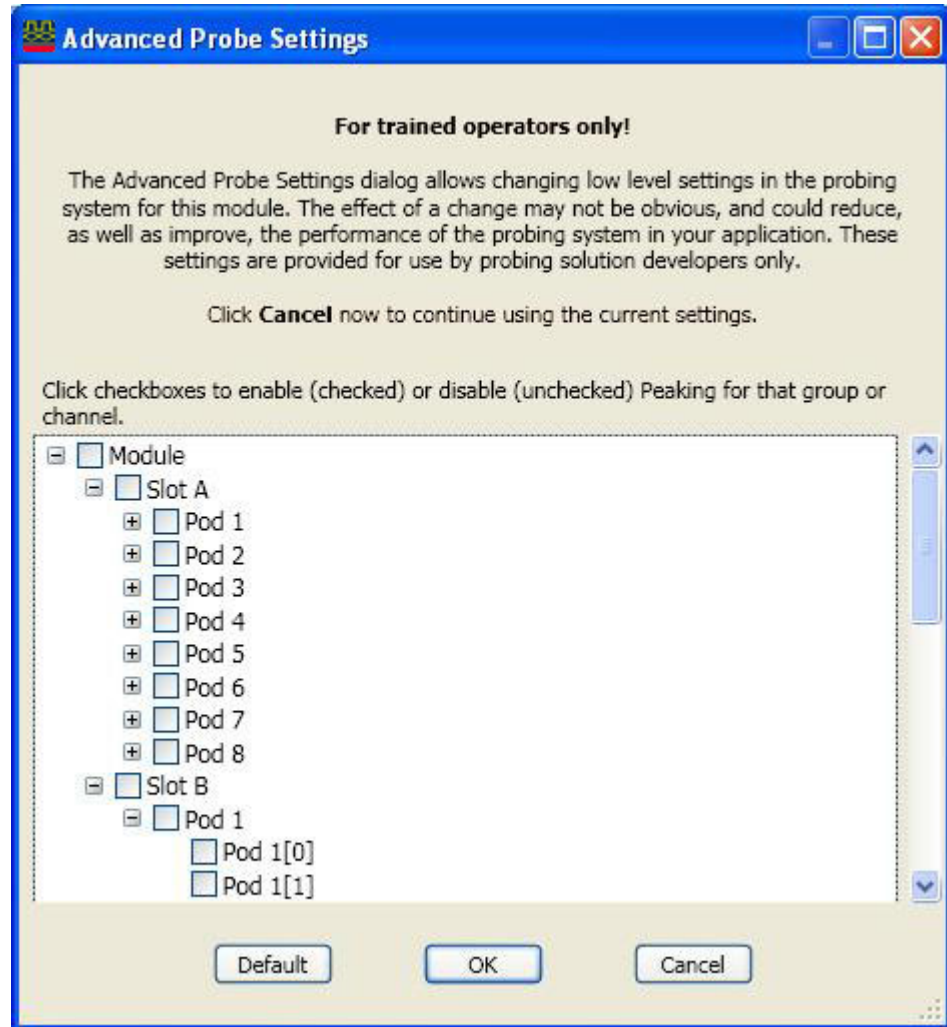
On enabling the advanced probe settings in the Options dialog, the **APS** button is added to the **Buses/Signals** tab of the **Analyzer Setup** dialog of the logic analyzer module. You can use this button to view and change the advanced probe settings.



To change the Advanced Probe Settings for the Logic Analyzer Module

- 1 From the menu bar, select **Setup > Bus/Signal**.
- 2 In the **Buses/Signals** tab of the Analyzer Setup dialog, click the **APS** button.

The **Advanced Probe Settings** dialog is displayed. All the logic analyzer cards that you installed in different slots of the chassis to make up the logical module are displayed. On expanding a slot, the pods available for that logic analyzer card are displayed. Each pod further expands to the supported channels per pod. The following screen displays the advanced probe settings dialog for the U4154A/B logic analyzer. In this screen, the logical module comprises of two U4154A/B modules installed in slot A and B of the AXIe chassis. For each U4154A/B, there are eight pods available. Each of these pods expands to 16 data channels and 1 clock channel.



- 3 You can choose to enable the peaking at the channel/pod/slot/logical module level by selecting the checkbox displayed with these options. By default, peaking is disabled at all these levels.
- 4 Click **OK**.

Setting Clock Hysteresis for the U4164A State Sampling Clock

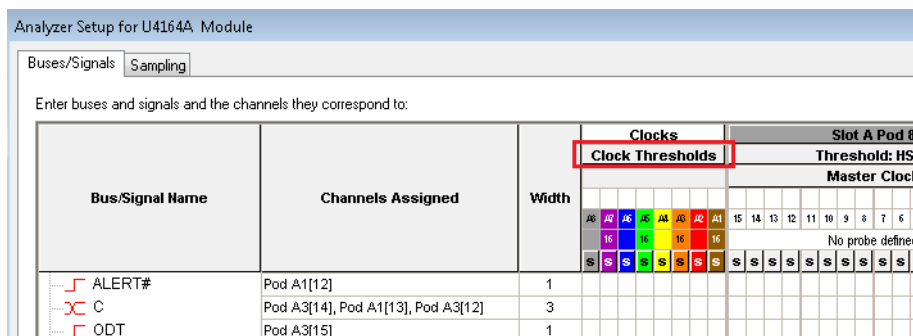
You can set the Clock hysteresis feature On or Off for the state sampling clock on Pod1 of the U4164A module. Using this setting, you can set a value (in milliVolts) between 0 and 1 volts around the state sampling clock threshold. This allows you to avoid false sampling on noisy clock inputs where a differential clock input turns off and floats to zero volts.

If you set Hysteresis to On, then in the event of your clock input going to zero volts, the problems related to the false clock signals due to noise on the clock and clock edges being too close together are eliminated. With the Hysteresis feature set to On, the Analyzer recognizes the clock signals only when the clock differentially exceeds the Hysteresis value that you specified thereby avoiding false clock signals.

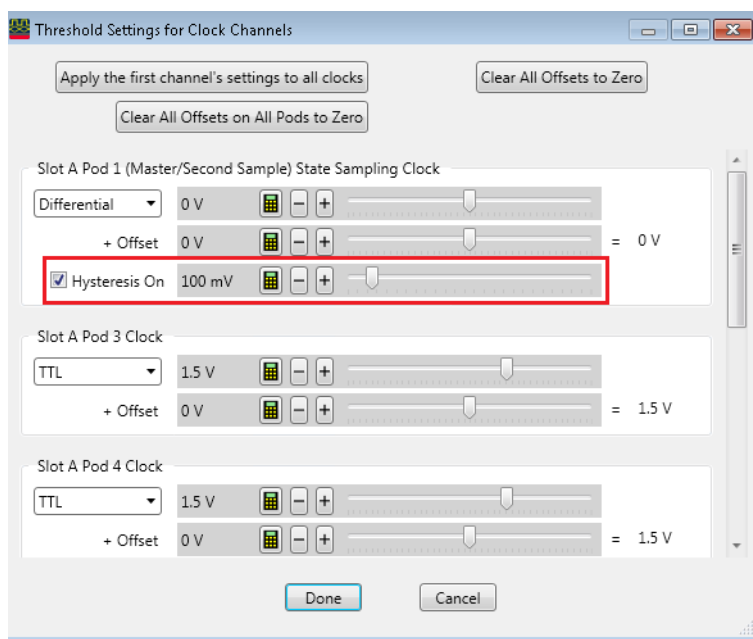
NOTE

The Clock hysteresis feature is available only for the U4164A and 16860-series logic analyzers.

- 1 Click the Clock Thresholds button displayed in the Buses/Signals tab of the Analyzer Setup dialog box.



- 2 The Threshold Settings for Clock Channels dialog box is displayed. You can set the Hysteresis feature On or Off using the Hysteresis On checkbox displayed for the state sampling clock, that is, the clock on Pod1.



- 3 Specify a value for Clock Hysteresis in the field next to the Hysteresis On checkbox. You can set a value between 0 to 1 V.
- 4 Click Done.

Setting up and Running Eyescans in Logic Analyzers

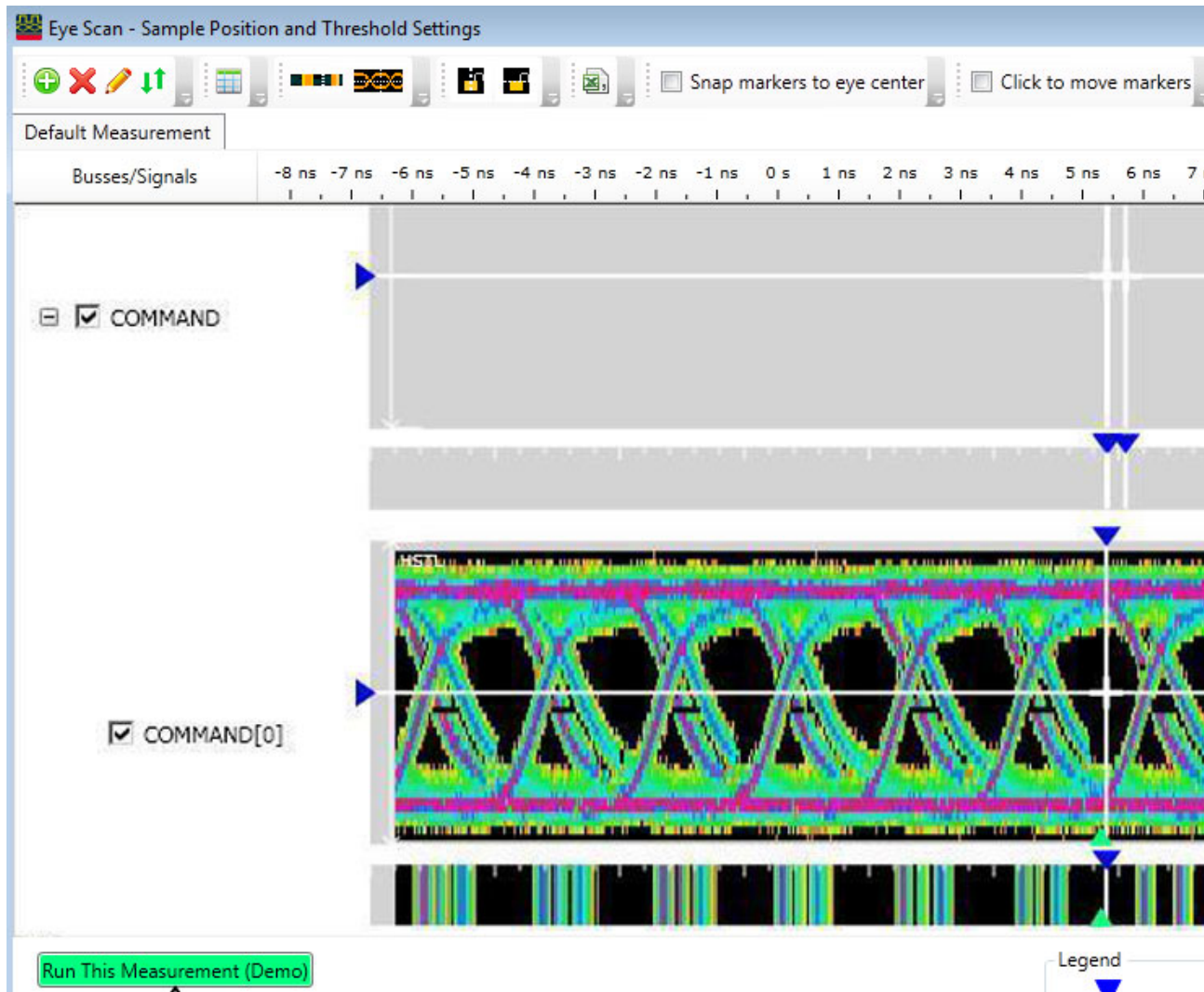
The U4154A/B logic analyzer allows you to set up and run eyescans for automatically adjusting the state mode sampling positions and threshold voltages of address and control signals so that data is sampled when it is valid. This topic describes how you can set up and run the eyescan measurements when using the U4154A/B Logic Analyzer.

For U4154A/B, you can set the sampling positions and threshold voltages for signals in an eyescan measurement and then run the measurement to allow U4154A/B logic analyzer to find out and suggest optimal sample positions and threshold voltages for these signals. You can then choose to set the sampling position and threshold voltages suggested in the eyescan run or manually adjust these settings based on the suggested values.

When you run an eyescan measurement, logic analyzer determines the threshold voltage that results in the widest possible data valid window by examining the signals from the DUT. Then logic analyzer determines the location of the data valid window in relation to the sampling clock, and suggests the ideal threshold voltage and sampling position.

Eyescan sets the sample position on individual channels of U4154A/B and therefore, is an easy way to get the smallest possible logic analyzer setup/hold window to accurately capture data.

You use the **Eye Scan - Sample Position and Threshold Settings** dialog box to set up and run eyescan measurements for U4154A/B channels. The following screen displays this dialog box with five predefined eyescan measurements - Clock, Chip Select, Command and Address, Data Read, and Data Write. The eyescan results of the Command and Address eye scan measurement is shown. The eyescan diagram is displayed for channel 1 of pod 1.1 mapped to a signal named COMMAND.



Creating an eyescan measurement

In an eyescan measurement, you:

- select buses/signals on which you want to run the eyescan measurement. For these buses/signals, you can select either all or specific U4154A/B channels mapped to these buses/signals.
- set the sample positions and thresholds for the selected signals.
- select the eyescan parameters including the type of eyescan run.

Before creating an eyescan measurement

Ensure that you have:



- 1 Connected the U4154A/B logic analyzer channels to the appropriate signals on the DUT to probe the DUT.
- 2 Assigned bus/signal names and mapped the names to the U4154A/B channels that you used in step 1. You use the Buses/Signals tab of the Analyzer setup dialog box to do this.
- 3 If you already set the threshold voltages for channels in the Buses/Signals tab, then these settings will be displayed as the current threshold settings in the eyescan setup.

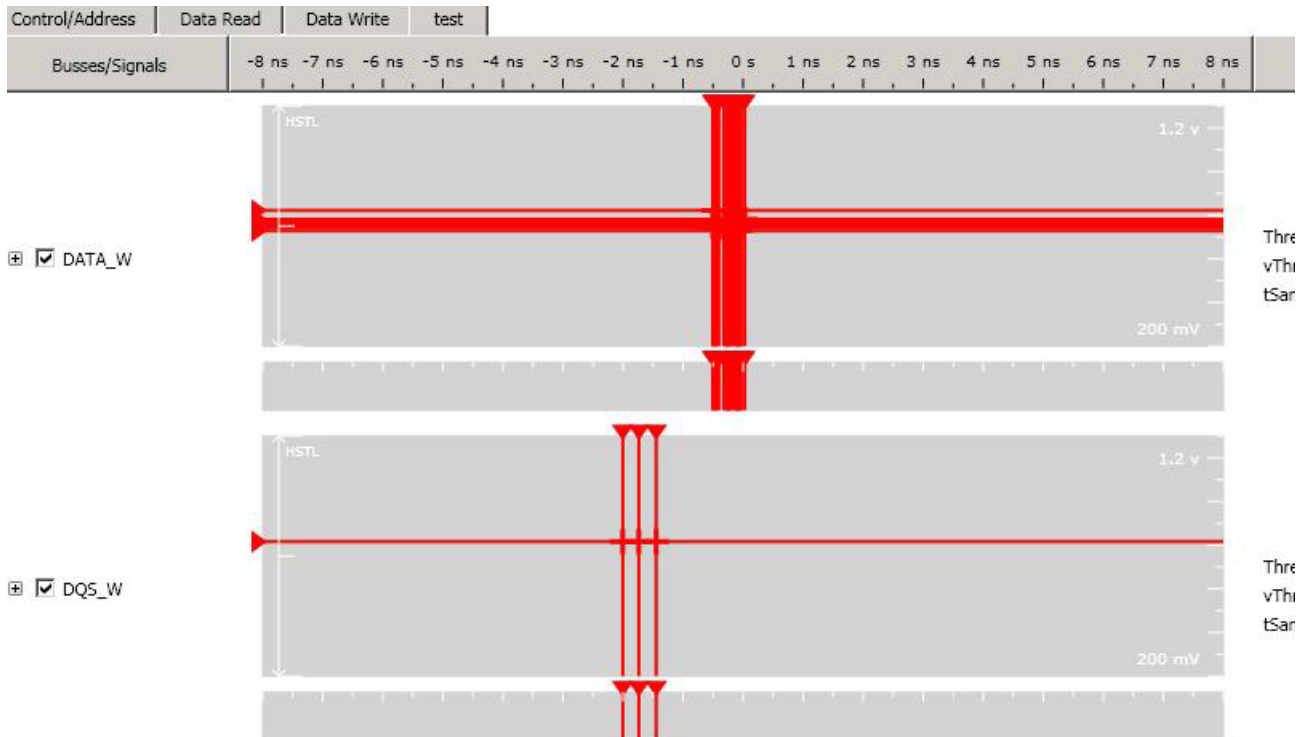
- 4 Selected the state (synchronous sampling) mode.
- 5 Set the state sampling clock options. (see Setting State Sampling Options in U4154A/B)

To create an eyescan measurement

- 1 Click the **Eye Scan: Sample Positions and Thresholds** button in the **Sampling** tab of the **Analyzer setup** dialog box.

The **Eye Scan - Sample Position and Threshold Settings** dialog box is displayed.

- 2 A **Default Measurement** is displayed with the currently mapped buses/signals and channels. For each channel, the current threshold voltage and sample position is displayed. You can either rename this measurement by right-clicking this measurement and selecting **Rename**. Or you can create a new measurement by clicking the  **Add New Measurement** toolbar button.
- 3 Click the tab of the new measurement that you created.
- 4 Assign signals/buses to this measurement so that eyescan is run on the assigned signals only. To do this:
 - a Click  **Assign Buses / Signals to Measurements** toolbar button.
 - b All the buses/signals that you created in the Analyzer setup dialog box are displayed. You can expand these to display the mapped U41554A channels.
 - c Select the checkbox displayed with a bus/signal on which you want to run the eyescan. This selects all the channels that are a part of the bus/signal. Alternatively, you can select individual channels from the different buses/signal names by selecting the checkbox displayed with a channel.
 - d Click **OK**.
- 5 All the signals that you selected are now displayed in the measurement tab. The current threshold type, threshold voltage, and sample position settings are also displayed for each of these signals. These settings are editable. If needed, you can edit these settings.



- 6 Click **Edit** to customize the eyescan run options.

- 7 Select the appropriate scan options and click **OK**.

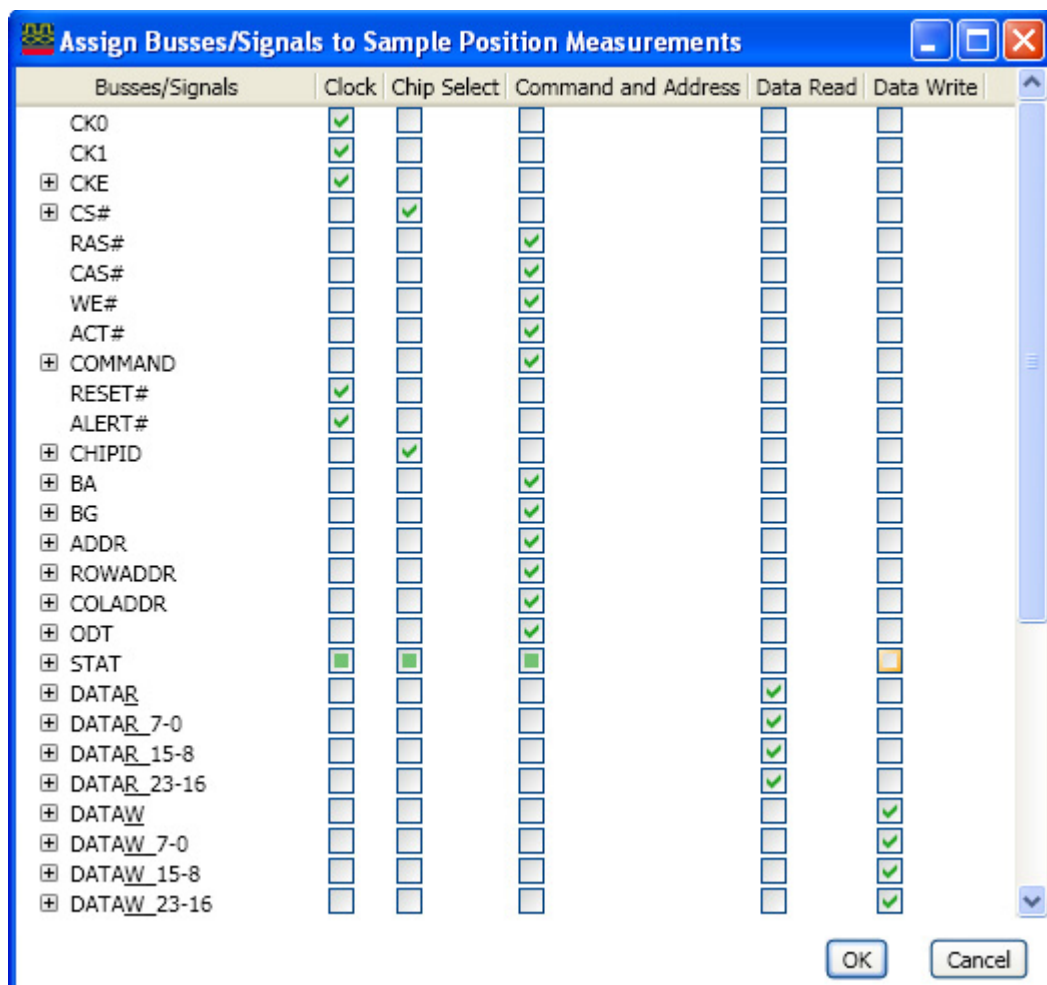
After you have created an eyescan measurement, you can edit the measurement to customize how you want the U4154A/B Logic Analyzer to run the eyescan measurement. You do this by clicking **Edit Current Measurement** button in the Eye Scan - Sample Position and Threshold Settings dialog box. Refer to the topic *Customizing an Eyescan Measurement* to know more.

Creating multiple eyescan measurements

You can also create multiple eyescan measurements to use different eye scan parameters for different signals. For instance, you can create a measurement for control signals to perform a simple time scan and another one for data read and write signals to do a full time and voltage scan.


NOTE

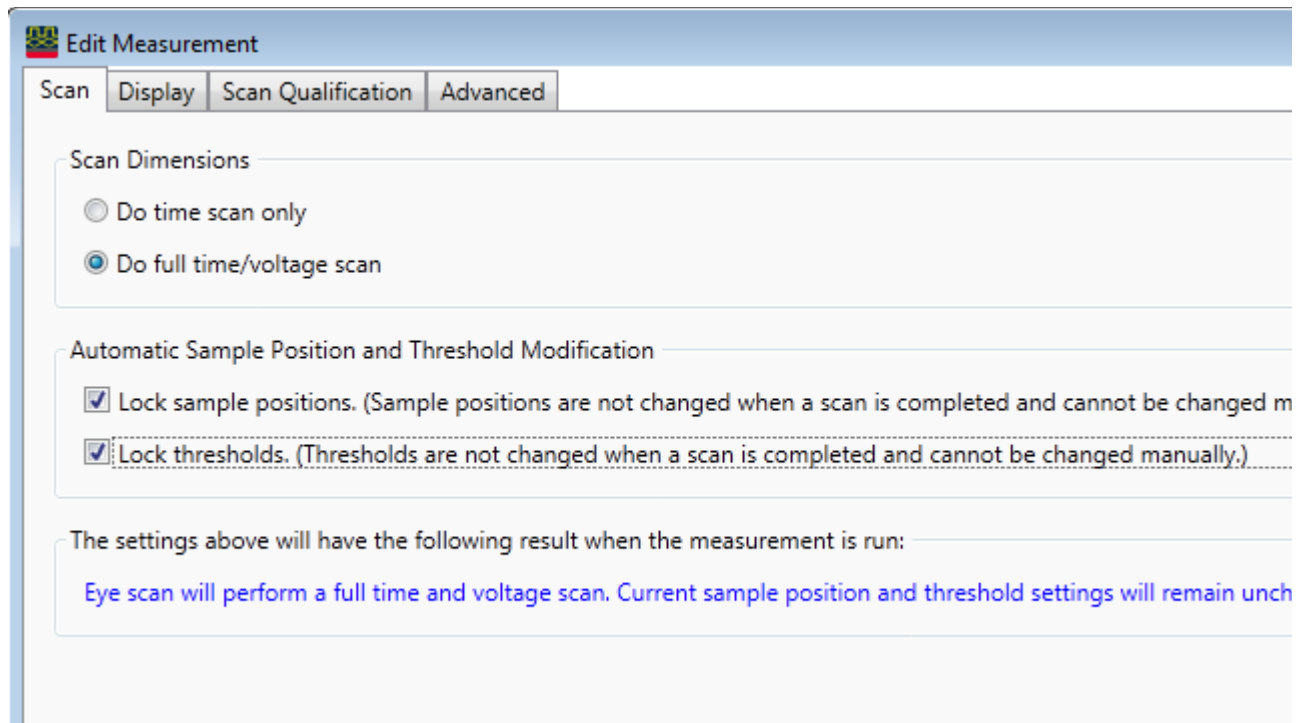
If you need to create multiple eye scan measurements, you can include a signal in only one of these measurements for setting the signal's sample position. This ensures that you get the eye scan results for a signal in only that measurement. If you assign a signal that already exists in a measurement to another measurement, then the signal is automatically removed from the earlier assigned measurement.



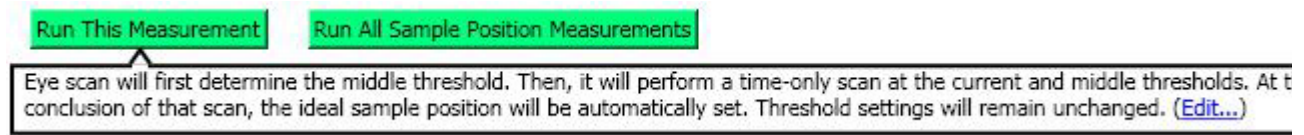
Customizing an Eyescan Measurement

Once you have created an eyescan measurement, you can configure various settings in the **Eye Scan - Sample Position and Threshold Settings** dialog box to customize how you want the U4154A/B Logic Analyzer to run the eyescan measurement. For instance, you can set eyescan parameters such as the type of eyescan run or the eyescan resolution.

To customize an eyescan measurement, you edit the measurement by clicking **Edit Current Measurement** toolbar button  in the Eye Scan - Sample Position and Threshold Settings dialog box. On clicking this button, the following dialog box is displayed with various configurable eyescan options organized in different tabs.



Based on the combination of edit options that you select to customize the eye scan, an explanation of the expected eyescan results is displayed in the **Eye Scan - Sample Position and Threshold Settings** dialog box as follows.



This topic describes some of the configurable options for eyescan.

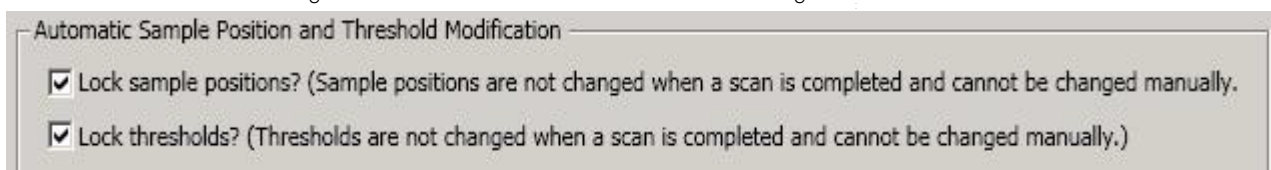
Types of eyescans:

- **Time scan only** - This eyescan option performs a full time scan at the middle threshold voltage and suggests the optimal sample position. It sets the threshold voltage and sample positions based on the optimal values depending on whether or not you selected to lock the current threshold and sample position settings.

- **Time and voltage scan** - This eyescan option finds the signal activity envelop and adjusts the threshold voltage to determine the optimal threshold voltage. Then it performs a full time scan at that threshold to suggest the optimal sample position. It sets the threshold voltage and sample positions based on the optimal values depending on whether or not you selected to lock the current threshold and sample position settings.

Locking the current settings

At times, you may want to run the eyescan to view the suggested optimal settings but do not want logic analyzer to automatically adjust your current settings based on the eyescan results. In such a situation, you can lock your current threshold and sample position settings to ensure that the eyescan run suggestions do not automatically alter your current settings. By locking the settings, you lock these for manual adjustments as well. If you do not lock these settings, then eye scan run automatically adjusts the current settings as per the suggested optimal settings. You can later choose to retain these adjusted settings or modify manually. The following screen displays these lock settings that are available in Edit Measurement dialog box.

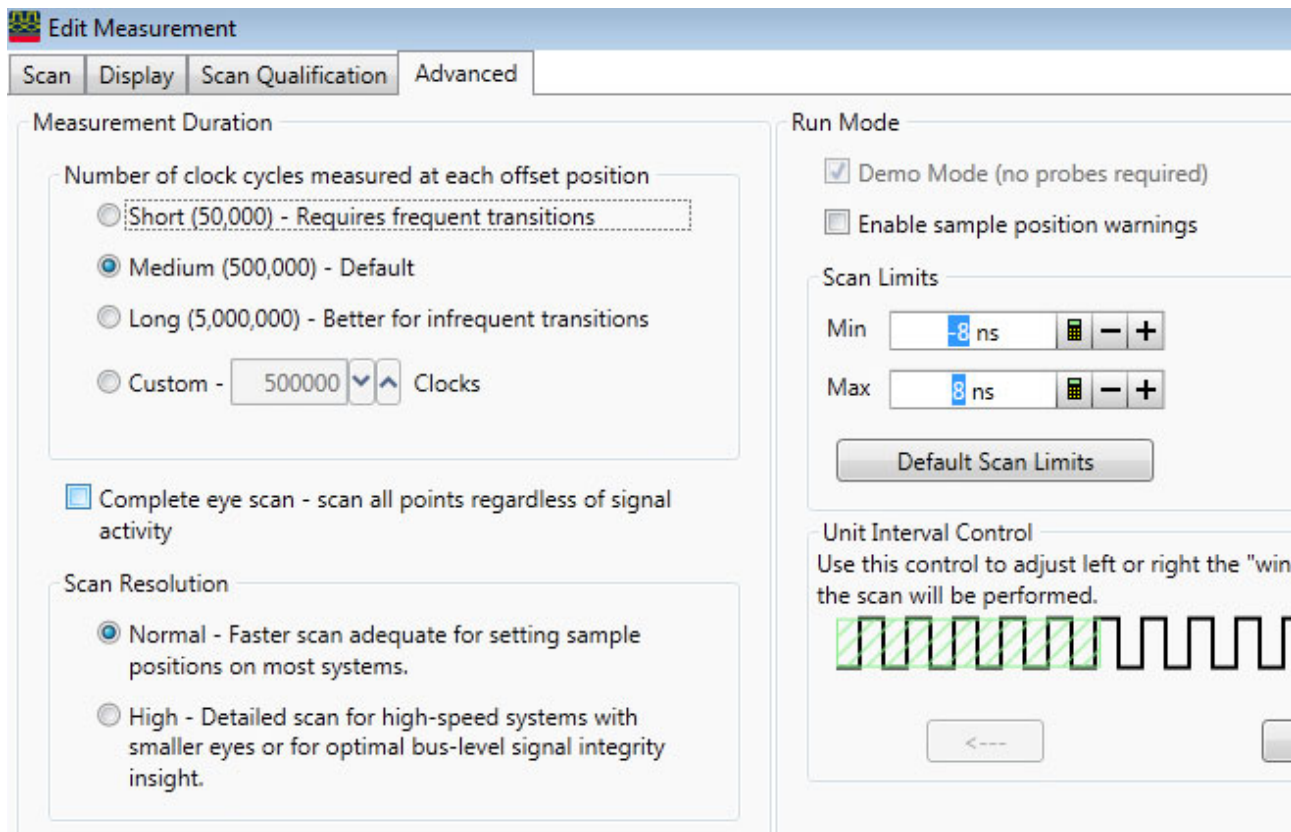


Setting eyescan resolution

You can select the eyescan resolution that you want for the resulting scans. If you have the 02G license option for U4154A/B module, then you can select either Normal or High resolution for the scans. However, if you have the 01G license option for the U4154A/B module, then only the Normal resolution option is available. The High resolution option is disabled for 01G license.

If you select the Normal option, you get a time resolution of 20 ps in scans. With the High resolution option, you get a time resolution of 4.8 ps in scans.

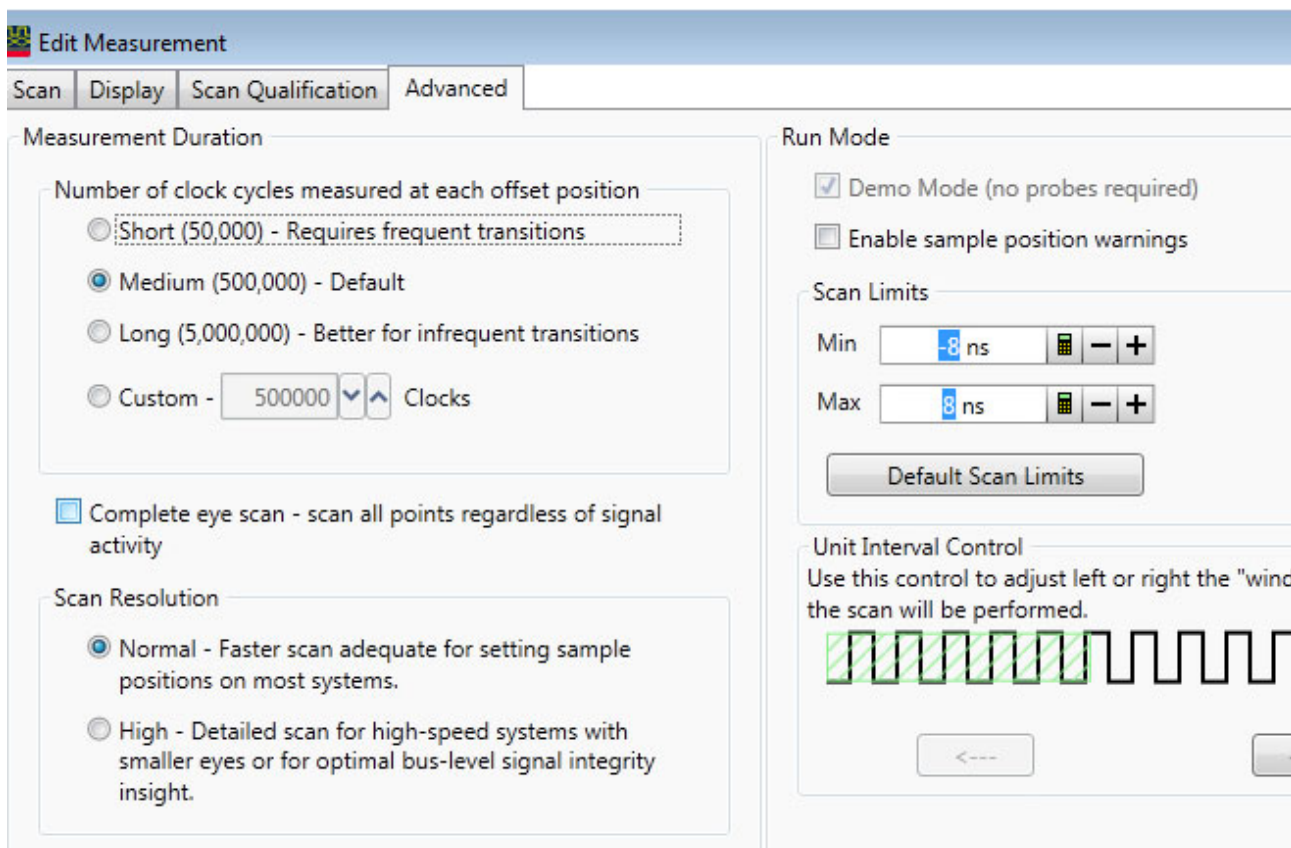
The following screen displays these two resolution options.



Configuring Unit Interval (UI) settings

There may be a situation when the eyes in which you want to set the sampling positions are outside of the adjustment range in the eyescan. In such a situation, you can configure the unit interval settings to move the signals left or right (in relation to the zero time) by Unit Interval (clock cycle) amounts. You can use the left and right arrow keys displayed in the **Unit Interface Controls** section to adjust the window on which the eyescan will be performed. The currently applicable scan window is represented by the green shaded area. Each mark to the left represents a whole clock cycle. Moving the slider to the left results in the data eyes moving left in the display after rerunning a scan.

After configuring the unit interval settings, you must rescan and then set the sampling positions again.



Running an Eyescan

After you have created an eyescan measurement and customized it to suit your specific requirements, you can run the eyescan measurement. This allows U4154A/B logic analyzer to determine and suggest optimal values for sample positions and thresholds.

If you created multiple eyescan measurements, you can either run these measurements separately or as a sequential batch. In a sequential batch, the measurements are run in the same order in which these are displayed in the tabbed list.

To run an eyescan measurement

- 1 Click the tab of the eyescan measurement that you want to run in the **Eye Scan - Sample Position and Threshold Settings** dialog box.
- 2 Click **Run this measurement**.

To run multiple eyescan measurements

- 1 Click **Run All Sample Position Measurements** in the **Eye Scan - Sample Position and Threshold Settings** dialog box.

Interpreting the eyescan results and eye diagrams

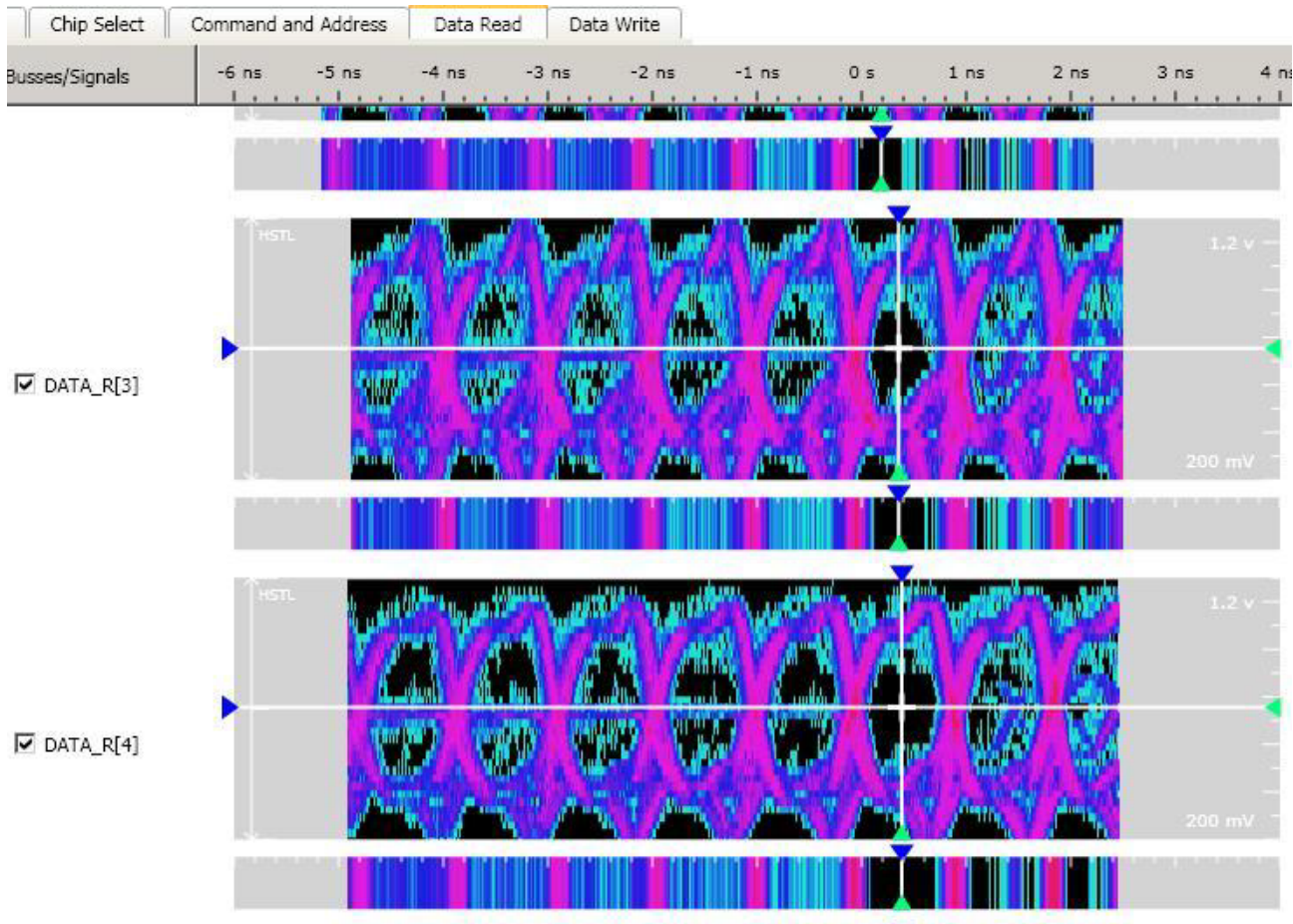
By performing full time and voltage scans, U4154A/B is able to give you a map of transitions detected in small windows of time and voltage. These scans are called eyescans.

The eyescan results show:

- Suggested sampling positions and threshold voltages (green triangles).

- The current sampling positions (vertical blue lines in stable regions, red lines in transitioning regions) and threshold voltages (horizontal blue lines in stable regions, red lines in transitioning regions).
- A digital "eye" diagram that represents many samples of data captured in relation to the sampling clock. The transitioning edges measured before and after the sampling clock result in a picture that is eye-shaped. Eye diagrams are used to display the measurement data.

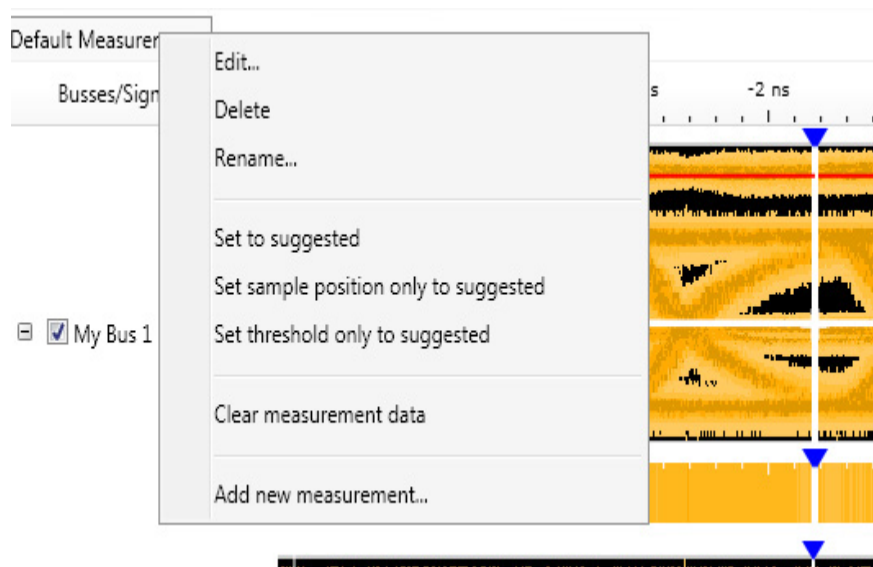
The following screen displays the results of a time and voltage eye scan run on the data read signals. The current settings have been automatically adjusted by the eye scan based on the optimal settings determined during the eye scan.



Adjusting sample positions and thresholds

Setting sample positions for all signals to their suggested positions

You can set the sample positions for all the signals included in a measurement to their suggested sample positions displayed in the eyescan results in a single click. To accomplish this, right-click the measurement and select the **Set sample position only to suggested** option.



You can also set the thresholds to their suggested threshold values by clicking the Set threshold only to suggested option. The Set to suggested option sets both the sample positions and thresholds to their suggested values.

Adjusting/moving the sample positions manually

After viewing the eyescan results, if needed, you can mark the ideal sample position for each signal by manually moving the sample position markers in the eyescan. You do this:

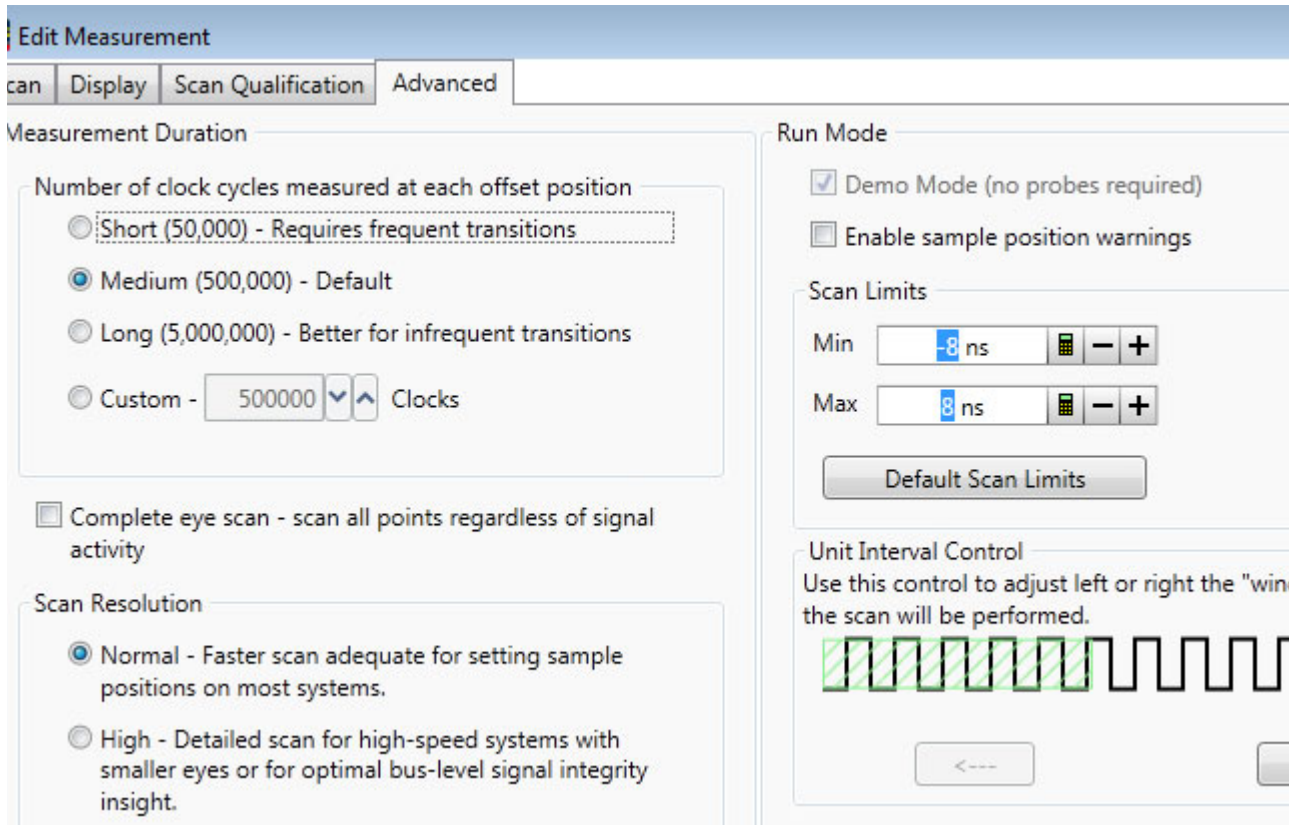
- either by dragging and dropping the sample position markers to the required positions.
- or by clicking on the required position. This automatically moves the marker to the clicked position. For this, you need to select the **Click to move markers** checkbox. By default, this checkbox is deselected.

While adjusting the sample positions, you may want to deselect the **Snap marker to eye center** checkbox to get greater control over the sample position placements. On deselecting this checkbox, the sample position markers are set exactly at the same position at which you dropped the marker or clicked. When this checkbox is selected, the sample position marker is set at the center of the eye irrespective of where you clicked or dropped the marker while moving it.



If the eyes in which you want to set the sampling positions are outside of the adjustment range in the eyescan, you can configure the unit interval settings to move the signals left or right (in relation to the zero time) by Unit Interval (clock cycle) amounts. You can use the left and right arrow keys displayed in the Unit Interface Controls section to adjust the window on which the eyescan will be performed. The currently applicable scan window is represented by the green shaded area. Each mark to the left represents a whole clock cycle. Moving the slider to the left results in the data eyes moving left in the display after rerunning a scan.

After configuring the unit interval settings, you must rescan and then set the sampling positions again.



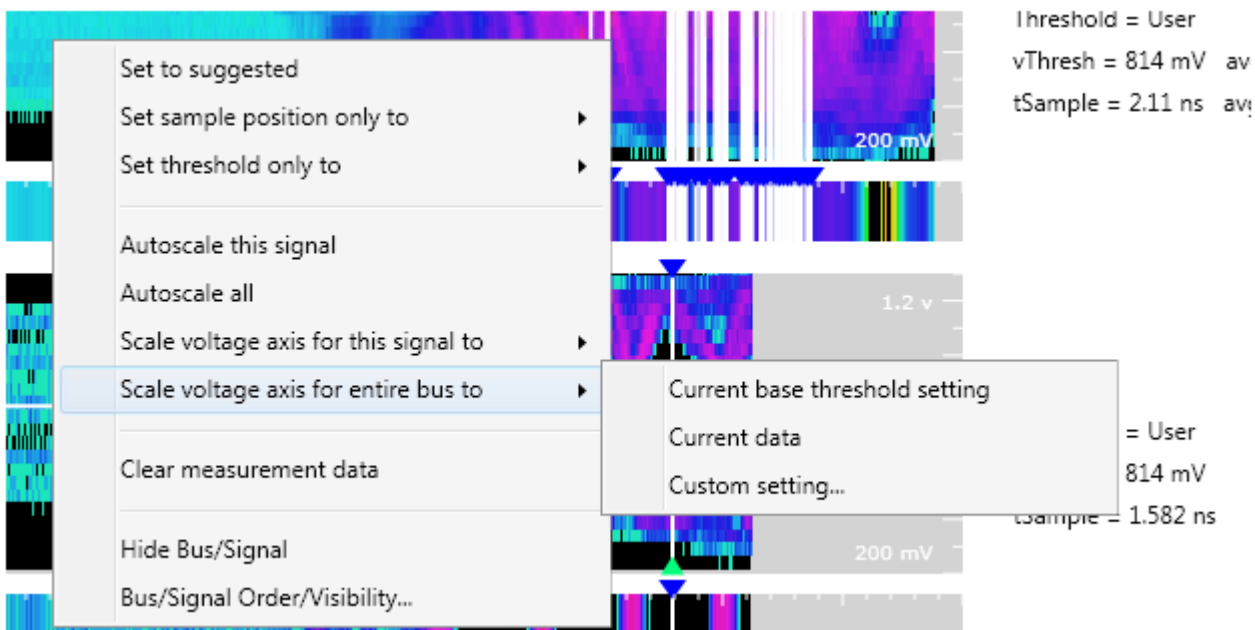
Scaling voltage axis for a signal or an entire bus

After you have run the eyescan measurement and acquired the eyescan data, you can scale the voltage axis for individual signals or an entire bus in that eyescan. When you scale the voltage axis for a bus, each child signal in that bus is scaled to the same value that you select.

Note: The scaling options are available only on a full time/voltage eyescan diagram. On a Time Only eyescan diagram, these options are not displayed as the eyescan data is time-only. Further, if you generated a full time/voltage eyescan diagram from a time-only scan, the scaling options will be displayed for use but not meaningful as the data is still time-only. Scaling provides meaningful results on a full time/voltage eyescan diagram that has been acquired by running a full time and voltage scan.

To scale the voltage axis

- 1 In the displayed eyescan diagram, navigate to the signal or the bus for which you want to scale the voltage axis.
- 2 Right-click the signal or the bus. A context menu is displayed.




- 3 To scale an entire bus, select the **Scale voltage axis for entire bus to** option. To scale an individual signal, select the **Scale voltage axis for this signal to** option.
- 4 A submenu is displayed with three scaling options that represent the value to be used to scale the signal(s). Select a menu option that suits your scaling requirement. You can select from:
 - **Current base threshold setting** - Selecting this option scales the signal(s) using the base threshold voltage setting with which you run the eyescan measurement. This threshold setting is displayed in the Threshold column on the right of the diagram.
 - **Current data** - Selecting this option scales the signal(s) using the voltage setting currently displayed in the acquired eyescan data.
 - **Custom setting** - Selecting this option scales the signal(s) using the minimum and maximum voltage values that you specify when you select this menu option.

The selected signal or the signals in the entire bus are then scaled to the selected value.

Exporting eyescan data to a .CSV file

After you have run the eyescan measurement, you can export the displayed eyescan data to a specified .csv file.

To export eyescan data to a .csv file

- 1 Click the  **Export Measurement Data to CSV File** toolbar button in the Eye Scan - Sample Position and Threshold Settings dialog box.
The **Save As** dialog box is displayed.
- 2 Specify the name and location of the .csv file to which you want to export the displayed eyescan data and then click **Save**.

Contents of the .csv file

In the CSV file, the eyescan data is exported for each signal that you included in the eyescan measurement. The following screen displays a sample .csv file in Excel with eyescan data for the MyBus1[0] signal.

A	B	C	D	E	F	G	H	I
Eye scan data output file.								
Signal_info	My Bus 1[0]	My Bus 1	0	Pod 2.1	0			
Scan_start_date	8/24/2011							
Scan_start_time	2:03:33 PM							
Threshold_scan	True							
tMin	-5.76E-09							
tMax	1.61E-09							
tRes	4.80E-12							
vMin	0.125976							
vMax	1.35424785							
vRes	0.03149415							
nCols	1537							
nRows	40							
eye_scan_data								
0	0	0	0	0	0	0	0	0
2.40E-05	2.40E-05	1.80E-05	1.80E-05	8.00E-06	8.00E-06	1.20E-05	1.40E-05	8.00E-06
4.80E-05	6.80E-05	6.80E-05	6.20E-05	5.20E-05	8.60E-05	8.00E-05	8.60E-05	9.60E-05
0.000759991	0.000611993	0.000682	0.000848	0.000548	0.000492	0.00058	0.000946	0.00076
0.001461982	0.002027976	0.001462	0.00281	0.00281	0.00226	0.00163	0.00252	0.001462

As displayed in the above screen, the .csv file contains information in the following two sections for each of the mapped bus/signal and channel that you included in the eyescan measurement.

- **General information on signal and scan** - This section displays:
 - **Signal_Info** - The name of the signal and mapped channel followed by the name of the parent bus name, the numerical index of the signal within the bus, the U4154A/B pod which contains the signal, and the bit index within that pod.
 - **Scan_start_date** - The date at which the eye scan was started.
 - **Scan_start_time** - The time at which the eye scan was started.
 - **Threshold_scan** - Displays True if a full time/voltage scan was performed and False if only a time scan was performed.
 - **tMin, tMax, and tRes** - Represent the minimum time, maximum time, and time resolution (step size).
 - **vMin, vMax, and vRes** - Represent the minimum voltage, maximum voltage, and voltage resolution.
 - **nCols and nRows** - The number of columns and rows of eyescan data displayed in the .csv file for the signal.
- **Eyescan data (eye_scan_data)** - In this section, the eyescan data for the signal is displayed for various voltage/time points. Rows in this section represent voltage points and columns represent time points in the scan. The first row in this section shows the scan data for the minimum voltage in the scan. The last row shows the scan data for the maximum voltage in the scan. The first cell in each of these rows is the earliest time in the scan.

Each cell in the eyescan data section has a value that represents a hit probability (signal activity) for the time/voltage point that the row and column combination of the cell represents. The cells can have the following values:

 - 0 - This value indicates that there was no signal activity at that voltage and time combination.
 - -1 - This value indicates that no scan data was taken at that voltage and therefore all other values displayed in that row should be ignored. In the .csv file generated for a full eye scan, none of the rows in the eyescan data section start with -1 because all voltage values have been measured.
 - 1 - This value indicates that there was a hit (signal activity) at every clock (a rare situation).

You can determine the voltage of a row in the eyescan data section by using the following formula:

$$vRow = vMin + (rowNum * vRes)$$

where rowNum is the 0-based index of the row.

You can determine the time for a column in the eyescan data section by using the following formula:

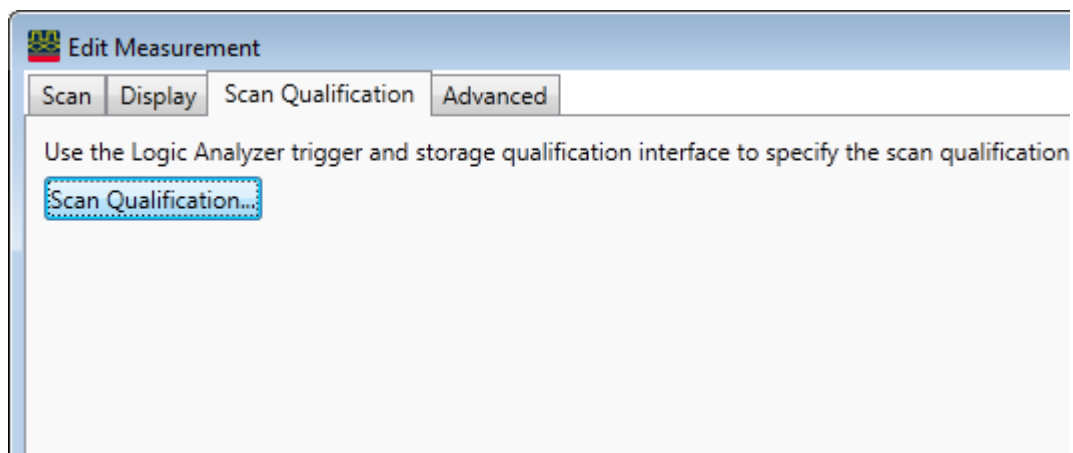
$$tCol = tMin + (colNum * tRes)$$

Modifying General or Target-specific Scan Qualification

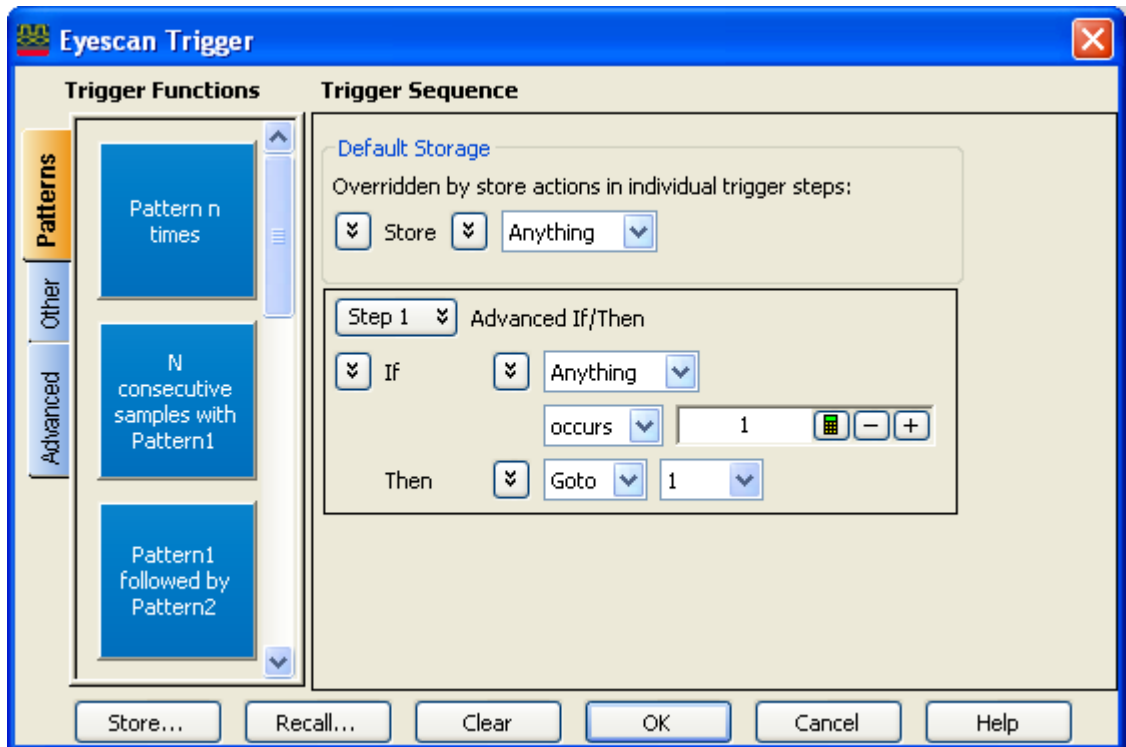
You use the **Scan Qualification** tab in the **Edit Measurement** dialog box invoked from the Eye Scan - Sample Positions and Threshold Settings dialog box to modify the scan qualification for general as well as target-specific scans such as DDR scans. This tab displays different fields based on whether you are modifying scan qualification for a general or a target-specific scan. This topic describes how to use this tab to modify general or target-specific scan qualification.

General Scan Qualification

For generating general trigger specifications when using the U4154 module, the Scan Qualification tab displays only the **Scan Qualification...** button as displayed below.



When you click the **Scan Qualification** button, the **Eyescan Trigger** dialog box is displayed. Using this dialog box, you can specify a sequence of trigger conditions for a generalized eyescan measurement. These trigger conditions allow you to control when U4154A/B logic analyzer takes samples that are used in that eyescan for determining the optimal threshold and sample positions.



The eyescan trigger feature in U4154A/B is similar to the general trigger function of logic analyzer (for specifying when to capture data from DUT). However, an eyescan trigger has only one type of action (Goto <trigger step>) when the trigger condition is met.

DDR/LPDDR-specific Scan Qualification

Once you have created an initial DDR/LPDDR setup and set the initial sample positions using the DDR Setup Assistant tool, there may be situations when you want to modify the DDR/LPDDR-specific scan qualification parameters before running subsequent eyescans for DDR/LPDDR signals. In such situations, you use the **Scan Qualification** tab of the **Edit Measurement** dialog box invoked from the Eye Scan - Sample Positions and Threshold Settings dialog box.

In a general eyescan usage scenario, the **Scan Qualification** tab displays a single button to modify a general trigger condition for scan qualification. However, when you load a DDR/LPDDR specific configuration file or an already saved DDR setup .ala file in the Logic Analyzer GUI, this tab displays DDR/LPDDR-specific scan qualification fields. By default, these fields display the values that you set while creating the initial DDR setup using the DDR Setup Assistant tool. You can modify these default values using this tab.

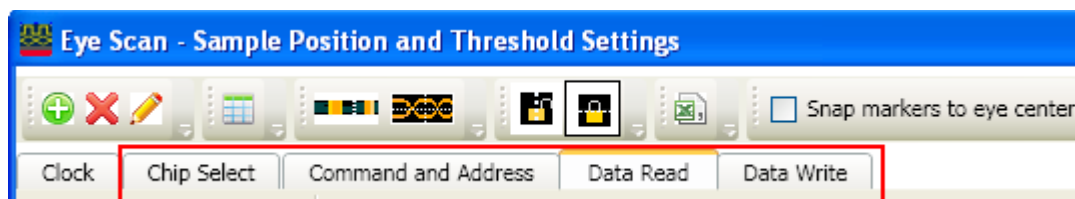
When you confirm and save the DDR-specific scan qualification settings, these settings are used to create a DDR trigger specification. The eyescan feature of the U4154A/B module then uses this trigger specification while performing subsequent eyescan runs to find DDR data signal eyes and set sample positions for DDR signals.

NOTE

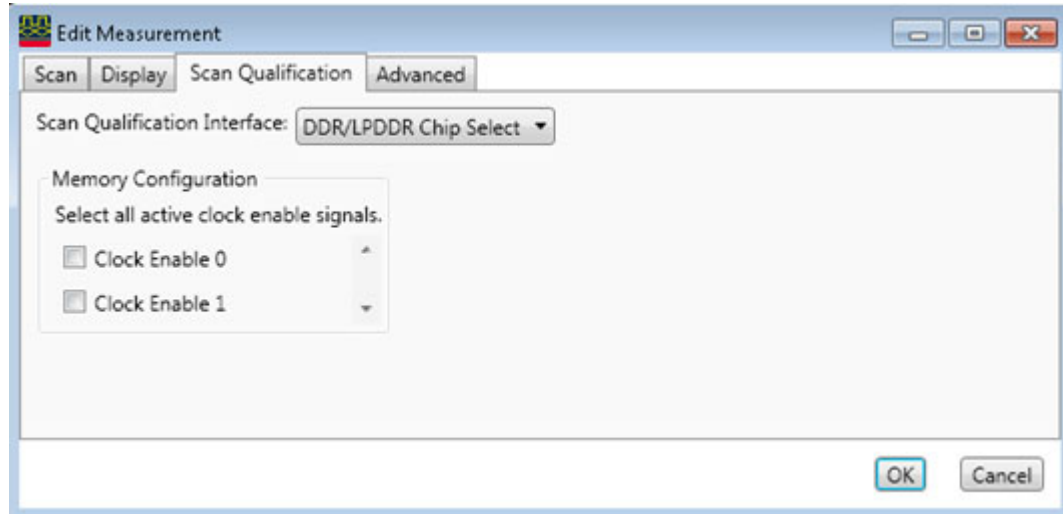
The Scan Qualification tab is used to set/modify the scan qualification only when you are using the U4154A/B module for capturing DDR data.

To modify the DDR-specific scan qualification

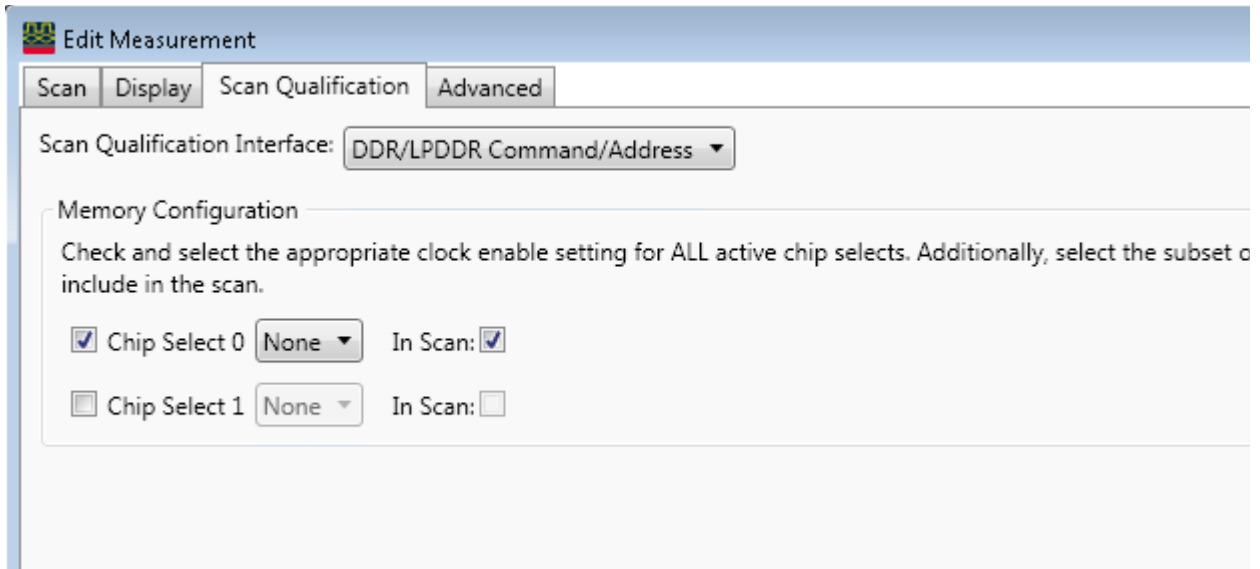
- 1 In the Keysight Logic Analyzer GUI, load the required DDR/LPDDR-specific configuration file for U4154A/B. (The default DDR/LPDDR configuration files are available if you have installed the DDR packages, *Keysight Bus Decoder for DDR2 and DDR3* and *Keysight Bus Decoder for LPDDR and LPDDR2*. Also, you should have the appropriate software license to use these software packages.)
OR
Open the Logic Analyzer configuration (.ala) file in which you saved the DDR setup (probes, module, tools, and windows) that you created using the DDR Setup Assistant tool.
- 2 Click the **Sampling Setup** link of the module displayed in the DDR setup.
The **Sampling** tab is displayed.
- 3 Ensure that the **State - Synchronous sampling** option is selected in the **Sampling** tab.
- 4 Click the **Eye Scan: Sample Positions and Thresholds...** button.
- 5 In the **Eye Scan - Sample Position and Threshold Settings** dialog box, select the tab for **Chip Select**, **Command and Address**, **Data Read** or **Data Write** to modify the scan qualification for the appropriate signals.



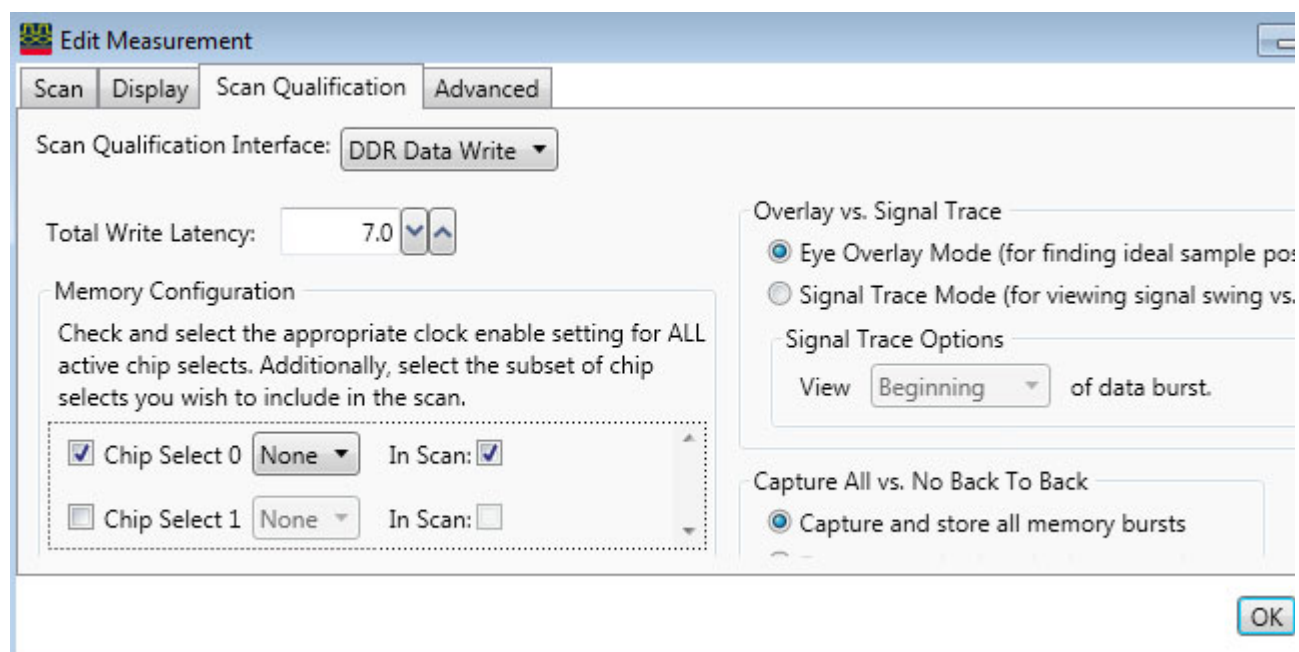
- 6 Click the **Edit Current Measurement** toolbar button.
 - 7 Click the **Scan Qualification** tab.
 - 8 Based on the signals tab you selected in step 5, the options are displayed in the **Scan qualification interface** listbox. Select one of the following options from this listbox:
 - DDR/LPDDR Chip Select option - to modify the scan qualification for chip select signals.
 - DDR/LPDDR Command and Address option - to modify the scan qualification for command and address signals.
 - DDR Data Read option - to modify the scan qualification for data read signals.
 - DDR Data Write option - to modify the scan qualification for data write signals.
- On selecting the **DDR/LPDDR Chip Select** option in the previous step, the following fields are displayed for chip select scan qualification:



- 9 For the DDR/LPDDR Chip Select signals, make the appropriate selections based on the active clock enable signals.
- On selecting the **DDR/LPDDR Command and Address** option in the step 8, the following fields are displayed for scan qualification:



- 10 For the DDR/LPDDR Command and Address signals, make the appropriate Memory Configuration selections based on the chip selects used in the DUT.
- On selecting the **DDR/LPDDR Data Read** or **Data Write** interface in step 8, the following fields are displayed for scan qualification:



- 11 For DDR Data Read or Write signals:
- The DUT's **Total Read / Write Latency** value that you set up in the initial DDR setup is displayed. If needed, modify this value. To find an appropriate value of Total Read/Write Latency, you can set up the Mode Register Settings (MRS) trigger, take a trace, and scan it for latency values. Refer to the DDR Setup Assistant online help to know more on how to find latency values.

NOTE

The latency values specified here represent the total latency for your system and therefore should include parameters that affect total latency. For instance, if your system has Additive Latency (AL), then you must include it in the Total Latency values. Similarly, if tDQSS or tDQSK parameters are greater than one full clock cycle, then you must add these values (rounded to the nearest integer) in the Total Latency values.

- In the **Memory Configuration** section, choose the appropriate **Chip Select(s)** for the memory path being traced.
- In the **Burst Length** field, specify the number of words read or written for each read/write command. To find appropriate value of Burst Length for the DUT, you can set up the Mode Register Settings (MRS) trigger, take a trace, and scan it for burst length. Refer to the DDR Setup Assistant online help to know how to find burst length.
- Select the appropriate scan mode:
 - Eye Overlay Mode** - Use this mode when setting sample positions for DDR data signals. In this mode, the eyescan feature of U4154A/B overlays the eyes for each bit in a burst in order to show a composite eye, without tri-state or noise, that helps you locate the best sample position.

- **Signal Trace Mode** - Use this mode for viewing signal swing vs. time. Although sample positions can be checked and modified in this mode, it is generally not recommended. In this mode, the eyescan feature of U4154A/B overlays complete bursts in order to provide additional qualitative insight into the signal integrity on the DDR system at the logic analyzer probe location. In this mode, you can view how the bits in a complete burst compare to one another. When you select the Signal Trace Mode, the following three options are provided to you to select the data burst area to view:
 - **Beginning** – Scans are made at the beginning of a data burst.
 - **Middle** – Scans are made at the middle of a data burst.
 - **End** – Scans are made at the end of a data burst.
- 12 Select the **Capture and store all memory bursts** radio button to capture and store back to back memory bursts.
 - 13 Select the **Do not store back-to-back memory bursts** radio button to eliminate the "double eye" effect in scan results.
 - 14 Click **OK** to confirm the settings.

You can save the modified scan qualification settings in the Logic Analyzer .ala or .xml configuration file.

Specifications and Characteristics

Describes the specifications, characteristics, and requirements of supported logic analyzers and logic analysis systems.

- U4164A Logic Analyzer Specifications and Characteristics (see [page 587](#))
- U4154A Logic Analyzer Specifications and Characteristics (see [page 592](#))
- U4154B Logic Analyzer Specifications and Characteristics (see [page 596](#))
- 16850-Series Logic Analyzer Specifications and Characteristics (see [page 600](#))
- 16860-Series Logic Analyzer Specifications and Characteristics (see [page 604](#))

- See Also
- What is a Specification (see [page 611](#))
 - What is a Characteristic (see [page 611](#))

U4164A Logic Analyzer Specifications and Characteristics

Describes the specifications and characteristics of the U4164A logic analyzer.

NOTE

Complete and up-to-date specifications and characteristics of the U4164A logic analyzer are available in the U4164A Logic Analyzer Data Sheet (5992-1057EN) on the Keysight web site.

NOTE

Specification (spec): Represents warranted performance of a calibrated instrument that has been stored for a minimum of 2 hours within the operating temperature range, unless otherwise stated, and after a 45 minute warm-up period. The specifications include measurement uncertainty.

Typical (typ): Represents characteristic performance, which 80% of the instruments manufactured will meet. This data is not warranted, does not include measurement uncertainty, and is valid only at room temperature (approximately 25 °C).

Nominal (nom): The expected mean or average performance, or an attribute whose performance is by design, such as the 50 O connector. This data is not warranted and is measured at room temperature (approximately 25 °C).

Measured (meas): An attribute measured during the design phase for purposes of communicating expected performance, such as amplitude drift vs. time. This data is not warranted and is measured at room temperature (approximately 25 °C).

- Module Channel Counts (see [page 588](#))
- State (Synchronous) Analysis Mode (see [page 588](#))
- Timing (Asynchronous) Analysis Modes (see [page 590](#))
- Timing Zoom (Asynchronous) Analysis Modes (see [page 590](#))
- Trigger Characteristics (see [page 590](#))
- General (see [page 591](#))
- Logic and Protocol Analyzer Software Compatibility (see [page 592](#))
- Power Requirements (see [page 592](#))
- Chassis Compatibility (see [page 591](#))
- Environmental Characteristics (see [page 592](#))

Module Channel Counts

Feature	Description
Number of Channels (nom)	136 in one U4164A 272 in two U4164As 408 in three U4164As
Maximum channels on single time base and trigger:	408

State (Synchronous) Analysis Mode

Feature	For Single Cock	For Multiple Clocks
Maximum state data rate Option 02G, 2.5 GHz state mode (spec)	2.5 Gb/s on 136 channels per U4164A, clocking on either or both edges of clock (spec) 4.0 Gb/s on 68 channels per U4164A, clocking on either edge of the clock (typ) Suitable for DDR/LPDDR memory systems running up to 4.0Gb/s	Not Supported
Maximum state data rate standard -01G, 1.4 GHz state mode (spec)	2.5 Gb/s on 136 channels per U4164A, clocking on both edges of clock (spec) 1.4 Gb/s on 136 channels per U4164A, clocking on either edge of the clock (spec) 2.8 Gb/s on 68 channels per U4164A, clocking on either or both edges of the clock (typ) Suitable for DDR/LPDDR memory systems running up to 2.5Gb/s	Not Supported
Maximum state data rate Option -700, 700 MHz state mode (spec)	1.4 Gb/s on 136 channels per U4164A, clocking on both edges of the clock (spec) 700 Mb/s on 136 channels per U4164A, clocking on either edge of the clock (spec) 1.4 Gb/s on 68 channels per U4164A, clocking on either or both edges of the clock (typ) Suitable for DDR/LPDDR memory systems running up to 1.4Gb/s	Captures a 350MHz signal on either edge of the clock up to 350 MSP/s (spec) Captures a 350MHz signal on both edges of the clock up to 700 MSP/s (spec)
Maximum state data rate Option -350, 350 MHz state mode (spec)	700 Mb/s on 136 channels per U4164A, clocking on both edges of the clock (spec) 350 Mb/s on 136 channels per U4164A, clocking on either edge of the clock (spec) 700 Mb/s on 68 channels per U4164A, clocking on either or both edges of the clock (typ) Suitable for DDR/LPDDR memory systems running up to 700Mb/s	Captures a 350MHz signal on either edge of the clock up to 350 MSP/s (spec) Captures a 350MHz signal on both edges of the clock up to 700 MSP/s (spec)
Maximum state clock frequency (typ)	2.5 GHz Option -02G 1.4 GHz -01G 700 MHz Option -700 350 MHz standard (base)	350 MHz
Minimum state clock frequency (typ) [†]	12.5 MHz (single edge) 6.25 MHz (both edges)	0 MSps
Sample position adjustment resolution (typ)	5 ps or 20 ps	80 ps typical
Sample position adjustment accuracy (typ)	± 150 ps	
Minimum data valid window (typ)	100 ps	500 ps
Minimum setup time (typ)	50 ps	250 ps

Feature	For Single Cock	For Multiple Clocks
Minimum hold time (typ)	50 ps	250 ps
Minimum eye height (typ)	100 mV	
Sample position adjustment range (typ)	7 ns	12 ns (typical)
Minimum state clock pulse width (typ)	200 ps	Single edge: 250 ps
Number of clocks (nom)	1	Up to 4 clocks (in an Ored combination). Master and slave clocks are counted separately.
Minimum time between active clock edges (typ)*	400 ps	2.86 ns (for 350 MHz state speed) 1.43 ns (for 700 MHz state speed)
Maximum time between active clock edges (typ)*	80 ns	
Number of clock qualifiers	4 (pods 2, 3, 4 and 5 on clocking module)	Up to two clock qualifiers (Pod 1, 2, 3) Note: Though there are three clock qualifiers displayed, you can use a maximum of only two qualifiers for a clock. The third qualifier is automatically set to "Don't Care" and can be used only as an Ored clock in the clock description.
Clock qualifier setup time	150 ps	250 ps
Clock qualifier hold time	150 ps	250 ps
Number of "RESET" clock qualifiers	1 (pod 7 of clocking module)	0
"RESET" clock qualifier setup time	2 ns	N/A
"RESET" clock qualifier hold time	0 ps	N/A
Minimum slave to master clock time	N/A	350 ps
Minimum master to slave clock time	N/A	150 ps
Minimum slave to slave clock time	N/A	1.43 ns
Time tag resolution (typ)	80 ps	80 ps
Maximum time count between stored states (typ)	66 days	66 days

* Clock can pause for up to 66 days once every 8 or more edges

Timing (Asynchronous) Analysis Modes

Feature	Description		
	Full-channel Mode	Half-channel Mode	Quarter-channel Mode
Maximum time between transitions (nom)	66 Days		
Minimum data pulse width (typ)	1 sample period + 200 ps		
Time interval accuracy (typ)	± (1 sample period + 400 ps + 0.01% of time interval reading)		
Maximum sample rate (nom)	2.5 GHz	5 GHz	10 GHz
Minimum sample period (nom)	400 ps	200 ps	100 ps
Pod usage (nom) 1 pod from each	All pods	Odd from each odd/even pod pair	Even signal inputs (0, 2, 4, 6, 8) from odd pod of each odd/even pair plus clock input

Timing Zoom (Asynchronous) sampling mode

Feature	Description
Timing Zoom sampling rate (nom)	12.5 GHz
Timing Zoom memory depth (nom)	256 K samples

Trigger Characteristics

Feature	For State (Single Clock) and Timing Modes	For State (Multiple Clocks)
Maximum trigger sequence speed (typ)	2500 MHz (400 ps) Option -02G 1400 MHz (714 ps) Option -01G 700 MHz (1.428 ns) Option -700 350 MHz (2.856 ns) standard (base) module	350 MHz (for 350 MHz state speed option) 700 MHz (for 700 MHz state speed option)
Trigger resources (nom)	16 patterns evaluated as =, !=, >, >=, <, <= 8 double-bounded ranges evaluated as in range, not in range 4 to 8 burst detectors 4 edge detectors in timing, 3 in transitional timing 1 occurrence counter persequence level 1 timer 4 flags 1 arm in Global counter not supported Event counter supported	16 patterns evaluated as =, !=, >, >=, <, <= 8 double-bounded ranges evaluated as in range, not in range Burst patterns not supported 4 edge detectors in timing, 3 in transitional timing 1 occurrence counter persequence level 3 timers 4 flags 1 arm in Global counters -2 Event counter not supported
Trigger resource Boolean conditions (nom)	Arbitrary Boolean combinations	
Trigger actions (nom)	Goto Trigger and fill memory Trigger and Goto Trigger, send e-mail, and fill memory	

Feature	For State (Single Clock) and Timing Modes	For State (Multiple Clocks)
Store qualification actions (nom)	Default (global) and per sequence level Store/don't store sample Turn on/off default storing	
Timer actions	Start from reset Stop and reset Pause Resume	
Flag actions	Set Clear Pulse set Pulse clear	
Maximum trigger sequence levels (nom)	8	16
Trigger sequence level branching (nom)	Arbitrary 4-way if/then/else	
Trigger position (nom)	Start, center, end, or user-defined	
Maximum occurrence counter (nom)	999,999,999	
Maximum pattern width (nom)	128 bits – single label 408 bits – AND of multiple labels across three-card set	
Maximum range width (nom)	64 bits	
Timer range (nom)	100 ns to 27 hours (in timing modes) 200 * state clock period to 27 hours (in state mode)	
Timer resolution (nom)	5 ns	
Timer accuracy (typ)	± (5 ns +0.01%) (in timing modes) ± (8 * state clock period +2 ns +0.01%) (in state mode)	
Timer reset latency (nom)	40 ns (in timing modes) 80 * state clock period (in state mode)	

General

Feature	Description
Input signal amplitude Vamp _{td} (typ)	=> 350 mV
Supported signal types	Single-ended and differential
Voltage threshold (typ)	-5 V to 5 V –4 V to +4 V when in dual sample or quad sample state modes
Threshold accuracy (typ)	± (30 mV + 1% of setting)
Minimum threshold resolution (typ)	2 mV
Threshold setting granularity	By channel

Chassis Compatibility

Keysight AXIe M9502A 2-slot or M9505A 5-slot chassis

Logic and Protocol Analyzer Software Compatibility

Version 6.2 or greater

Power Requirements

All necessary power is supplied by the backplane connector of the Keysight AXIe chassis.

Environmental Characteristics

Indoor use only.

Operating Environment

Feature	Description
Temperature (nom)	0°C to 40°C (+32°F to 104°F)
Humidity (nom)	0 to 80% relative humidity at 40°C (+104°F)
Altitude	0 to 3,000 m (10,000 ft)
Vibration	Random vibration 5-500 Hz, 10 minutes per axis, approximately 0.2 g rms

Non-operating Environment

Feature	Description
Temperature (nom)	-40°C to +75°C (-40°F to +167°F).
Humidity (nom)	0 to 90% relative humidity at 65°C (149°F)
Altitude	0 to 15,300 m (50,000 ft)
Vibration (in shipping carton)	Random vibration 5 to 500 Hz, 10 minutes per axis, approximately 2.41 g rms; and swept sine resonant search, 5 to 500 Hz, 0.50 g (0-peak), 5-minute resonant dwell at 4 resonances per axis.

Weight 2.34 kg

- See Also
- What is a Specification (see [page 611](#))
 - What is a Characteristic (see [page 611](#))

U4154A Logic Analyzer Specifications and Characteristics

Describes the specifications and characteristics of the U4154A logic analyzer.

NOTE

Complete specifications and characteristics of the U4154A logic analyzer are available in the [U4154A Logic Analyzer Data Sheet \(5990-7513EN\)](#) on the Keysight web site.

NOTE

Items marked with an asterisk (*) are specifications. All others are characteristics. "Typical" represents the average or median value of the parameter based on measurements from a significant number of units.

- Module Channel Counts (see [page 593](#))
- State (Synchronous) Analysis Mode (see [page 593](#))
- Timing (Asynchronous) Analysis Modes (see [page 593](#))

- Trigger Characteristics (see [page 594](#))
- Other (see [page 595](#))
- Power Requirements (see [page 595](#))
- Chassis Compatibility (see [page 595](#))
- Environmental Characteristics (see [page 595](#))

Module Channel Counts

Menu	Description
Channels per U4154A module (unused clock and clock ready inputs can be used as data channels):	136 channels State analysis: 128 data channels + 8 clock channels Timing (Full channel): 128 data channels + 8 clock channels Timing (Half channel): 64 data channels + 4 clock channels
Maximum channels on single time base and trigger:	272
Number of AXie chassis slots per card:	1
Number of U4154A modules that can be connected in an AXie chassis to form a multi-card set	2

State (Synchronous) Analysis Mode

Menu	Description
	2.5 Gb/s State data rate with 02G license option 1.4 Gb/s State data rate with 01G license option
Setup/hold window ¹	500 ps (350 ps typ)
S/H adjustment resolution	5 ps
Sample range	-3 ns to + 3 ns
Minimum state clock pulse width	200 ps
Number of clocks	1
Number of qualifiers	4
Time tag resolution	80 ps
Maximum time count between stored states	66 days
* Items marked with an asterisk (*) are specifications. All others are characteristics. "Typical" represents the average or median value of the parameter based on measurements from a significant number of units.	
¹ Dependant on probing system	

Timing (Asynchronous) Analysis Modes

Menu	Description
Sample rate on all channels	2500 MHz
Sample rate in 1/2 channel mode	5000 MHz
Sample period (full channel)	400 ps to 10 ns

Menu	Description
Sample period (half channel)	200 ps
Maximum time between transitions	66 days
Minimum data pulse width	1 sample period + 200 ps
Time interval accuracy	± (1 sample period + 400 ps + 0.01% of time interval reading)
* Items marked with an asterisk (*) are specifications. All others are characteristics. "Typical" represents the average or median value of the parameter based on measurements from a significant number of units.	

Trigger Characteristics

Menu	Description
Sample rate on all channels	2500 MHz
Sample rate in 1/2 channel mode	5000 MHz
Sample period (full channel)	400 ps
Sample period (half channel)	200 ps
Maximum trigger sequence speed	2500 MHz
Maximum trigger sequence levels	8
Trigger sequence level branching	Arbitrary 4-way if/then/else
Trigger position	Start, center, end, or user-defined
Trigger resources	16 patterns evaluated as =, !=, >, >=, <, <= 8 double-bounded ranges evaluated as in range, not in range 1 timer 3 flags 1 occurrence counter per sequence level
Trigger resource conditions	Arbitrary Boolean combinations
Trigger actions	Go To Trigger, send e-mail, and fill memory Trigger and Go To Trigger and fill memory
Store qualification actions	Default (global) and per sequence level Store/don't store sample Turn on/off default storing
Maximum global counter	N/A
Maximum occurrence counter	999,999,999
Maximum pattern width	128 bits – single label 272 bits – AND of multiple labels across two-card set
Maximum range width ⁴	64 bits

Menu	Description
Timers range	100 ns to 27 hours (in timing modes) 200 * state clock period to 27 hours (in state mode)
Timer resolution	5 ns
Timer accuracy	± (5 ns +0.01%) (in timing modes) ± (8 * state clock period +2ns +0.01%) (in state mode)
Timer reset latency	40 ns (in timing modes) 80 * state clock period (in state mode)
* Items marked with an asterisk (*) are specifications. All others are characteristics. "Typical" represents the average or median value of the parameter based on measurements from a significant number of units.	

Other

Menu	Description
Supported signal types:	Single-ended and differential
Probe compatibility:	90-pin cable connector
Voltage threshold:	-5 V to 5 V
Threshold accuracy:	±(30 mV + 1% of setting)
Threshold setting granularity:	By channel

Chassis
Compatibility

Keysight AXIe 2-slot or 5-slot chassis with software revision 5.0 or greater.

Power
Requirements

All necessary power is supplied by the backplane connector of the Keysight AXIe chassis.

Environmental
Characteristics

Indoor use only.

Operating
Environment

Menu	Description
Temperature:	0°C to 40°C (+32°F to 104°F). Reliability is enhanced when operating within the range +20°C to +35°C (+68°F to +95°F).
Humidity:	0 to 80% relative humidity at 40°C (+104°F). Reliability is enhanced when operating within the range 20% to 80% non-condensing.
Altitude:	0 to 3,000 m (10,000 ft)
Vibration:	Random vibration 5-500 Hz, 10 minutes per axis, approximately 0.2 g rms

Non-operating
Environment

Menu	Description
Temperature:	-40°C to +75°C (-40°F to +167°F). Protect the instrument from temperature extremes which cause condensation on the instrument.

Menu	Description
Humidity:	0 to 90% relative humidity at 65°C (149°F)
Altitude:	0 to 15,300 m (50,000 ft)
Vibration (in shipping carton):	Random vibration 5 to 500 Hz, 10 minutes per axis, approximately 2.41 g rms; and swept sine resonant search, 5 to 500 Hz, 0.50 g (0-peak), 5-minute resonant dwell at 4 resonances per axis.

- See Also
- What is a Specification (see [page 611](#))
 - What is a Characteristic (see [page 611](#))

U4154B Logic Analyzer Specifications and Characteristics

Describes the specifications and characteristics of the U4154B logic analyzer.

NOTE

Complete specifications and characteristics of the U4154B logic analyzer are available in the [U4154B Logic Analyzer Data Sheet \(5992-0108EN\)](#) on the Keysight web site.

NOTE

Items marked with an asterisk (*) are specifications. All others are characteristics.

Typical (typ): Represents characteristic performance, which 80% of the instruments manufactured will meet. This data is not warranted, does not include measurement uncertainty, and is valid only at room temperature (approximately 25 °C).

Nominal (nom): The expected mean or average performance, or an attribute whose performance is by design, such as the 50 O connector. This data is not warranted and is measured at room temperature (approximately 25 °C).

Measured (meas): An attribute measured during the design phase for purposes of communicating expected performance, such as amplitude drift vs. time. This data is not warranted and is measured at room temperature (approximately 25°C).

Module Channel Counts

Feature	Description
Number of Channels (nom)	136 in one U4154B 272 in two U4154Bs combined 408 with three U4154B combined
Maximum channels on single time base and trigger (nom)	408

State
(Synchronous)
Analysis Mode

Feature	Description
Maximum state data rate Option 2.5 GHz state mode (spec)	2.5 Gb/s on 136 channels per U4154B, using either or both edges of clock (spec) 4 Gb/s on 68 channels per U4154B, clocking on either edge of the clock (typ)
Maximum state data rate standard -01G 1.4 GHz state mode (spec)	1.4 Gb/s on 136 channels per U4154B, using either or both edges of clock (spec) 2.8 Gb/s on 68 channels per U4154B, clocking on either edge of the clock (typ)
Maximum state clock frequency (typ)	2.5 GHz Option -02G 1.4 GHz standard
Minimum data valid window (typ)	100 ps Option -02G 160 ps standard -01G
Sample position adjustment accuracy (typ)	± 150 ps
Sample position adjustment resolution (typ)	5 ps or 20 ps Option -02G 20 ps standard
Minimum state clock frequency ¹ (typ)	12.5 MHz (single edge) 6.25 MHz (both edges)
Minimum setup time (typ)	50 ps
Minimum hold time (typ)	50 ps
Minimum eye height (typ)	100 mV Option -02G 160 mV standard
Sample position adjustment range (typ)	7 ns
Minimum state clock pulse width (typ)	200 ps
Number of clocks (nom)	1
Minimum time between active clock edges (typ)	400 ps
Maximum time between active clock edges 1 (typ)	80 ns
Number of clock qualifiers	4 (pods 2, 3, 4 and 5 on clocking module)
Clock qualifier setup time	150 ps
Clock qualifier hold time	150 ps
Number of "RESET" clock qualifiers	1 (pod 7 of clocking module)
"RESET" clock qualifier setup time	2 ns
"RESET" clock qualifier hold time	0 ps
Time tag resolution (typ)	80 ps
Maximum time count between stored states (typ)	66 days

¹ Clock can pause for up to 66 days once every 8 or more edges

Timing (Asynchronous) Analysis Modes

Feature	Half Channel Mode	Full Channel Mode
Maximum sample rate (nom)	5 GHz 2.5 GHz	
Minimum sample period (nom)	200 ps	400 ps
Pod usage (nom) 1 pod from each	1 pod from each odd/even pod pair, user selectable	All pods
Timing Zoom sampling rate (nom)	12.5 GHz	
Timing Zoom memory depth (nom)	256 K samples	
Maximum time between transitions (nom)	66 days	
Minimum data pulse width (typ)	1 sample period + 200 ps	
Time interval accuracy (typ)	$\pm (1 \text{ sample period} + 400 \text{ ps} + 0.01\% \text{ of time interval reading})$	

Trigger Characteristics

Menu	Description
Maximum trigger sequence speed	2500 MHz (400 ps) Option -02G 1400 MHz (714 ps) Option -01G
Maximum trigger sequence levels (nom)	8
Trigger sequence level branching (nom)	Arbitrary 4-way if/then/else Trigger position (nom) Start, center, end, or user-defined
Trigger resources	16 patterns evaluated as =, !=, >, >=, <, <= 8 double-bounded ranges evaluated as in range, not in range 4 to 8 burst detectors 4 edge detectors in timing, 3 in transitional timing 1 occurrence counter per sequence level 1 timer 3 flags 1 arm in
Trigger resource Boolean conditions (nom)	Arbitrary Boolean combinations
Trigger actions	Goto Trigger and fill memory Trigger and Goto Trigger, send e-mail, and fill memory
Store qualification actions	Default (global) and per sequence level Store/don't store sample Turn on/off default storing

Menu	Description
Timer actions	Start from reset Stop and reset Pause Resume Flag actions Set Clear Pulse set Pulse clear
Maximum occurrence counter (nom)	999,999,999
Maximum pattern width (nom)	128 bits - single label 408 bits - AND of multiple labels across three-card set
Maximum range width (nom)	64 bits
Timers range (nom)	100 ns to 27 hours (in timing modes) 200 * state clock period to 27 hours (in state mode)
Timer resolution (nom)	5 ns
Timer accuracy (typ)	$\pm (5 \text{ ns} + 0.01\%)$ (in timing modes) $\pm (8 * \text{state clock period} + 2 \text{ ns} + 0.01\%)$ (in state mode)
Timer reset latency (nom)	40 ns (in timing modes) 80 * state clock period (in state mode)

Other

Menu	Description
Input signal amplitude V _{amptd} (typ)	=> 350 mV
Supported signal types	Single-ended and differential
Probe compatibility	90-pin cable connector
Voltage threshold (typ)	-5 V to 5 V
Threshold accuracy (typ)	$\pm(30 \text{ mV} + 1\% \text{ of setting})$
Minimum threshold resolution (typ)	2 mV Option -02G 20 mV standard
Threshold setting granularity	By channel

Chassis Compatibility Keysight AXIe 2-slot or 5-slot chassis with Logic and Protocol Analyzer software version 6.0 or greater.

Power Requirements All necessary power is supplied by the backplane connector of the Keysight AXIe chassis.

Environmental Characteristics Indoor use only.

Operating Environment

Menu	Description
Temperature:	0°C to 40°C (+32°F to 104°F).
Humidity:	0 to 80% relative humidity at 40°C (+104°F).
Altitude:	0 to 3,000 m (10,000 ft)
Vibration:	Random vibration 5-500 Hz, 10 minutes per axis, approximately 0.2 g rms

Non-operating Environment

Menu	Description
Temperature:	-40°C to +75°C (-40°F to +167°F). Protect the instrument from temperature extremes which cause condensation on the instrument.
Humidity:	0 to 90% relative humidity at 65°C (149°F)
Altitude:	0 to 15,300 m (50,000 ft)
Vibration (in shipping carton):	Random vibration 5 to 500 Hz, 10 minutes per axis, approximately 2.41 g rms; and swept sine resonant search, 5 to 500 Hz, 0.50 g (0-peak), 5-minute resonant dwell at 4 resonances per axis.

Weight 2.34 Kg

- See Also
- What is a Specification (see [page 611](#))
 - What is a Characteristic (see [page 611](#))

16850-Series Logic Analyzer Specifications and Characteristics

This topic describes the specifications and characteristics of the 16850-series logic analyzers.

NOTE

"Typical" represents the average or median value of the parameter based on measurements from a significant number of units.

- Channel Count per Measurement Mode (see [page 600](#))
- Probes (see [page 601](#))
- Timing Zoom (see [page 601](#))
- State (Synchronous) Analysis Mode (see [page 601](#))
- Timing (Asynchronous) Analysis Mode (see [page 602](#))
- Other (see [page 603](#))
- General Information (see [page 603](#))
- Environmental Characteristics (see [page 604](#))

Channel Count per Measurement Mode

16851A	16852A	16853A	16854A
34 (includes 1 clock + 1 clock qualifier)	68 (includes 1 clock + 3 clock qualifiers)	102 (includes 1 clock + 3 clock qualifiers)	136 (includes 1 clock + 3 clock qualifiers)
Unused clock channels can be used as data channels.			

Probes A probe must be used to connect the logic analyzer to your device under test. For specifications and characteristics of a particular probe, see the documentation that is supplied with your probe or search for the probe's model number at "www.keysight.com".

Timing Zoom

Menu	Description
Timing analysis sample rate:	12.5 GHz (80 ps sample resolution)
Timing interval accuracy:	Within a 16 channel block +/- (80 ps + 130ps + 0.01% of time interval reading) Between 16 channel blocks +/- 80 ps + 400ps + 0.01% of time interval reading)
Memory depth:	256 K samples
Trigger position:	Start, center, end, or user-defined
Minimum data pulse width:	1 sample period + 200 ps

State (Synchronous) Analysis Mode

	Default	Option 700
Minimum setup time (typical)	80 ps	
Minimum hold time (typical)	80 ps	
Minimum data valid window	160 ps	
Sample position adjustment range	7 ns typical	
Sample position adjustment resolution	20 ps typical	
Maximum state data rate on each channel	700 Mb/s	1400 Mb/s
Maximum state clock frequency	single edge clocking 350 MHz	single edge clocking 700 MHz
Minimum state clock frequency	12.5 MHz (single edge) 6.25 MHz (both edges)	
Number of clocks	1 Clock on Pod1 of the master card	
Number of clock qualifiers	For 16851A - 1 For 16852A, 3A, 4A models - 3	
Minimum time between active clock edges	1429 ps	714 ps
Minimum state clock pulse width	Single edge: 200 ps	
Clock qualifier setup time	200 ps	
Clock qualifier hold time	200 ps	
Time tag resolution	80 ps	
Maximum time count between stored states	66 days	
Maximum trigger sequence speed	700 MHz	1.4 GHz
Maximum trigger sequence steps	8	
Trigger sequence step branching	Arbitrary 4-way if/then/else	

	Default	Option 700
Trigger position	Start, center, end, or user-defined	
Trigger resources	<ul style="list-style-type: none"> · 16 patterns evaluated as =, !=, >, >=, <, <= · 8 double-bounded ranges evaluated as in range, not in range · 4 edge detectors in timing, 3 in transitional timing · 1 occurrence counter per sequence level · 1 timer · 3 flags · 1 arm in 	
Trigger resource conditions	Arbitrary Boolean combinations	
Trigger actions	<ul style="list-style-type: none"> · Go To · Trigger and fill memory · Trigger and Go To · Trigger, send e-mail, and fill memory · Occurrence counter reset · Store qualification · Default (global) and per sequence level · Store/don't store sample · Turn on/off default storing 	
Maximum occurrence counter	999,999,999	
Maximum pattern width	128 bits –single label	
Maximum range width	64 bits	
Timer value range	200 * sample clock period to 27 hours	
Timer resolution	5 ns	
Timer accuracy	+/- (8 * sample clock period + 2ns + 0.01%)	
Timer reset latency	80 * sample clock period	

Timing (Asynchronous) Analysis Mode

Conventional and Transitional Timing	
Maximum Sample rate on all channels	2.5 GHz
Maximum Sample rate in half channel mode	5 GHz
Sample period (half channel)	200 ps
Sample period (full channel)	400 ps to 10 ns
Minimum data pulse width	1 sample period + 200 ps
Time interval accuracy	Within a 16 channel pod +/- (1 sample period + 130ps + 0.01% of time interval reading) ¹ Across 16 channel pods +/- (1 sample period + 400ps + 0.01% of time interval reading) ²
Memory depth in full channel mode	Up to 128M depth
Memory depth in half channel mode	Up to 256 M depth

Conventional and Transitional Timing	
Maximum trigger sequence speed	2.5 GHz
Maximum trigger sequence steps	8
Trigger sequence step branching	Arbitrary 4-way if/then/else
Trigger position	Start, center, end, or user-defined
Trigger resources	<ul style="list-style-type: none"> · 16 patterns evaluated as =, !=, >, >=, <, <= · 8 double-bounded ranges evaluated as in range, not in range · 4 edge detectors in timing, 3 in transitional timing · 1 occurrence counter per sequence level · 1 timer · 3 flags · 1 arm in
Trigger resource conditions	Arbitrary Boolean combinations
Trigger actions	<ul style="list-style-type: none"> · Go To · Trigger and fill memory · Trigger and Go To · Trigger, send e-mail, and fill memory · Occurrence counter reset
Flag actions	<ul style="list-style-type: none"> · Set · Clear · Pulse Set · Pulse Clear
Maximum occurrence counter	999,999,999
Maximum range width	64 bits
Maximum pattern width	128 bits –single label
Timer range	200 * sample clock period to 27 hours
Timer resolution	5 ns
Timer accuracy	+/- (8 * sample clock period + 2ns + 0.01%)
Timer reset latency	80 * sample clock period

Others

Supported signal types	Single-ended, differential
Automated threshold/sample position	Yes
Simultaneous eye diagrams, all channels	Yes

General Information

Power:

100-120VAC 50/60/400Hz & 100-240VAC 50/60 mains capable of supplying 325 watts

CAUTION

This instrument has auto-ranging line voltage input, be sure the supply voltage is within the specified range and voltage fluctuations do not to exceed 10 percent of the nominal supply voltage.

Weight:

Max Net	Max Shipping
14.6 kg (32.2 lbs)	21.7 kg (48 lbs)

Operating System:

Microsoft Windows 7 Embedded (64-bit)

Printers:

Can print to any local or network printer supported by the installed operating system (Windows 7).

Environmental Characteristics

- Intended for use in an indoor lab environment
- Pollution degree 2
- Installation category II

See individual probe Specifications and Characteristics for probe environmental characteristics.

- Operating Environment

Operating Environment

Attribute	Description
Temperature:	5°C to 40°C
Humidity:	90% to temperatures up to 31°C decreasing linearly to 50% rH at 40°C; /// max 90% rh, non-condensing. Type tested at 95% RH, +40 degrees C (non-condensing)
Altitude:	4000 m

See Also

- What is a Specification (see [page 611](#))
- What is a Characteristic (see [page 611](#))

16860-series Logic Analyzer Specifications and Characteristics

Describes the specifications and characteristics of the 16860-series logic analyzer.

NOTE

Complete and up-to-date specifications and characteristics of the 16860-series logic analyzer are available in the 16860-series Logic Analyzer Data Sheet on the Keysight web site.

NOTE

Specification (spec): Represents warranted performance of a calibrated instrument that has been stored for a minimum of 2 hours within the operating temperature range, unless otherwise stated, and after a 45 minute warm-up period. The specifications include measurement uncertainty.

Typical (typ): Represents characteristic performance, which 80% of the instruments manufactured will meet. This data is not warranted, does not include measurement uncertainty, and is valid only at room temperature (approximately 25 °C).

Nominal (nom): The expected mean or average performance, or an attribute whose performance is by design, such as the 50 Ω connector. This data is not warranted and is measured at room temperature (approximately 25 °C).

Measured (meas): An attribute measured during the design phase for purposes of communicating expected performance, such as amplitude drift vs. time. This data is not warranted and is measured at room temperature (approximately 25 °C).

- Channel Counts (see [page 605](#))
- State (Synchronous) Analysis Mode (see [page 605](#))
- Timing (Asynchronous) Analysis Modes (see [page 608](#))
- Timing Zoom (see [page 610](#))
- Probes (see [page 610](#))
- Others (see [page 610](#))
- General Information (see [page 610](#))
- Environmental Characteristics (see [page 610](#))
- Operating Environment (see [page 611](#))

Channel Counts

	16861A	16862A	16863A	16864A
Channels	34 (32 data channels and 2 clock channels)	68 (64 data channels and 4 clock channels)	102 (96 data channels and 6 clock channels)	136 (128 data channels and 8 clock channels)

Unused clock channels can be used as data channels.

State
(Synchronous)
Analysis Mode

Feature	For Single Clock	For Multiple Clocks
Maximum state data rate for the 350 MHz option (spec)	Captures a 350MHz signal on either edge of the clock up to 350 MSP/s (spec) Captures a 350MHz signal on both edges of the clock up to 700 MSP/s (spec)	Captures a 350MHz signal on either edge of the clock up to 350 MSP/s (spec) Captures a 350MHz signal on both edges of the clock up to 700 MSP/s (spec)
Maximum state data rate for the 700 MHz option (spec)	Captures a 700MHz signal on either edge of the clock up to 700 MSP/s (spec) Captures a 700MHz signal on both edges of the clock up to 1400 MSP/s (spec)	Captures a 350MHz signal on either edge of the clock up to 350 MSP/s (spec) Captures a 350MHz signal on both edges of the clock up to 700 MSP/s (spec)
Maximum state clock frequency	700 MHz (Option -700) 350 MHz standard (base)	350MHz
Minimum state clock frequency	12.5 MHz (single edge) 6.25 MHz (both edges)	0 MSPs
Minimum setup time (typical)	80 ps	250 ps

Feature	For Single Clock	For Multiple Clocks
Minimum hold time (typical)	80 ps	250 ps
Minimum data valid window	160 ps	500 ps
Sample position adjustment range	7 ns typical	12 ns typical
Sample position adjustment resolution	20 ps typical	80 ps typical
Number of clocks	1 (Clock on Pod1)	For 16861A - Up to 2 clocks (in an Ored combination) For 16862A/3A/4A - Up to 4 clocks (in an Ored combination). Master and slave clocks are counted separately
Number of clock qualifiers	For 16864A - Up to five clock qualifiers (Pod 2, 3, 4, 5, 7). For 16862A/3A - Up to three clock qualifiers (Pod 2, 3, 4) For 16861A - One clock qualifier (Pod 2)	For 16862A/3A/4A - Up to two clock qualifiers (Pod 1, 2, 3, 4) Note: Though there are three clock qualifiers displayed, you can use a maximum of only two qualifiers for a clock. The third qualifier is automatically set to "Don't Care" and can be used only as an Ored clock in the clock description. For 16861A - One clock qualifier (Pod 1 or 2)
Minimum time between active clock edges	1430 ps (for 350 MHz state speed) 715 ps (for 700 MHz state speed)	2.86 ns (for 350 MHz state speed) 1.43 ns (for 700 MHz state speed)
Minimum state clock pulse width	Single edge: 200 ps	Single edge: 250 ps
Clock qualifier setup time	200 ps	250 ps
Clock qualifier hold time	200 ps	250 ps
Number of RESET clock qualifiers	1 (pod 7 of clocking module)	0
RESET clock qualifier setup time	2 ns	N/A
RESET clock qualifier hold time	0 ps	N/A
Minimum slave to master clock time	N/A	350 ps
Minimum master to slave clock time	N/A	150 ps
Minimum slave to slave clock time	N/A	1.43 ns
Time tag resolution	80 ps	80 ps
Maximum time count between stored states	66 days	66 days
Maximum trigger sequence speed	700 MHz (for 350 MHz state speed option) 1.4 GHz (for 700 MHz state speed option)	350 MHz (for 350 MHz state speed option) 700 MHz (for 700 MHz state speed option)

Feature	For Single Clock	For Multiple Clocks
Maximum trigger sequence steps	8	16
Trigger sequence step branching	Arbitrary 4-way if/then/else	
Trigger position	Start, center, end, or user-defined	
Trigger resources	16 patterns evaluated as =, !=, >, >=, <, <= 8 double-bounded ranges evaluated as in range, not in range 4 edge detectors in timing, 3 in transitional timing 1 occurrence counter per sequence level 1 timer 4 flags 1 arm in Burst patterns feature Event counters - 2 Global counters - Not supported	16 patterns evaluated as =, !=, >, >=, <, <= 8 double-bounded ranges evaluated as in range, not in range 4 edge detectors in timing, 3 in transitional timing 1 occurrence counter per sequence level 3 timers 4 flags 1 arm in Number of timers - 3 Global counters - 2 Event counters - Not supported Burst patterns - Not supported
Trigger resource conditions	Arbitrary Boolean combinations	
Trigger actions	Go To Trigger and fill memory Trigger and Go To Trigger, send e-mail, and fill memory Occurrence counter reset	
Store qualification	Default (global) and per sequence level Store/do not store sample Turn on/off default storing	
Maximum occurrence counter	999,999,999	
Maximum pattern width	128 bits single label	
Maximum range width	64 bits	
Timer value range	200 * sample clock period to 27 hours	
Timer resolution	5 ns	
Timer accuracy	+/- (8 * sample clock period + 2ns + 0.01%)	
Timer reset latency	80 * sample clock period	

Timing (Asynchronous) Analysis Modes

Feature	Conventional and Transitional Timing
Maximum Sample rate on all channels	2.5 GHz
Maximum Sample rate in half channel mode	5 GHz
Sample period (half channel)	200 ps
Sample period (full channel)	400 ps to 10 ns
Minimum data pulse width	1 sample period + 200 ps

Feature	Conventional and Transitional Timing
Time interval accuracy	Within a 16 channel pod +/- (1 sample period + 130ps + 0.01% of time interval reading) ¹ Across 16 channel pods +/- (1 sample period + 400ps + 0.01% of time interval reading) ²
Memory depth in full channel mode	Up to 128M depth
Memory depth in half channel mode	Up to 256 M depth
Maximum trigger sequence speed	2.5 GHz
Maximum trigger sequence steps	8
Trigger sequence step branching	Arbitrary 4-way if/then/else
Trigger position	Start, center, end, or user-defined
Trigger resources	16 patterns evaluated as =, !=, >, >=, <, <= 8 double-bounded ranges evaluated as in range, not in range 4 edge detectors in timing, 3 in transitional timing 1 occurrence counter per sequence level 1 timer 4 flags 1 arm in Burst trigger 2 Event counters
Trigger resource conditions	Arbitrary Boolean combinations
Trigger actions	Go To Trigger and fill memory Trigger and Go To Trigger, send e-mail, and fill memory Occurrence counter reset
Flag actions	Set Clear Pulse Set Pulse Clear
Maximum occurrence counter	999,999,999
Maximum range width	64 bits
Maximum pattern width	128 bits –single label
Timer range	200 * sample clock period to 27 hours
Timer resolution	5 ns
Timer accuracy	+/- (8 * sample clock period + 2ns + 0.01%)
Timer reset latency	80 * sample clock period

Timing Zoom (Asynchronous) sampling mode

Feature	Description
Timing analysis sample rate	12.5 GHz (80 ps sample resolution)
Timing interval accuracy	Within a 16 channel block +/- (80 ps + 130ps + 0.01% of time interval reading) Between 16 channel blocks +/- 80 ps + 400ps + 0.01% of time interval reading)
Memory depth	256 K samples
Trigger position	Start, center, end, or user-defined
Minimum data pulse width	1 sample period + 200 ps

Probes For specifications and characteristics of a particular probe, see the documentation that is supplied with your probe or search for the probe's model number at www.keysight.com.

Others

Feature	Description
Supported signal types	Single-ended, differential
Automated threshold/sample position	Yes
Simultaneous eye diagrams, all channels	Yes

General Information

Feature	Description
Power	100 to 120 VAC ± 10% 50/60/400 Hz 100 to 240 VAC ± 10% 50/60 Hz Maximum input power 325 W
	CAUTION This instrument has auto-ranging line voltage input, be sure the supply voltage is within the specified range and voltage fluctuations do not to exceed 10 percent of the nominal supply voltage.
Weight	Max Net 12.5 kg (28 lbs) Max Shipping 23.3 kg (51 lbs)
Operating system	Microsoft Windows 7 Embedded (64-bit)
Printers	Can print to any local or network printer supported by the installed operating system (Windows 7).
Logic and Protocol Analyzer Software Compatibility	Version 6.3 or greater

Environmental Characteristics

Intended for use in an indoor lab environment
Pollution degree 2
Installation category II

See individual probe Specifications and Characteristics for probe environmental characteristics

Operating Environment

Feature	Description
Temperature	+5 to +40°C (Operating) -40 to +65°C (Non-operating)
Humidity	Up to 90% relative humidity (non-condensing) at +40°C (Operating) Up to 90% relative humidity (non-condensing) at +65°C (Non-operating) Type tested at 95% RH, +40 degrees C (non-condensing)
Altitude	Operating up to 4,000 meters (13,000 feet) Non-operating up to 15,300 meters (50,000 feet)

- See Also
- What is a Specification (see [page 611](#))
 - What is a Characteristic (see [page 611](#))

What is a Specification?

A specification is a numeric value, or range of values, that bounds the performance of a product parameter. The product warranty covers the performance of parameters described by specifications. Products shipped from the factory meet all specifications. Additionally, products sent to Keysight Customer Service Centers for calibration, and returned, meet all specifications. Specifications are verified by *calibration procedures*.

What is a Calibration Procedure?

Calibration procedures verify that products or systems operate within the specifications. Parameters covered by specifications have a corresponding calibration procedure. Calibration procedures include both performance tests and system verification procedure. Calibration procedures are traceable and must specify adequate calibration standards.

Calibration procedures verify products meet the specifications by comparing measured parameters against a pass-fail limit. The pass-fail limit is the specification less any required guardband.

The term "calibration" refers to the process of measuring parameters and referencing the measurement to a calibration standard rather than the process of adjusting products for optimal performance.

NOTE

Self-tests are not a substitute for calibration.

- See Also
- What is a Characteristic (see [page 611](#))

What is a Characteristic?

Characteristics describe product performance that is useful in the application of the product, but that is not covered by the product warranty. Characteristics describe performance that is typical of the majority of a given product, but not subject to the same rigor associated with specifications. Characteristics are verified by *function tests*.

What is a Function Test?

Function tests are quick tests designed to verify basic operation of a product. Function tests include operator's checks and operation verification procedures. An operator's check is normally a fast test used to verify basic operation of a product. An operation verification procedure verifies some, but not all, specifications, and often at a lower confidence level than a calibration procedure.

See Also · [What is a Specification \(see page 611\)](#)

Glossary

A

acquisition

Denotes one complete cycle of data gathering by a measurement module. For example, if you are using an analyzer with 200K memory depth, one complete acquisition will capture and store 200K states in acquisition memory.

acquisition depth

The acquisition depth is the amount of memory that is filled with data on an acquisition. The choices available depend on the maximum memory depth available in the analyzer that is being used.

action

Actions are things that the analyzer does as a part of triggering, for example "Then Trigger and Fill Memory" or "Start Timer."

activity indicator

Symbols next to logic analyzer channels that indicate whether a signal is a logic-high, or logic-low, or whether the signal is changing between highs and lows.

advanced trigger

Advanced triggers provide more power than simple triggers, but are more complex.

analysis probe

A probe connected to a microprocessor or standard bus in the device under test. An analysis probe provides an interface between the signals of the microprocessor or standard bus and the inputs of the logic analyzer.

arming

Typically, instruments are armed immediately when Run or Run Repetitive is selected. For example, logic analyzers are commonly used to arm oscilloscopes.

asynchronous sampling

When the logic analyzer acquires samples from the device under test asynchronously, that is, at regular intervals, such as every 100 ns. Also known as timing mode.

B

beginning of acquisition

The beginning of the acquisition is the point in time where the collection of data begins.

bits

A bit is a single signal in a bus. Numbering of bits begins with 0.

bus

A bus is a group of associated signals, such as ADDR or DATA.

C

captured data

Signal values that have been sampled by the logic analyzer and stored in its memory.

card

A logic analyzer that can be inserted into a slot (see [page 618](#)) in a frame (see [page 615](#)). Cards can be combined with others to increase the channel count available in a single time domain.

channel

A single line of input to the logic analyzer. Each channel corresponds to a lead that is connected to the device under test. Each channel is used to acquire one and only one signal from the device under test.

chassis

A chassis is a modular instrument chassis that supports complex and high density testing. The Keysight AXIe chassis provides slots for installing multiple instrument modules such as the U4154A module.

clock channel

A special logic analyzer input channel that can be used to determine the analyzer's sampling. Clock channels are identified on a pod by CLK.

CLOCK IN BNC

This input is for a 10 MHz reference clock used to synchronize the logic analysis system with other external instruments.

Do not confuse this BNC input with logic analyzer clock inputs (which are present on the logic analyzer pod cables).

D

data channel

A channel that carries data. Data channels cannot be used to clock logic analyzers. Data channels are numbered as opposed to clock channels which are labeled CLK.

default storage

Default storage means "unless sequence step storage specifies otherwise, this is what should be stored". Sequence step storage always overrides default storage.

delay

Delay is the horizontal position of the waveform on the screen for the timing analyzer. Delay time is measured from the trigger point in seconds.

device under test

The system under development whose digital signals are captured by the logic analyzer.

don't care

A "don't care" means that the state of the signal (high or low) is not relevant to the measurement. The analyzer ignores the state of this signal when determining whether a match occurs on an input bus/signal.

double-click

When using a mouse as a pointing device, to double-click an item, position the cursor over the item, and then quickly press and release the left mouse button twice.

drag and drop

Position the cursor over the item, and then press and hold the left mouse button. While holding the left mouse button down, move the mouse to drag the item to a new location. When the item is positioned where you want it, release the mouse button.

E**edge**

Logic analyzer trigger resources that allow detection of transitions on a signal. An edge term can be set to detect a rising edge, falling edge, or either edge.

event

Events are the things you are looking for in your device under test, for example ADDR=0 or ADDR=5.

external trigger

A signal outside the logic analyzer that is used to synchronize measurements between instruments. For example, the logic analyzer can be armed (activated) by a signal that comes from another instrument. Logic analyzers are commonly used to trigger oscilloscopes through a BNC connection.

F**frame**

A modular logic analysis system that has slot (see [page 618](#))s in the back for the insertion of logic analyzer card (see [page 614](#))s. Multiple frames can be connected together to form a multiframe logic analysis system.

I**inverse assembler**

A tool that displays the assembly language instructions for captured machine code.

L**logic analyzer**

An instrument that captures and displays digital signal values. A logic analyzer is like an oscilloscope, except that it only displays two voltage levels (a logic high or 1, and a logic low or 0) instead of many voltage levels. Because a logic analyzer only captures 1s and 0s, its sample rate can be slower than an oscilloscope that needs to capture more voltage detail. Consequently, a logic analyzer can capture a greater amount of overall execution time.

M**macro**

- In COM/DCOM, "macros" (in the online help) are **Sub** procedures that can be run from the user interface to automate an application.
- In previous Keysight logic analysis systems, macros, or *trigger macros*, were what are now called trigger function (see [page 620](#))s: preprogrammed components that are used to build trigger sequences.

marker

A relocatable reference point in the data display. Markers can be used to measure time intervals or sample intervals. Markers are assigned to patterns in order to find patterns or track sequences of state in the data.

menu bar

The menu bar is located at the top of all windows. Use it to select drop down menus that contain tool or system options.

module

A logical collection of logic analyzer card (see [page 614](#))s that are connected together. This gives you the flexibility to increase channel count by using more than one card. A module can be a single card or several cards. By definition, a module consists of a single time domain. While a module can consist of a single card, a module is not a physical entity.

O

occurrence

Occurrence is used in triggering to define how many times something happens during the acquisition.

offline analysis

Analyzing previously captured and saved logic analysis data without data acquisition hardware. In other words, you can use the *Keysight Logic Analyzer* application by itself on a Windows XP/Vista computer to analyze data in the waveform, listing, and compare windows.

P

pattern

Logic analyzer resources that represent single states to be found on buses/signals; for example, an address on the address bus or a status on the status lines.

pod

A physical collection of logic analyzer channels within a card (see [page 614](#)). Pods are numbered relative to cards only. Pods are used to physically connect data and clock signals from the device under test to the analyzer.

pod index

A logical number for a pod (see [page 616](#)). If a module (see [page 616](#)) has 20 pods, the pod indexes are 1 through 20 with no renumbering at card boundaries. In a multi-card module, numbering begins with the master card then continues from the bottom card up. Pod indexes can be used without considering how many pods are on each card (see [page 614](#)), or in which slot (see [page 618](#))s the cards are located.

pod pair

A group of two pods containing 16 data channels and 1 clock channel each. Pod pairs are used to physically connect data and clock signals from the device under test to the analyzer. Pods are assigned by pairs in the analyzer interface. The number of pod pairs available is determined by the channel width of the instrument.

pod truncation

This occurs when opening a configuration file that was saved from a logic analyzer module that had more pods. Because the current module has fewer pods, truncation occurs. Any buses/signals assigned to truncated pods are unusable.

point

To point to an item, move the mouse cursor over the item.

polarity

Positive polarity is when an incoming low voltage is shown with a high waveform and a logical value of 1. Negative polarity is when an incoming high voltage is shown with a low waveform and a logical value of 0. Polarity affects the display of values and waveforms, and does not affect the trigger.

preprocessor

See analysis probe (see [page 613](#)).

probe

A device to connect the various instruments of the logic analysis system to the device under test. There are many types of probes and the one you should use depends on the instrument and your data requirements. As a verb, "to probe" means to attach a probe to the device under test.

protocol

An agreed-upon format for transmitting data between two devices. The protocol determines: the type of error checking, data compression, encoding, how sending devices indicate they have finished sending a message, and how receiving devices indicate they have received a message.

Q**quick trigger**

Quick trigger allows you to quickly set up a simple trigger within the waveform and listing displays, by drawing a rectangle in the display area with the mouse. After a simple trigger has been defined, and the analyzer is run, the trigger is stored and can be recalled at any time.

R**range pattern**

Logic analyzer resources which let you set up patterns that represent a range of values, such as "ADDR in range 1000 to 2000". Most logic analyzers support a "not in range" operator as well as the "in range" operator. Range patterns are a convenient shortcut that can be used instead of AND'ing or OR'ing two patterns, such as "ADDR >= 1000 and ADDR <= 2000".

repetitive measurement

A measurement in which the logic analyzer's trigger condition is searched for, and data storage is filled, repetitively.

right-click

When using a mouse for a pointing device, to right-click an item, position the cursor over the item, and then quickly press and release the right mouse button.

run

The single run measurement will save captured data to trace memory one time. The amount of data stored during a single run is equal to the amount of trace memory allotted.

run repetitive

The run repetitive measurement will save the captured data to trace memory repetitively. The amount of data stored in a repetitive run is the same as a single run. During a repetitive run once the trace memory is full the system clears the trace memory and begins to refill with new data. This cycle will continue until the run is stopped.

S**sample**

A data sample is a single measurement. When an instrument samples the device under test, it takes a single measurement as part of its data acquisition cycle. The number of samples acquired is equal to the logic analyzers memory depth.

sample period

The sample period is the period of time between samples. The sample period can be based on an internal sampling clock (also known as timing analysis or asynchronous sampling). Or, the sampling can be based on a signal in the device under test (also known as state analysis, or synchronous sampling).

sampled data

Signal values that are sampled by the logic analyzer (not necessarily stored).

sampling

The process by which the logic analyzer looks at digital signals.

search

Searches through the acquired data for specified data pattern or value, time value, sample number, or marker. Search criteria can range from specific bits to multiple events, depending on which search option you choose.

simple trigger

Simple triggers include triggers such as edges and bus patterns.

single measurement

A measurement in which the logic analyzer's trigger condition is searched for, and data storage is filled, once.

skew

Skew is the difference in channel delays between measurement channels.

slot

An opening in the back of a frame (see [page 615](#)) where card (see [page 614](#))s can be inserted. The slots are lettered from A to F with A being the topmost slot. Slots are physical entities and are always referred to by slot letter.

snap to edge markers

Snap to edge markers enable easy placement of markers on waveform edges. When a marker is moved in the data display area, the cursor changes to a green "direction arrow" indicating the direction of the next valid edge. A red "valid edge" bar is placed on the next edge that the marker will be placed on.

state analyzer

A logic analyzer that samples based on a clock signal in the device under test.

state measurement

In a state measurement, the logic analyzer is clocked by a signal from the system under test. Each time the clock signal becomes valid, the analyzer samples data from the system under test. Since the analyzer is clocked by the system, state measurements are synchronous with the test system.

state mode

When the logic analyzer acquires samples from the device under test synchronously, in other words, when a signal or signals from the device under test indicates when to acquire a sample. For example, the logic analyzer might take a sample whenever there is a rising edge on a signal from the device under test. Typically, the signal used to set up the sampling is a state machine clock signal or microprocessor clock signal. Also known as synchronous sampling.

stop

Stops the measurement currently in progress.

storage qualification

Storage qualification is only available in a state measurement, not timing measurements. Store qualification allows you to specify the type of data (all samples, no samples, or selected states) to be stored in memory. Use store qualification to prevent memory from being filled with unwanted activity such as wait-loops. Storage qualification lets you filter out specific types of data as the acquisition is running, which saves memory. In contrast filters can hide data after it has been collected.

symbols

Names assigned to particular bus or signal values. Symbols in a display of captured data values are easy to read. Also, symbols make it easy to set up triggers on particular values. For example, in a communication protocol you could display the value FF as "end of file."

synchronous sampling

When the logic analyzer acquires samples from the device under test synchronously, in other words, when a signal or signals from the device under test indicates when to acquire a sample. For example, the logic analyzer might take a sample whenever there is a rising edge on a signal from the device under test. Typically, the signal used to set up the sampling is a state machine clock signal or microprocessor clock signal. Also known as state mode.

T

target system

See device under test (see [page 614](#)).

threshold voltage

The voltage level that the signal must cross before the logic analyzer recognizes a change in voltage levels. A high voltage level is indicated by a "1" and a low voltage level is indicated by a "0." TTL and ECL are two examples of voltage levels that the signal must cross.

time/division

Time/division controls the "zooming" of a waveform display. Increasing the time/division zooms out, while decreasing the time/division zooms in.

timer

Timers are used to create either a user-defined delay or a time standard which valid data duration is evaluated against.

timing analyzer

A logic analyzer that samples at regular intervals based on an internal clock signal.

timing measurement

In a timing measurement, the logic analyzer samples data at regular intervals according to a clock signal internal to the timing analyzer. Since the analyzer is clocked by a signal that is not related to the device under test, timing measurements capture traces of electrical activity over time. These measurements are asynchronous with the device under test.

timing mode

When the logic analyzer samples from the device under test asynchronously, that is, at regular intervals, such as every 100 ns. Also known as asynchronous sampling.

tool tip

Small information display (text readout) that appears during mouse operations such as hovering over a waveform or bus/signal name, moving markers, or drawing a rectangle in data. Use them as comments (see [page 94](#)) or to read current positions, waveform transition widths, or trigger specifications (when setting up quick triggers (see [page 143](#)) with mouse).

trace

See acquisition (see [page 613](#)).

transitional timing

When the logic analyzer is in transitional timing mode, the timing analyzer samples data at regular intervals, but only stores data when there is a threshold level transition (high-to-low transition, or low-to-high transition). Each time a level transition occurs on any of the bits, data on all channels is stored. A time tag is stored with each stored data sample so the measurement can be reconstructed and displayed later.

trigger

The event about which acquired data is stored; in other words, the event that you are looking for. For example, you may want to trigger on an edge in order to see the events that lead up to it and the events that happen after it. The event that triggers the logic analyzer becomes a reference point in the data display.

trigger function

Trigger functions are preprogrammed components that are used to build trigger sequences.

trigger history

Each time you set up a new trigger and run the measurement, the trigger setup is saved in the configuration file. Each saved trigger can be retrieved and reused. The default number of triggers saved is 10.

trigger position

The location of the trigger event in trace memory. If you want to view data after, about, or before the trigger event, you set the trigger position to the start, center, or end of trace memory, respectively.

trigger sequence

A trigger sequence is a sequence of events that you specify. The logic analyzer compares this sequence with the samples it is collecting to determine when to trigger.

V**value at measurement**

The value at measurement measures the value of a bus or a single signal at a specified marker location in data. Measurement results are displayed in the marker measurement display bar.

Z**zooming**

To expand and contract the waveform along the time base by varying the value in the time/div field. This action allows you to view specific portions of a particular waveform.

Index

Symbols

!= operator, pattern trigger, [147, 159](#)
< operator, pattern trigger, [147, 159](#)
<= operator, pattern trigger, [147, 159](#)
= operator, pattern trigger, [147, 159](#)
> operator, pattern trigger, [147, 159](#)
>= operator, pattern trigger, [147, 159](#)

Numerics

01G license, [547](#)
02G license, [547](#)
16850-series logic analyzer front panel operation, [527](#)
16850-series logic analyzer notes, [553](#)
16850-series logic analyzer specifications and characteristics, [600](#)
16860, [520, 528](#)
16860-series logic analyzer notes, [555](#)
8b, [299](#)

A

abbreviated marker name, [246, 254](#)
about, help, [397](#)
absolute time, [241](#)
accumulate waveforms, [229](#)
acquiring data, [141](#)
acquisition, [613](#)
acquisition depth, [613](#)
acquisition errors, [334](#)
acquisition memory depth, [128, 129, 139, 428, 545, 549, 553, 600](#)
acquisition memory depth and channel count trade-offs, [363](#)
acquisition mode, [428](#)
acquisition mode, choosing, [96](#)
acquisition mode, setting, [43](#)
acquisition mode, state, [98](#)
acquisition mode, timing, [96, 97](#)
acquisition time, transitional timing, [365](#)
action, [613](#)
action statements, [153, 154, 157](#)
action statements, reading, [153](#)
action, counter, [180](#)
action, flag, [180](#)
action, reset occurrence counter, [154, 180](#)
action, storage control, [181](#)
action, timer, [179](#)

actions and events, trigger, [152](#)
actions in trigger sequence, [371, 424](#)
actions, inserting or deleting, [179](#)
activation, software licenses, [323, 463](#)
active software licenses, [322, 461](#)
activity (bus/signal), [83](#)
activity indicator, [613](#)
activity indicators, [426](#)
activity indicators in Listing window, [237](#)
activity indicators in Waveform window, [226](#)
Add Application dialog, [436](#)
add external scope, [389](#)
add icons to toolbar, [402](#)
add logic analysis system, [459](#)
Add or Remove Logic Analyzer Software tool, [327](#)
add software, [327](#)
adding a bus or signal, [84](#)
adding a folder, [95](#)
adding connections, [307](#)
adding display windows, [307, 313](#)
adding system to use list, [61](#)
adding tools, [307](#)
address alignment, trigger in Source window, [467](#)
adjustable delay on state sampling clock input, [367](#)
advanced 2-way branch trigger function, state mode, [500](#)
advanced 2-way branch trigger function, timing mode, [488](#)
advanced 3-way branch trigger function, state mode, [501](#)
advanced 3-way branch trigger function, timing mode, [488](#)
advanced 4-way branch trigger function, state mode, [501](#)
advanced 4-way branch trigger function, timing mode, [489](#)
advanced clocking, [127](#)
Advanced Clocking Setup dialog, [422](#)
advanced if/then trigger function, state mode, [500](#)
advanced if/then trigger function, timing mode, [487](#)
advanced If/Then trigger functions, [478](#)
advanced setup, [389](#)
advanced trigger, [141, 152, 389, 399, 613](#)
Advanced Trigger dialog, [424](#)
advanced trigger example, [359](#)

advanced trigger functions, state mode, [490](#)
advanced trigger functions, timing mode, [478](#)
agLogicSvc.exe service, [348](#)
agSymbolBuild.exe, [135](#)
agSymbolQuery.exe, [135](#)
ALA format, [504](#)
ALA format configuration files, [198, 208, 210](#)
ALA format configuration files, automatic module mapping, [445](#)
ALA format configuration files, offline analysis, [218](#)
ALA format configuration files, won't open, [333](#)
ALA vs. XML format, when to use, [380](#)
ALB module binary file format, [203, 511](#)
ALB module binary file header format, [505](#)
alias bus/signal name, [95](#)
align trigger with, timing zoom, [476](#)
aligning reference data, [275](#)
alignment, bus/signal column, [241](#)
alignment, packet column, [305](#)
alignment, Set Quick Trigger, [282](#)
all reference data, [276](#)
altitude, [595, 600](#)
analog properties, [234](#)
analog signal row properties, [233](#)
analog signal, Voltage number base in Listing, [241](#)
analog signals, overlaying, [223](#)
analog signals, voltage markers for, [257](#)
analogy, conveyor belt, [369](#)
analysis probe, [57, 613](#)
analyzer, [72](#)
Analyzer Setup dialog, [426](#)
Analyzer Setup toolbar, [399](#)
analyzer triggering, understanding, [369](#)
analyzing captured data, [207](#)
analyzing listing data, [235](#)
analyzing waveform data, [220](#)
AND events, [168](#)
AND'ed find events, [270](#)
anything event, [168](#)
application (external), adding to Tools menu, [436](#)
arm from another module, [186](#)
arm from another module, wait for, [487, 499](#)
arm in from event, [168, 170](#)

arming, [613](#)
 arrange icons, [396](#)
 ASCII format symbol file, [514](#)
 Ascii number base, [159, 232, 241](#)
 ASCII symbol file, creating, [135](#)
 ASCII text file, print to, [314](#)
 Assign Names dialog, [86](#)
 asynchronous sampling, [96, 613](#)
 asynchronous sampling mode, [428](#)
 auto-connect at application
 startup, [439](#)
 auto-connect, selecting logic analysis
 system for, [66](#)
 automatically mapping modules, [444](#)
 autosize columns, [241](#)
 available status, [63](#)
 AXle chassis slots per card, [593](#)

B

B4661A, [317](#)
 background color, Listing
 window, [238, 239](#)
 background color, marker, [246, 256](#)
 background color, Source window, [282](#)
 background color, timing zoom
 listing, [239](#)
 background color, timing zoom
 waveforms, [229](#)
 background color, Waveform
 window, [227, 228](#)
 base, number, [93, 232, 241](#)
 base, symbol, [458](#)
 beginning of acquisition, [613](#)
 beginning of data, [222](#)
 Beginning of Data marker, [312](#)
 beginning of data, go to, [249](#)
 beginning of data, go to in Listing
 window, [236](#)
 best-fit algorithm, module
 mapping, [445](#)
 big endian, [91](#)
 Binary number base, [159, 232, 241](#)
 bit order, [89](#)
 bit reordering, [91](#)
 bits, [613](#)
 block boundary addresses, [282](#)
 BNC connectors, trigger in/out, [186](#)
 boolean expressions, [372](#)
 boolean expressions in trigger
 sequence, [371](#)
 boolean expressions, setting
 parentheses, [177](#)
 boolean expressions, triggering
 capability, [370](#)
 borrow floating licenses, [326, 465](#)
 borrow licenses messages, [466](#)
 branch expression too complex, [337](#)
 branches, [373](#)
 browsing source files, [278](#)
 buffer, reference, copying data to, [274](#)
 burst patterns, [170](#)

burst triggering, [170](#)
 bus, [613](#)
 bus analysis tools, new, [390](#)
 bus clock, data valid relative to, [367](#)
 bus pattern trigger functions, timing
 mode, [478](#)
 bus pattern triggers, [147, 159](#)
 bus width, [83](#)
 bus, group signals into, [227](#)
 bus, renaming the bits, [86](#)
 bus, source code correlation,
 selecting, [280](#)
 bus, source correlation, [468](#)
 bus/signal column alignment, [241](#)
 bus/signal column properties, [240](#)
 bus/signal column width in Listing
 window, [241](#)
 bus/signal column, finding a, [237](#)
 bus/signal columns, re-arranging, [237](#)
 bus/signal data color, Listing
 window, [240](#)
 bus/signal data to export, [199](#)
 bus/signal errors, [334](#)
 bus/signal event, [168](#)
 bus/signal folders, adding, [95](#)
 bus/signal name, alias, [95](#)
 bus/signal names, setting up, [42](#)
 bus/signal names, sorting, [95](#)
 bus/signal naming, short or
 unique, [447](#)
 bus/signal number base, [232, 241](#)
 bus/signal patterns, finding, [264, 265](#)
 bus/signal patterns, finding
 complex, [270](#)
 bus/signal row properties, [230](#)
 bus/signal row, finding a, [224](#)
 bus/signal setup, [83, 389, 399](#)
 bus/signal values displayed as
 symbols, [243](#)
 bus/signal values, symbolic, [138](#)
 bus/signal, adding a new, [84](#)
 bus/signal, assigning channels in
 default bit order, [88](#)
 bus/signal, assigning channels in
 selected bit order, [89](#)
 bus/signal, deleting a, [84](#)
 bus/signal, renaming a, [85](#)
 buses, expanding/collapsing, [227](#)
 buses/signals in data import
 module, [213, 442](#)
 Buses/Signals tab, [426](#)
 buses/signals, define by importing
 netlist, [446](#)
 buses/signals, exporting data from
 selected, [452](#)
 buses/signals, inserting/deleting in
 Listing window, [238](#)
 buses/signals, inserting/deleting in
 Waveform window, [227](#)
 buses/signals, load warning, [339](#)
 bus-specific probes, [57](#)

buttons, 16800-series logic
 analyzers, [527](#)

C

calibration, [611](#)
 calibration failed, [337](#)
 calibration procedure, [611](#)
 cancel, [392](#)
 cancel operation, [401](#)
 cancel processing,
 module/tool/window, [469](#)
 captured data, [141, 614](#)
 captured data, analyzing, [207](#)
 captured data, comparing, [274](#)
 captured data, exporting to module
 binary (ALB) files, [203](#)
 captured data, exporting to module
 CSV files, [201](#)
 captured data, printing, [314](#)
 captured data, saving, [198](#)
 card, [614](#)
 card, logic analyzer, [431](#)
 cards (logic analyzer), combining, [72](#)
 cards in frame, [472](#)
 cards in module, [474](#)
 cascade windows, [396, 398](#)
 center about edges, [221](#)
 center about marker pair, [249, 391, 400](#)
 center alignment, bus/signal
 columns, [241](#)
 center rectangle, go to source line
 of, [279](#)
 center rectangle, showing/hiding, [239](#)
 center screen, placing marker at, [247](#)
 changing the display scale, [221](#)
 changing the time/division, [221](#)
 channel, [614](#)
 channel and pod connections, [474](#)
 channel count and memory depth
 trade-offs, [363](#)
 channel counts, [588, 593, 596, 600, 605](#)
 channel naming conventions, [361](#)
 channel numbers, [426](#)
 channel numbers (bus/signal), [83](#)
 channel order selection, enable, [89](#)
 channel order, reverse, [91](#)
 channel reordering, [91](#)
 channel width, [97](#)
 channels, [545, 549, 553, 588, 593, 596, 600, 605](#)
 channels (directions), packet
 column, [305](#)
 channels assigned (bus/signal), [83, 89](#)
 channels assigned (bus/signal),
 reordering, [91](#)
 channels on a single time base and
 trigger, [593, 600](#)
 channels per module, [593](#)
 channels, assigning in default bit
 order, [88](#)

- channels, assigning in selected bit order, **89**
- channels, data on unassigned, transitional timing, **365**
- character names, **299**
- characteristics, **587, 611**
- characteristics, 16800-series logic analyzer, **600**
- chart, viewing Waveform bus data as, **224**
- chassis, **614**
- Chat dialog, **429**
- chat messages, **67**
- Chat Select Destination dialog, **430**
- Choose a Protocol Family and Bus dialog, **430**
- clear favorite/recently used triggers, **453**
- clear flag, **157**
- clear flag action, **180**
- clear trigger sequence, **185**
- clearing a remote access password, **68**
- clearing the trigger sequence, **424**
- clip board, copying
 - module/tool/window status to, **469**
- clip board, copying screens to, **315**
- clip board, copying text to, **314**
- clipped (show) analog property, **234**
- clock channel, **88, 89, 614**
- CLOCK IN BNC, **614**
- clock mode (state sampling), Demultiplex, **103, 115**
- clock mode (state sampling), Dual Sample, **106**
- clock mode (state sampling), Master Only, **100, 114**
- clock mode (state sampling), Master/Slave, **101, 114**
- clock mode (state sampling), selecting, **99**
- clock pod/channel naming, **361**
- clock qualifier hold time, **600**
- clock qualifier setup time, **600**
- clock qualifiers, **600**
- clock setup, state mode, **334**
- clock threshold voltage, **77**
- clock, slow or missing, **339**
- clock, state mode sampling, **358, 359**
- clock, state mode sampling, setting up, **124**
- clock, timing mode sampling, **356**
- clocks, **600**
- clocks pod, **89**
- Coff/Stabs symbol files, **136**
- collapse a bus, **227**
- color, difference
 - foreground/background, **276**
- color, filtered data in Waveform window, **229**
- color, Listing window background, **239**
- color, overlaid waveform, **228**
- color, reference data background, **276**
- color, selected packet box, **303**
- color, Source window background, **282**
- color, Source window text, **282**
- color, waveform, **231**
- color, Waveform window background, **227**
- colorize symbols, **133**
- colorize/filter tool, **390**
- column alignment in Listing window, **241**
- column autosize, **241**
- column base in Listing window, **237**
- column channels (directions), protocol, **288**
- column headings in Waveform window, **226**
- column properties, bus/signal, **240**
- column properties, packet summary, **304**
- COM automation, use of XML format, **380**
- COM/DCOM, extending capture capabilities, **206**
- combining logic analyzer cards, **72**
- commands, keyboard, **532**
- comma-separated value files, **199, 504**
- comments, **94**
- comments, bus/signal, **83, 426**
- comments, General-Purpose ASCII symbol file format, **519**
- comments, If clause, displaying/hiding, **183**
- comments, marker, **257**
- comments, system in use, **63, 65, 459**
- communication timeout, floating license server, **342**
- compare differences, run until a number, **275**
- Compare menu, **394**
- compare operation keyboard commands, **532**
- Compare Overview bar, **274**
- compare packets, **294**
- Compare window, **411**
- Compare window properties, **276**
- Compare window, new, **396**
- compared data, finding differences, **274**
- comparing a range of samples, **275**
- comparing captured data, **274**
- compatible modules, **380**
- complex bus/signal patterns, finding, **270**
- complex triggers, setting up, **378**
- concepts, **317, 353**
- conclusions, understanding logic analyzer triggering, **379**
- conditions in trigger sequence, **371**
- configuration file header information, **208**
- configuration file, ALA format, **504**
- configuration file, creating new after offline startup, **447**
- configuration file, mapping modules from, **468**
- configuration file, new/open/save, **398**
- configuration file, offline creation of, **431**
- configuration file, opening, **208**
- configuration file, opening after offline startup, **447**
- configuration file, recently-used, **210**
- configuration file, saving, **198**
- configuration files, default folders, **447**
- configuration files, loading/saving, **46**
- configuration files, new/open/save, **386**
- configuration files, trigger setup, **194**
- configuration files, won't open, **333**
- configuring counters, **154**
- configuring timers, **153**
- connect samples analog property, **234**
- connect samples in chart, **224**
- connect to logic analysis system, **459**
- connecting multiple frames, **381**
- connecting to a LAN, **315**
- connection attributes, external oscilloscope, **399**
- connection rules, **418**
- connections to device under test, **474**
- connections, deleting/adding, **307**
- connectivity, **315**
- connectorless probing, Soft Touch, **57**
- consecutive files, storing repetitive run data, **206**
- contact information, **351**
- content information, configuration file, **208**
- convert trigger step to if/then, **183**
- conveyor belt analogy, **369**
- copy data to Compare reference buffer, **394**
- copy to clip board, **387, 398**
- copying data to reference buffer, **274**
- copying module/tool/window status to clip board, **469**
- copying screens to clip board, **315**
- copying text to clip board, **314**
- copyright information, **397**
- correlated source line, showing, **279**
- Correlation Bus tab, **468**
- correlation bus, selecting, **280**
- counter action, **179, 180**
- counter event, **168, 169, 337**
- counter event check without increment action, **339**
- counter in trigger sequence, **337**
- counters, **152**
- counters, triggering capability, **370**
- counters, using, **154**
- counters, using in triggers, **373**
- creating new markers, **246**
- CSV file (module) header format, **505**

CSV format, [199, 504](#)
 CSV format files, offline analysis in
 Excel, [218](#)
 CSV module file format, [201, 504, 510](#)
 custom default storage, [167](#)
 customizing toolbars, [402](#)
 cut to clip board, [387, 398](#)
 cutting trigger sequence steps, [182](#)

D

data and setup, loading, [208](#)
 data channel, [614](#)
 data formats, [504](#)
 data import module setup, [442](#)
 data import module,
 buses/signals, [442](#)
 data import module, file
 information, [215, 443](#)
 data import modules, [211](#)
 Data Import toolbar, [400](#)
 data on unassigned channels,
 transitional timing, [365](#)
 data storage, transitional timing, [365](#)
 data valid window, [367](#)
 data values in waveform,
 showing/hiding, [232](#)
 data, buses/signals to export, [452](#)
 data, comparing captured to
 reference, [274](#)
 data, copying to reference buffer, [274](#)
 data, exporting to module binary (ALB)
 files, [203](#)
 data, exporting to module CSV
 files, [201](#)
 data, finding bus/signal patterns
 in, [264, 265](#)
 data, finding compare differences, [274](#)
 data, loading saved, [208](#)
 data, print to file, [314](#)
 data, printing, [314, 451](#)
 data, range to export, [452](#)
 data, reference, offset, [275](#)
 data, saving, [198](#)
 data, viewing saved, [380](#)
 date information, configuration
 file, [208](#)
 Decimal number base, [159, 232, 241](#)
 default base, [93](#)
 default base (bus/signal), [83](#)
 default bit order, assigning
 channels, [88](#)
 default bit order, resetting to, [89, 91](#)
 default folders, [447](#)
 default number base, [426](#)
 default storage, [152, 167, 375, 424, 614](#)
 default storage, action to turn
 on/off, [181](#)
 default trigger sequence, [424](#)
 defining probes, [474](#)
 delay, [614](#)
 delay from trigger in waveform
 display, [222](#)
 delay, adjustable, on state sampling
 clock input, [367](#)
 delay, repetitive run, [447](#)
 delete Compare window, [394](#)
 delete event, [168](#)
 delete external scope, [389](#)
 Delete Find Event, [270](#)
 delete Listing window, [393](#)
 delete logic analysis system, [459](#)
 delete marker, [391](#)
 delete Overview window, [393](#)
 delete Source window, [395](#)
 delete Waveform window, [394](#)
 delete, Overview window, [311](#)
 deleting a bus or signal, [84](#)
 deleting actions, [179](#)
 deleting buses/signals from Listing
 window, [238](#)
 deleting buses/signals from Waveform
 window, [227](#)
 deleting connections, [307](#)
 deleting display windows, [307, 313](#)
 deleting favorite find patterns, [271](#)
 deleting markers, [250](#)
 deleting system from use list, [61](#)
 deleting tools, [307](#)
 deleting trigger sequence steps, [182](#)
 demangled symbols, [513](#)
 Demo Center, [55](#)
 demo data, creating when offline, [447](#)
 Demultiplex state sampling clock
 mode, [103, 115](#)
 depth, acquisition memory, [128, 129,](#)
 [139, 428, 545, 549, 553, 600](#)
 description information, configuration
 file, [208](#)
 description, configuration file, [198](#)
 details, module/tool/window, [469](#)
 details, trigger, [453](#)
 details, viewing system, [64](#)
 device under test, [614](#)
 device under test connection, [474](#)
 device under test, connecting, [41](#)
 device under test, controlling signals
 in, [58](#)
 device under test, loading in the, [334](#)
 device under test, probing, [57](#)
 device under test, remote control for
 switches, [475](#)
 dialogs, [421](#)
 difference foreground/background
 color, [276](#)
 difference pair reference data, [276](#)
 differences in compared data,
 finding, [274](#)
 differential probes and single-ended
 signals, [474](#)
 differential threshold voltage level, [77](#)
 directories, source file, [277, 467](#)
 directory list, source file, [279](#)

disabling module, [72, 389](#)
 disk space, freeing, [328](#)
 display center rectangle Listing
 window, [239](#)
 display line numbers, [282](#)
 display scale, changing the, [221](#)
 display window properties,
 editing, [310](#)
 display windows, [404](#)
 display windows, adding/deleting, [313](#)
 display windows,
 adding/deleting/duplicating, [307](#)
 display windows, lock scrolling, [229,](#)
 [239](#)
 display windows, renaming, [311](#)
 display windows, using, [313](#)
 displaying symbols, [243](#)
 displaying the Overview window, [307](#)
 documenting trigger sequences, [378](#)
 don't care, [614](#)
 don't store sample action, [181](#)
 Dont Care signal pattern value, [159](#)
 double-click, [614](#)
 drag and drop, [615](#)
 drag and drop markers, [247](#)
 draw box triggers, [143](#)
 draw rectangle in data, [221](#)
 Dual Sample state sampling clock
 mode, [106](#)
 dual sample state sampling clock
 mode, [106, 116](#)
 duplicate Compare window, [394](#)
 duplicate Listing window, [393](#)
 duplicate Source window, [395](#)
 duplicate Waveform window, [394](#)
 duplicating display windows, [307](#)
 duration counts, [178](#)
 duration of storage, transitional
 timing, [365](#)
 dynamic triggers, [206](#)

E

early return of borrowed floating
 licenses, [326, 465](#)
 edge, [615](#)
 edge and pattern trigger function, [480](#)
 edge followed by edge trigger
 function, [480](#)
 edge followed by pattern trigger
 function, [481](#)
 edge operator, pattern trigger, [159](#)
 edge resources, [337](#)
 edge trigger, [146, 357](#)
 edge trigger function, [479](#)
 edge trigger functions, timing
 mode, [478](#)
 edge, triggering on signal, [148](#)
 edges on a bus, triggering on, [147](#)
 edges too far apart trigger
 function, [481](#)

- edges, center waveform display
 - about, [221](#)
 - edges, state clock, too close, [334](#)
 - edges, triggering capability, [370](#)
 - edges, triggering on, [373](#)
 - Edit Application dialog, [436](#)
 - Edit menu, [387](#)
 - edit operation keyboard
 - commands, [532](#)
 - editing window/tool properties, [310](#)
 - Either Edge signal edge value, [159](#)
 - either edge, triggering on, [373](#)
 - elastomeric probe adapter, [57](#)
 - ELF/DWARF object file format, [513](#)
 - ELF/stabs object file format, [513](#)
 - ELF/Stabs symbol files, [136](#)
 - else if branches, [373](#), [488](#), [489](#), [500](#), [501](#), [502](#)
 - E-mail dialog, [432](#)
 - e-mail on trigger, [166](#)
 - e-mail, send on compare
 - differences, [275](#)
 - e-mail, send on interval marker
 - statistic, [251](#), [252](#)
 - e-mail, send when marker value
 - found, [247](#)
 - enable channel order selection, [89](#)
 - enable module, [389](#)
 - enabling modules, [72](#)
 - end of data, [222](#)
 - End of Data marker, [312](#)
 - end of data, go to, [249](#)
 - end of data, go to in Listing
 - window, [236](#)
 - end value of symbol, [458](#)
 - endian, [91](#)
 - English online help, [397](#)
 - Entering, find qualifier, [265](#)
 - equal to operator, pattern trigger, [147](#), [159](#)
 - error dialogs, showing/hiding, [447](#)
 - error messages, [334](#)
 - error messages in the status log, [469](#)
 - evaluation order of events, [177](#)
 - evaluation order of find events, [270](#)
 - event, [615](#)
 - event checks without action, [339](#)
 - Event Editor dialog, [433](#)
 - event logging, [447](#)
 - event statements, [153](#), [154](#), [157](#)
 - event statements, reading, [153](#)
 - event, arm in from, [170](#)
 - event, counter, [169](#)
 - event, flag, [170](#)
 - event, timer, [169](#)
 - events (advanced trigger),
 - grouping, [177](#)
 - events and actions, trigger, [152](#)
 - events in trigger sequence, [424](#)
 - events, duration or occurrence counts
 - in timing mode, [178](#)
 - events, inserting or deleting, [168](#)
 - events, negating, [177](#)
 - excluding bus/signal transitions, [167](#)
 - Exiting, find qualifier, [265](#)
 - expand a bus, [227](#)
 - expand bus into signal waveforms, [224](#)
 - experienced users, tips for, [39](#)
 - export bus/signal data to CSV file, [199](#)
 - export CSV files, [386](#)
 - Export dialog, [434](#)
 - Export file selection dialog, [435](#)
 - export files, default folders, [447](#)
 - exporting a range of data, [452](#)
 - exporting data, [199](#)
 - exporting data from selected
 - buses/signals, [452](#)
 - exporting data to module binary (ALB)
 - files, [203](#)
 - exporting data to module CSV
 - files, [201](#)
 - expressions, boolean, [372](#)
 - External Application Setup dialog, [436](#)
 - external arm, wait for, [486](#), [498](#)
 - external instrument, trigger in
 - from, [192](#)
 - external instrument, triggering, [437](#)
 - external instruments, trigger out
 - to, [191](#)
 - External Oscilloscope Setup
 - toolbar, [399](#)
 - external scope, add/delete, [389](#)
 - external trigger, [170](#), [186](#), [389](#), [615](#)
 - External Trigger dialog, [437](#)
 - extra data channels, [88](#), [89](#)
- ## F
- Falling Edge signal edge value, [159](#)
 - falling edge, triggering on, [373](#)
 - fan out/in to tools in Overview
 - window, [418](#)
 - Fast Zoom In option for Waveform
 - windows, [230](#)
 - favorite find pattern, [271](#)
 - favorite packet events, organizing, [165](#)
 - favorite packet events, saving, [165](#)
 - favorite trigger, [194](#), [195](#), [453](#)
 - feedthrough mode, external
 - trigger, [437](#)
 - field decode, [299](#)
 - field patterns (packet), finding, [267](#)
 - file errors, [336](#)
 - file format, [218](#), [504](#)
 - file format, module binary (ALB), [203](#), [511](#)
 - file format, module CSV, [201](#), [504](#), [510](#)
 - file format, netlist, [90](#)
 - file information in data import
 - module, [443](#)
 - File menu, [386](#)
 - file operation keyboard
 - commands, [532](#)
 - file, print to, [314](#)
 - file, symbol, [458](#)
 - files (configuration), opening, [208](#)
 - files (configuration),
 - recently-used, [210](#)
 - files (configuration), saving, [198](#)
 - files (configuration), translating 167xG
 - and 16700-series, [198](#)
 - files (configuration), won't open, [333](#)
 - files, source, browsing, [278](#)
 - files, symbol, [471](#)
 - fill memory action, [166](#)
 - fill memory storage qualifiers, [167](#)
 - filling memory after trigger, [340](#)
 - filter/colorize tool, [390](#)
 - filtered data color, Waveform
 - window, [229](#)
 - find, [387](#), [390](#)
 - find a packet trigger function, [496](#)
 - find anything N times trigger function,
 - state mode, [497](#)
 - find anything N times trigger function,
 - timing mode, [484](#)
 - Find Bus/Signal dialog, [224](#), [237](#)
 - find data, [398](#)
 - Find dialog, [438](#)
 - find events, [270](#)
 - Find Options dialog, [272](#)
 - find patterns, favorites, [271](#)
 - find qualifiers, [265](#)
 - find symbol name, [458](#)
 - find, quick, [264](#)
 - FindEdges macro, [391](#)
 - finding bus/signal patterns, [264](#), [265](#)
 - finding complex bus/signal
 - patterns, [270](#)
 - finding differences in compared
 - data, [274](#)
 - fixed unit for time column, Listing
 - window, [242](#)
 - fixed unit for time column, Protocol
 - Viewer window, [306](#)
 - flag action, [179](#), [180](#)
 - flag action delay, [170](#)
 - flag event, [168](#), [170](#)
 - flag line used for arming in multiframe
 - systems, [381](#)
 - flags, [152](#), [373](#)
 - flags status, [469](#)
 - flags, using, [157](#)
 - FLEXlm License Finder dialog, [461](#)
 - floating license server communication
 - timeout, [342](#)
 - floating license servers, [324](#), [464](#)
 - floating licenses, [321](#)
 - flying lead probe type, [474](#)
 - folder (bus/signal), adding, [95](#)
 - folders, default, [447](#)
 - font size, Listing window, [238](#), [239](#)
 - font size, Protocol Viewer, [303](#)
 - font size, Source window, [282](#)
 - font size, waveform, [229](#)
 - font size, Waveform window, [227](#)

footers when printing, [314](#)
 force prestore, [128](#)
 foreground color, marker, [246, 256](#)
 format, netlist file, [90](#)
 formats, data, [504](#)
 Found marker placement, [272](#)
 frame, [615](#)
 Frame Module Information dialog, [439](#)
 frame that module is in, [474](#)
 frame, connecting to, [459](#)
 frame, slots/cards/modules in, [472](#)
 frequency of changes in time
 interval, [251](#)
 front panel operation, 16880-series
 logic analyzers, [527](#)
 full channel timing mode, [97, 545, 550, 553](#)
 full screen, [388](#)
 full screen display, [398](#)
 function names, [132](#)
 function tests, [611](#)
 FUNCTIONS, General-Purpose ASCII
 symbol file format, [516](#)

G

general purpose knob, 16800-series
 logic analyzer, [527, 531](#)
 general purpose probing, [57](#)
 general state mode, [98, 553](#)
 General-Purpose ASCII (GPA) symbol
 file, [135, 471, 514](#)
 General-Purpose ASCII (GPA) symbol
 file format, [514](#)
 generic text only printer, [314](#)
 getting started, [39](#)
 glitch drawing, [147, 148](#)
 Glitch signal edge value, [159](#)
 glitch, triggering on signal, [148](#)
 glitches on a bus, triggering on, [147](#)
 global counter, [155](#)
 global counter status, [469](#)
 global counters, [373](#)
 global counters, transitional
 timing, [365](#)
 glossary, [613](#)
 go offline, [386](#)
 go online, [386](#)
 go to action, [371](#)
 go to beginning/end/trigger, [387, 398](#)
 go to here (in markers overview
 bar), [245](#)
 go to locations in displayed packet
 data, [286](#)
 go to locations in Listing window, [236](#)
 go to locations in Waveform
 window, [222](#)
 go to marker, [391, 400](#)
 Go To Marker dialog, [249](#)
 go to trigger on run, [196, 447](#)
 goto actions, [166](#)

GPA (General-Purpose ASCII) symbol
 file, [135, 471](#)
 GPA (General-Purpose ASCII) symbol
 file format, [514](#)
 gray bus and signal icons, [83, 84](#)
 greater than duration, [600](#)
 greater than operator, pattern
 trigger, [147, 159](#)
 greater than or equal to operator,
 pattern trigger, [147, 159](#)
 grid style/color analog property, [234](#)
 group events (advanced trigger), [177](#)
 group signals into bus, [227](#)
 grouping, numeric, [447](#)

H

half channel timing mode, [97, 545, 550, 553](#)
 hardware errors, [337](#)
 Hardware Update Utility program, [425](#)
 hardware upgrades, installing, [139](#)
 hardware_log.txt file, [336](#)
 hardware, selecting offline, [431](#)
 header (packet), viewing, [297](#)
 header format in module CSV and
 module binary files, [505](#)
 headers when printing, [314](#)
 height, waveform row, [232](#)
 help, [40](#)
 help file errors, [337](#)
 Help menu, [397](#)
 help operation keyboard
 commands, [532](#)
 help, Japanese, accessing on Windows
 XP, [345](#)
 Hex number base, [159, 232, 241](#)
 hide marker measurement display
 bar, [400](#)
 hide measurements list, [391](#)
 hiding markers, [256](#)
 hiding source file line numbers, [282](#)
 hiding/showing data values in
 waveforms, [232](#)
 High signal pattern value, [159](#)
 high-level source code, viewing, [277](#)
 history depth, trigger, [194, 195](#)
 host offline status, [63](#)
 hostname, logic analysis system
 frame, [61, 459](#)
 humidity, [595, 600](#)

I

I/O channel error, [337](#)
 IEEE 1394 interface card/cable, [332](#)
 IEEE-695 object file format, [513](#)
 If clause comments,
 displaying/hiding, [183](#)
 if/then, show trigger step as, [376](#)
 Import dialog, [442](#)

import module setup, [442](#)
 import module toolbar, [400](#)
 import module, buses/signals, [442](#)
 import module, file information, [443](#)
 import netlist, [90](#)
 import netlist to define
 buses/signals, [446](#)
 Import Setup dialog, [442](#)
 in range operator, pattern trigger, [147, 159](#)
 in use status, [63](#)
 incompatible remote service, [332](#)
 incompatible remote service version
 status, [63](#)
 increment counter action, [180](#)
 independent analyzers, [600](#)
 information messages in the status
 log, [469](#)
 informational dialogs,
 showing/hiding, [447](#)
 informational messages, [340](#)
 input only reference data, [276](#)
 insert action after, [179](#)
 insert action before, [179](#)
 insert bus/signal into window, [387](#)
 insert event after (AND/OR), [168](#)
 insert event before (AND/OR), [168](#)
 Insert Find Event After (AND/OR), [270](#)
 Insert Find Event Before
 (AND/OR), [270](#)
 inserting buses/signals into Listing
 window, [238](#)
 inserting buses/signals into Waveform
 window, [227](#)
 inserting trigger functions into trigger
 sequence steps, [158](#)
 installation guides, [351](#)
 installing a printer, [315](#)
 installing on to LAN, [315](#)
 interactive self tests, [343](#)
 interpreting error messages, [334](#)
 intersecting Y-Axis and time
 markers, [259](#)
 interval marker statistics, [251, 252](#)
 interval measurement, [403](#)
 inverse assembler, [615](#)
 inverse assembler tool, [277](#)
 inverse assembler tool, Software
 Address bus, [280](#)
 inverse assembler, locked
 bus/signals, [83](#)
 inverse assembly tools, new, [390](#)
 IP address, logic analysis system
 frame, [61, 459](#)

J

Japanese help, accessing on Windows
 XP, [345](#)
 Japanese online help, [397](#)

K

keyboard commands, [532](#)
 keyboard, on-screen, [198, 208](#)
 knob, 16800-series logic analyzer, [527, 531](#)

L

LAN, connecting to, [315](#)
 lanes (packet), viewing, [299](#)
 lanes associated with packet, viewing, [299](#)
 language options, Windows XP, [345](#)
 language, help, [397](#)
 latency, timer reset, [153](#)
 left alignment, bus/signal columns, [241](#)
 less than duration, [600](#)
 less than operator, pattern trigger, [147, 159](#)
 less than or equal to operator, pattern trigger, [147, 159](#)
 level, triggering on signal, [148](#)
 license activation wizard, [323, 463](#)
 license file, [139](#)
 license not available, [342](#)
 license problems, [342](#)
 license server communication timeout, [342](#)
 license servers, [321](#)
 licensed hardware upgrades, installing, [139](#)
 licenses (software), activation, [323, 463](#)
 licenses (software), borrow floating, [326, 465](#)
 licenses (software), floating license servers, [324, 464](#)
 licenses (software), summary, [322, 461](#)
 Licensing dialog, [461](#)
 licensing host ID, [323, 463](#)
 licensing, software, [397](#)
 limits in chart, [224](#)
 line number symbols, [277, 280](#)
 line number symbols, source correlation bus, [468](#)
 line numbers in Source window, [467](#)
 line numbers, source file, showing/hiding, [282](#)
 listing center rectangle, go to source line, [279](#)
 Listing menu, [393](#)
 listing operation keyboard commands, [532](#)
 Listing window, [409](#)
 Listing window, analyzing data, [235](#)
 Listing window, background color, [239](#)
 Listing window, bus/signal column properties, [240](#)
 Listing window, bus/signal data color, [240](#)
 Listing window, center rectangle, [239](#)
 Listing window, column width, [241](#)
 Listing window, font size, [239](#)
 Listing window, go to locations in, [236](#)
 Listing window, inserting/deleting buses/signals, [238](#)
 Listing window, lock scrolling, [239](#)
 Listing window, marker-relative times, [242](#)
 Listing window, new, [396](#)
 Listing window, properties, [238](#)
 Listing window, Quick Trigger in, [144](#)
 Listing window, re-arranging bus/signal columns, [237](#)
 Listing window, showing/hiding parts of, [237](#)
 Listing window, timing zoom background color, [239](#)
 little endian, [91](#)
 loading in the device under test, [334](#)
 loading saved data and setups, [208](#)
 local area network, [315](#)
 Local online mode, [59](#)
 local printer, [315](#)
 local session, [332](#)
 locate bus/signal patterns, [264](#)
 lock chart limits to setup, [224](#)
 lock display window scrolling, [227, 238](#)
 lock in viewer, marker property, [256](#)
 lock scrolling, [229, 239](#)
 lock to relative marker, [257](#)
 locked bus/signal, [83, 84, 334, 426](#)
 lockstep, [293](#)
 lockstep window, Protocol Viewer, [303](#)
 log, status, [469](#)
 logic analysis system details, [439](#)
 logic analysis system self tests, [343](#)
 logic analysis system, adding, [61](#)
 logic analysis system, deleting, [62](#)
 logic analysis system, going online, [459](#)
 logic analysis systems, refresh list, [63](#)
 logic analyzer, [615](#)
 logic analyzer basics, [40](#)
 logic analyzer cards, combining, [72](#)
 logic analyzer memory, [369](#)
 logic analyzer notes, [536](#)
 logic analyzer pod and channel connections, [474](#)
 logic analyzer triggering, understanding, [369](#)
 Logic Analyzer Upgrade utility, [425](#)
 logic analyzer, setting up, [71](#)
 logic analyzer, turning on, [40](#)
 logic analyzer, what is, [356](#)
 logic analyzer, when should I use, [355](#)
 logic analyzers, offline analysis on, [217](#)
 long marker name, [246, 254](#)
 Low signal pattern value, [159](#)
 LTSSM States and Transitions, [300](#)

M

macro, [615](#)
 macro (Visual Basic), [401](#)
 macro, COM/DCOM, [390](#)
 mail applications, [432](#)
 manage releases, [328](#)
 manually mapping modules, [444](#)
 mapping modules, [210, 444](#)
 mapping modules from configuration file, [468](#)
 margins when printing, [314](#)
 marker, [615](#)
 marker background color, [246, 256](#)
 marker comments, [257](#)
 marker display bar, [246, 254](#)
 marker for system trigger, [312](#)
 marker foreground color, [246, 256](#)
 marker lock in viewer setting, [256](#)
 marker measurement display bar, [403](#)
 marker measurements, sample interval, [252](#)
 marker measurements, time interval, [251](#)
 marker pair, center about, [249](#)
 marker position, [246](#)
 marker properties, comments, [257](#)
 marker properties, lock to relative, [257](#)
 marker properties, position, [247](#)
 marker properties, snap to edge, [247, 250](#)
 marker quick measurements, [47](#)
 marker tool tip, [246, 254, 257](#)
 marker, Found, [272](#)
 marker, go to in Waveform window, [222](#)
 marker-relative times in Listing window, [241, 242](#)
 marker-relative times, Protocol Viewer, [306](#)
 markers, [245](#)
 markers display bar, [407](#)
 markers display bar in Waveform window, [226](#)
 markers display bar, reading, [245](#)
 markers display in Listing window, [237](#)
 Markers menu, [391](#)
 markers overview bar, [408](#)
 markers overview bar, reading, [245](#)
 Markers toolbar, [400](#)
 markers, abbreviated name, [246](#)
 markers, creating new, [246](#)
 markers, deleting, [250](#)
 markers, go to, [249](#)
 markers, go to in Listing window, [236](#)
 markers, hiding/showing, [256](#)
 markers, long name, [246](#)
 markers, placing in data, [247](#)
 markers, placing where data is found, [272](#)
 markers, properties, [255](#)
 markers, renaming, [254](#)

markers, sending to back, [254](#)
 markers, using, [44](#)
 markers, value at measurements, [253](#)
 master clock, [100, 101, 103, 106, 114, 115, 116](#)
 master frame, [381](#)
 Master Only state sampling clock mode, [100, 114](#)
 master pods, [450](#)
 Master/Slave state sampling clock mode, [101, 114, 127](#)
 maximized, starting, [447](#)
 maximum 128 channels per bus, [334](#)
 maximum channels on a single time base and trigger, [593, 600](#)
 maximum global counter, [600](#)
 maximum occurrence counter, [600](#)
 maximum pattern width, [600](#)
 maximum range width, [600](#)
 maximum state clock speed, [334](#)
 maximum state data rate on each channel, [600](#)
 maximum state sampling speed, [553](#)
 maximum time count between stored states, [600](#)
 maximum transitions stored, [365](#)
 maximum trigger sequence steps, [600](#)
 measure time, [245](#)
 measurement data, printing, [451](#)
 measurement examples, [50, 51](#)
 memory depth, [128, 129, 139, 428, 553, 600](#)
 memory depth and channel count trade-offs, [363](#)
 memory depth in full channel mode, [600](#)
 memory depth in half channel mode, [600](#)
 memory, logic analyzer, [369](#)
 menu (Tools), adding external applications, [436](#)
 menu access keyboard commands, [532](#)
 menu bar, [615](#)
 menu, File, [386](#)
 menus, [386](#)
 message dialog options, [447](#)
 messages, borrow licenses, [466](#)
 messages, error, [334](#)
 messages, return borrowed licenses, [466](#)
 minimum data pulse width, [600](#)
 minimum detectable glitch, [148](#)
 minimum master-to-slave clock time, [600](#)
 minimum PC requirements for offline analysis, [218](#)
 minimum slave-to-master clock time, [600](#)
 minimum slave-to-slave clock time, [600](#)
 minimum state clock pulse width, [600](#)

minimum time between active clock edges, [600](#)
 minimum transitions stored, [365](#)
 miscellaneous keyboard commands, [532](#)
 missing or slow clock, [339](#)
 missing pods, [362](#)
 module, [616](#)
 module binary (ALB) file format, [203, 511](#)
 module binary and module CSV file header format, [505](#)
 Module binary file, [442](#)
 module binary file, [435](#)
 module CSV file format, [201, 504, 510](#)
 Module CSV text file, [442](#)
 module CSV text file, [435](#)
 module details, [469](#)
 module mapping, [210](#)
 Module Mapping dialog, [444](#)
 module setup information, transferring, [380](#)
 module setup, mapping from configuration file, [468](#)
 module setups, transferring, [210](#)
 Module Skew and System Trigger dialog, [446](#)
 Module Summary tab, [474](#)
 module upgrade, [397](#)
 module, cancel processing, [469](#)
 module, disable/enable/rename, [389](#)
 module, number of cards in, [431](#)
 module, resume canceled processing, [469](#)
 module, splitting, [72](#)
 modules in frame, [472](#)
 modules only, loading, [208](#)
 modules, configuring, [72](#)
 modules, disabling/enabling, [72](#)
 modules, renaming, [311](#)
 mouse cursor, placing marker at, [247](#)
 mouse shortcuts, [49](#)
 moving markers, [247](#)
 MSB ordering in bus waveform, [224](#)
 multiframe, [381](#)
 multiframe cable, [381](#)
 multiframe logic analysis system, flags, [157](#)
 multiframe remote access passwords, [68](#)
 multi-module systems, transferring setups, [210](#)
 multiple clocks, [99, 125, 154, 158](#)
 multiple conditions, testing in trigger sequence, [373](#)
 multiple frames, [381](#)

N

N consecutive samples with pattern1 trigger function, [491](#)

N number of edges trigger function, [479](#)
 name, symbol, [458](#)
 negating events, [177](#)
 negative logic, [94](#)
 netlist file format, [90](#)
 Netlist Import dialog, [446](#)
 Netlist Import folder, [90](#)
 netlist, importing, [90](#)
 network, [315](#)
 network printer, [315](#)
 network troubleshooting, [348](#)
 new Compare window, [396](#)
 new configuration file, [386, 398](#)
 new Listing window, [396](#)
 new marker, [391, 400](#)
 new sample interval measurement, [391, 400](#)
 new Source window, [396](#)
 new time interval measurement, [391, 400](#)
 new toolbar, creating, [402](#)
 new value at measurement, [391, 400](#)
 new Waveform window, [396](#)
 next data for source line, [278](#)
 next difference, [274](#)
 next difference in Compare window, [394](#)
 next source line, [277](#)
 next window, [396, 398](#)
 no hardware found, [139](#)
 no matching symbol found, [279](#)
 node-locked licenses, [321](#)
 not equal to operator, pattern trigger, [147, 159](#)
 not in range operator, pattern trigger, [147, 159](#)
 not in range, triggering on, [373](#)
 Not Present for Range, find qualifier, [265](#)
 Not Present, find qualifier, [265](#)
 notes, 16800-series logic analyzer, [553](#)
 notes, 16962 logic analyzer, [544, 549](#)
 notes, logic analyzer, [536](#)
 nothing event, [168](#)
 number base, [93, 159, 232, 241](#)
 number base, default (bus/signal), [83](#)
 number base, packet column, [306](#)
 numeric data values, showing/hiding, [232](#)
 numeric grouping, [447](#)

O

object file formats supported by symbol reader, [513](#)
 object file symbols, [132](#)
 object file symbols, converting to ASCII, [135](#)
 observed value on target control port signals, [475](#)

- occurrence, [616](#)
 - occurrence counter status, [469](#)
 - occurrence counter, reset action, [179](#), [180](#)
 - occurrence counters, [154](#), [178](#), [373](#)
 - Octal number base, [159](#), [232](#), [241](#)
 - offline analysis, [208](#), [217](#), [218](#), [616](#)
 - offline analysis warnings, [339](#)
 - offline file formats, [218](#)
 - offline hardware, selecting, [431](#)
 - offline mode, [444](#)
 - offline mode unexpected, [332](#)
 - Offline Startup Options dialog, [447](#)
 - offline, creating demo data when, [447](#)
 - offline, go, [386](#)
 - offset in Compare window, [394](#)
 - offset, adding to symbol value, [458](#)
 - offset, reference data, [275](#)
 - OMF96 object file format, [513](#)
 - OMFx86 object file format, [513](#)
 - ONFi Analysis window, [416](#)
 - online analysis, [59](#)
 - online help, [397](#), [398](#)
 - online, go, [386](#)
 - online, going after offline startup, [447](#)
 - on-screen keyboard, [198](#), [208](#)
 - open configuration file, [386](#), [398](#)
 - opening configuration files, [208](#), [333](#)
 - opening the Overview window, [307](#)
 - option attributes, external
 - oscilloscope, [399](#)
 - options, [387](#)
 - options, symbol reader, [136](#)
 - OR events, [168](#)
 - or trigger, [186](#)
 - OR'ed find events, [270](#)
 - order, channel selection, [89](#)
 - oscilloscope, when should I use, [354](#)
 - other trigger functions, state mode, [490](#)
 - other trigger functions, timing mode, [478](#)
 - overlaid waveform color, [228](#)
 - overlying waveforms, [223](#)
 - overview, [40](#)
 - Overview menu, [393](#)
 - Overview window, [307](#), [390](#), [418](#)
 - Overview window and multiframe systems, [381](#)
 - Overview window, delete, [311](#)
 - Overview window, open/display, [307](#), [396](#), [398](#)
 - Overview window, redraw, [311](#)
 - owner information, configuration file, [208](#)
 - owner, configuration file, [198](#)
- P**
- packet column, width, [305](#)
 - packet columns, inserting or deleting, [288](#)
 - packet columns, re-arranging, [287](#)
 - packet data, analyzing, [283](#)
 - packet decode column, alignment, [305](#)
 - packet decode column, channels (directions), [305](#)
 - packet decode column, number base, [306](#)
 - packet details, viewing and comparing, [294](#)
 - packet event editor, [164](#)
 - packet event, save as favorite, [165](#)
 - packet event, view as bits, [165](#)
 - packet events, organizing favorites, [165](#)
 - packet events, specifying, [164](#)
 - packet header, viewing, [297](#)
 - packet information, viewing, [293](#)
 - packet lanes, [299](#)
 - packet lanes, viewing, [299](#)
 - packet patterns, finding, [267](#)
 - packet payload, viewing, [298](#)
 - packet summary, [285](#)
 - packet summary event colors, [301](#)
 - packet summary, column properties, [304](#)
 - packet summary, showing/hiding parts of the display, [292](#)
 - packet, viewing selected, [293](#)
 - page headers/footers/margins when printing, [314](#)
 - parentheses in boolean expressions, [177](#)
 - partial logic analyzer setup, transferring, [380](#)
 - passwords for remote access, [68](#)
 - paste from clip board, [387](#), [398](#)
 - pattern, [616](#)
 - pattern absent for greater than time trigger function, [484](#)
 - pattern absent for less than time trigger function, [484](#)
 - pattern and pattern trigger function, state mode, [502](#)
 - pattern and pattern trigger function, timing mode, [489](#)
 - pattern N times trigger function, [491](#)
 - pattern or pattern trigger function, state mode, [503](#)
 - pattern or pattern trigger function, timing mode, [490](#)
 - pattern present for greater than time trigger function, [483](#)
 - pattern present for less than time trigger function, [483](#)
 - pattern resources, [337](#)
 - pattern too late after edge trigger function, [482](#)
 - pattern trigger, [147](#), [159](#), [357](#)
 - pattern trigger function, [482](#)
 - pattern1 followed by pattern2 before pattern3 trigger function, [493](#)
 - pattern1 followed by pattern2 trigger function, [492](#)
 - pattern1 immediately followed by pattern2 trigger function, [492](#)
 - pattern2 occurring too late after pattern1 trigger function, [496](#)
 - pattern2 occurring too soon after pattern1 trigger function, [495](#)
 - patterns trigger functions, state mode, [490](#)
 - patterns, finding bus/signal, [264](#), [265](#)
 - patterns, finding complex bus/signal, [270](#)
 - pause timer action, [179](#)
 - payload (packet), viewing, [298](#)
 - PC requirements, minimum for offline analysis, [218](#)
 - personal computers, offline analysis on, [218](#)
 - physical view of cards in system, [472](#)
 - pin mapping details, [90](#)
 - place marker on screen, [391](#), [400](#)
 - place on edge markers, [250](#)
 - placing markers in data, [247](#)
 - placing markers with find, [272](#)
 - pod, [616](#)
 - pod and channel connections, [426](#), [474](#)
 - pod assignment, [389](#)
 - Pod Assignment dialog, [450](#)
 - pod index, [616](#)
 - pod naming conventions, [361](#)
 - pod pair, [616](#)
 - pod pair, reserving for time tag storage, [363](#)
 - pod truncation, [208](#), [616](#)
 - Pods, missing, [362](#)
 - point, [616](#)
 - polarity, [83](#), [94](#), [426](#), [616](#)
 - polarity, external trigger, [437](#)
 - position marker property, [247](#)
 - position, marker, [246](#)
 - position, trigger, [369](#)
 - poststore, [128](#)
 - preprocessor, [57](#), [617](#)
 - Present for Range, find qualifier, [265](#)
 - Present greater than or equal to, find qualifier, [265](#)
 - Present greater than, find qualifier, [265](#)
 - Present less than or equal to, find qualifier, [265](#)
 - Present less than, find qualifier, [265](#)
 - Present, find qualifier, [265](#)
 - prestore, [128](#)
 - previous data for source line, [278](#)
 - previous difference, [274](#)
 - previous difference in Compare window, [394](#)
 - previous source line, [277](#)
 - previous window, [396](#), [398](#)
 - print data, [386](#), [398](#)
 - print to file, [314](#)
 - printer, installing, [315](#)

printing captured data, [314](#)
 Printing Data dialog, [451](#)
 probe, [617](#)
 probe compatibility, [595](#)
 probe grounding, [334](#)
 Probe Summary tab, [474](#)
 probes, [600](#)
 Probes toolbar, [400](#)
 probing, [57](#)
 probing documentation, [351](#)
 probing options, [57](#)
 problems, solving, [331](#)
 procedure names, [132](#)
 processing, module/tool/window,
 canceling or resuming, [469](#)
 processor-specific probes, [57](#)
 product information web sites, [351](#)
 product overviews, [520](#)
 project information, configuration
 file, [208](#)
 project, configuration file, [198](#)
 Properties dialog, [452](#)
 properties, Compare window, [394](#)
 properties, editing window/tool, [310](#)
 properties, Listing window, [238, 393](#)
 properties, marker, [255, 391, 400](#)
 properties, Source window, [281, 395,](#)
 [466](#)
 properties, Waveform window, [227,](#)
 [394](#)
 properties, window, [387](#)
 properties, Y-Axis marker, [262](#)
 protocol, [617](#)
 protocol column channels
 (directions), [288](#)
 Protocol Viewer, [413](#)
 Protocol Viewer menu, [395](#)
 Protocol Viewer panes,
 showing/hiding, [301](#)
 Protocol Viewer, window
 properties, [302](#)
 pulse clear flag action, [157, 180](#)
 pulse set flag action, [157, 180](#)
 pulse target control port signals, [475](#)
 pulse width in waveform tool tip, [230](#)
 pulse width, flag pulse set/clear, [180](#)
 pulsed mode, external trigger, [437](#)

Q

QFP package probing, [57](#)
 qualifier, state sampling clock, [124](#)
 quick find, [264](#)
 quick start for 16700-series users, [39](#)
 quick start guides, [351](#)
 quick trigger, [617](#)
 Quick Trigger alignment, setting, [282](#)
 Quick Trigger in Listing window, [144](#)
 Quick Trigger in Source window, [145](#)
 Quick Trigger in Waveform
 window, [143](#)
 quick triggers, [141, 143](#)

R

range and offset in Compare
 window, [394](#)
 range of data to export, [199](#)
 range of samples, comparing, [275](#)
 range operator, pattern trigger, [53,](#)
 [147, 159](#)
 range pattern, [617](#)
 Range Properties dialog, [452](#)
 ranges, triggering capability, [370](#)
 ranges, triggering on, [373](#)
 RDP (Remote Desktop Protocol), [350](#)
 reader, symbol, [136](#)
 readers.ini file, [136](#)
 reading markers display and overview
 bars, [245](#)
 readme information, [331](#)
 re-arranging bus/signal columns, [237](#)
 re-arranging waveforms, [223](#)
 re-assign pods to split analyzers, [450](#)
 recall recently-used configuration
 file, [210](#)
 recall trigger, [46, 194, 389](#)
 Recall Trigger dialog, [453](#)
 recalling favorite find patterns, [271](#)
 recalling triggers, [424](#)
 recent configuration files, [386](#)
 recent file list entries, [447](#)
 recently used bus/signal names, [159](#)
 recently-used configuration file,
 recalling, [210](#)
 recently-used trigger, [194, 195, 453](#)
 redraw, Overview window, [311, 393](#)
 reference, [385](#)
 reference buffer, copy data to, [274, 394](#)
 reference buffer, range compare, [275](#)
 reference data background color, [276](#)
 reference data offset, [275](#)
 reference data, comparing to, [274](#)
 reference data, display options, [276](#)
 refresh logic analysis system, [459](#)
 refreshing system to use list, [63](#)
 regional and language options,
 Windows XP, [345](#)
 re-group events (advanced
 trigger), [177](#)
 relative marker, lock to, [257](#)
 relative time, [241](#)
 remote access passwords, [68](#)
 remote control for switches in
 DUT, [475](#)
 Remote Desktop Protocol (RDP), [350](#)
 Remote Desktop, setting up, [350](#)
 Remote online mode, [59](#)
 remote session, [332](#)
 remove bus/signal from window, [387](#)
 remove icons to toolbar, [402](#)
 remove software, [328](#)
 rename bits of a bus, [86](#)
 rename Compare window, [394](#)
 rename Listing window, [393](#)
 rename module, [389](#)
 rename Source window, [395](#)
 rename trigger, [453](#)
 rename Waveform window, [394](#)
 renaming a bus or signal, [85](#)
 renaming markers, [254](#)
 renaming
 windows/tools/modules, [311](#)
 reorder bits, [91](#)
 reordering symbols, [132](#)
 repetitive measurement, [617](#)
 repetitive run, [196, 392, 401](#)
 repetitive run delay, [447](#)
 repetitive run, go to trigger
 behavior, [196](#)
 repetitive run, stop on compare
 differences, [275](#)
 repetitive run, stop on interval marker
 statistic, [251, 252](#)
 repetitive run, stop when marker value
 found, [247](#)
 repetitive run, storing data to
 consecutive files, [206](#)
 RepetitiveSaveToFile macro, [391](#)
 replacement failed, [337](#)
 replacing trigger functions in trigger
 sequence steps, [158](#)
 reporting level, self tests, [343](#)
 requirements, minimum PC for offline
 analysis, [218](#)
 reset and start timer trigger
 function, [497](#)
 reset counter action, [180](#)
 reset occurrence counter action, [154,](#)
 [179, 180, 337](#)
 reset toolbar, [402](#)
 resume, [392](#)
 resume canceled operation, [401](#)
 resume canceled processing,
 module/tool/window, [469](#)
 resume timer action, [179](#)
 return borrowed floating licenses
 early, [326, 465](#)
 return borrowed licenses
 messages, [466](#)
 reverse channel order, [91](#)
 right alignment, bus/signal
 columns, [241](#)
 right-click, [617](#)
 Rising Edge signal edge value, [159](#)
 rising edge, triggering on, [373](#)
 row height, [232](#)
 row properties, analog signal, [233](#)
 row properties, bus/signal
 waveform, [230](#)
 rules, Overview window
 connection, [418](#)
 run, [617](#)
 Run Properties dialog, [454](#)
 run repetitive, [617](#)
 run status, [196, 469](#)
 Run toolbar, [401](#)

- run until a number of compare differences, [275](#)
 - run until user stop trigger function, state mode, [498](#)
 - run until user stop trigger function, timing mode, [486](#)
 - run, go to trigger on, [196, 447](#)
 - run, repetitive, storing data to consecutive files, [206](#)
 - Run/Stop menu, [392](#)
 - run/stop operation keyboard commands, [532](#)
 - running self tests repetitively, [343](#)
 - running the analyzer, [196](#)
- ## S
- sample, [617](#)
 - sample interval measurement, [252, 403](#)
 - sample interval measurement, new, [391, 400](#)
 - sample period, [600, 617](#)
 - sample period, timing zoom, [131](#)
 - sample rate in half channel mode, [600](#)
 - sample rate on all channels, [600](#)
 - sample, go to in Listing window, [236](#)
 - sample, go to in Waveform window, [222](#)
 - sample, position marker by, [246, 247](#)
 - sampled data, [618](#)
 - samples, comparing range of, [275](#)
 - samples, how they are stored in memory, [369](#)
 - samples, storage qualification, [375](#)
 - sampling, [618](#)
 - sampling mode, state, [98](#)
 - sampling options, [97](#)
 - sampling options, memory depth and channel count trade-offs, [363](#)
 - sampling options, state, [98](#)
 - sampling period, [96, 98, 356](#)
 - sampling period, timing zoom, [476](#)
 - sampling period, transitional timing mode, [363](#)
 - sampling positions, state mode, understanding, [367](#)
 - sampling setup, [389, 399, 428](#)
 - Sampling tab, [428](#)
 - save configuration file, [386, 398](#)
 - saved data and setups, loading, [208](#)
 - saving captured data, [198](#)
 - saving configuration files, [198](#)
 - saving trigger sequences, [378](#)
 - saving trigger setups, [46](#)
 - scale of waveform display, [249](#)
 - scaling analog property, [234](#)
 - screen, copy to clip board, [387](#)
 - screens, copying to clip board, [315](#)
 - scroll lock on display windows, [227, 238](#)
 - search, [618](#)
 - search for symbol name, [458](#)
 - search operation keyboard commands, [532](#)
 - search order of source file directories, [467](#)
 - search, quick, [264](#)
 - searching data, [48](#)
 - searching for bus/signal patterns, [264](#)
 - searching for text in Source window, [279](#)
 - SECTIONS, General-Purpose ASCII symbol file format, [515](#)
 - Select Offline Hardware dialog, [431](#)
 - Select Symbol dialog, [458](#)
 - Select System to Use dialog, [459](#)
 - selected bit order, assigning channels, [89](#)
 - selected packet box color, [303](#)
 - selected source line, go to data for, [278](#)
 - self test, [343, 397](#)
 - send e-mail on trigger, [166](#)
 - sending markers to the back, [254](#)
 - SendToExcel macro, [391](#)
 - separator rows in Waveform window, [227](#)
 - sequence step branching, transitional timing, [365](#)
 - sequence step storage, [167, 375](#)
 - sequence steps, trigger, [337, 371, 372, 373](#)
 - sessions, saving and loading, [380](#)
 - set flag action, [157, 180](#)
 - Set Quick Trigger alignment, [282](#)
 - Setup menu, [389](#)
 - setup only, loading, [208](#)
 - setup/hold time, logic analyzer, [334](#)
 - setup/hold window, logic analyzer, [367](#)
 - setups, loading saved, [208](#)
 - short bus/signal names, [447](#)
 - show axis in chart, [224](#)
 - show clipped analog property, [234](#)
 - show correlated source line, [279](#)
 - show marker measurement display bar, [400](#)
 - show measurements list, [391](#)
 - show trigger step as if/then, [182, 376](#)
 - show values in waveform tool tip, [230](#)
 - showing markers, [256](#)
 - showing source file line numbers, [282](#)
 - showing/hiding data values in waveforms, [232](#)
 - signal activity envelope, [367](#)
 - signal names, [83](#)
 - signal, grouping into a bus, [227](#)
 - signal/bus column alignment, [241](#)
 - signal/bus column properties, [240](#)
 - signal/bus column width in Listing window, [241](#)
 - signal/bus data color, Listing window, [240](#)
 - signal/bus errors, [334](#)
 - signal/bus folders, adding, [95](#)
 - signal/bus name, alias, [95](#)
 - signal/bus names, sorting, [95](#)
 - signal/bus naming, short or unique, [447](#)
 - signal/bus patterns, finding, [264, 265](#)
 - signal/bus patterns, finding complex, [270](#)
 - signal/bus row properties, [230](#)
 - signal/bus values displayed as symbols, [243](#)
 - signal/bus, adding a new, [84](#)
 - signal/bus, assigning channels in default bit order, [88](#)
 - signal/bus, assigning channels in selected bit order, [89](#)
 - signal/bus, deleting a, [84](#)
 - signal/bus, renaming a, [85](#)
 - signals/buses, define by importing netlist, [446](#)
 - signals/buses, exporting data from selected, [452](#)
 - signals/buses, inserting/deleting in Listing window, [238](#)
 - signals/buses, inserting/deleting in Waveform window, [227](#)
 - signals/buses, load warning, [339](#)
 - Signed Decimal number base, [159, 232, 241](#)
 - simple trigger, [141, 146, 389, 618](#)
 - simple trigger example, [359](#)
 - simple trigger in Listing window, [237](#)
 - simple trigger in Waveform window, [226](#)
 - simple trigger, setting up, [43](#)
 - single clock, [99, 124](#)
 - single measurement, [618](#)
 - single run, [196](#)
 - single-ended signals and differential probes, [474](#)
 - skew, [618](#)
 - skew and system trigger, [389](#)
 - skew and system trigger, setting, [312](#)
 - skew, module, [446](#)
 - slave clock, [101, 103, 114, 115, 127](#)
 - slave frame, [381](#)
 - slot, [618](#)
 - slot naming, [361](#)
 - Slot Summary tab, [472](#)
 - slot, starting, [431](#)
 - slots of cards in module, [474](#)
 - slow or missing clock, [339](#)
 - snap to edge markers, [247, 250, 618](#)
 - Soft Touch connectorless probing, [57](#)
 - Software Address bus, [280, 468](#)
 - software licenses, activation, [323, 463](#)
 - software licenses, borrow floating, [326, 465](#)
 - software licenses, floating license servers, [324, 464](#)
 - software licenses, summary, [322, 461](#)
 - software licensing, [397](#)
 - software, updating, [327](#)

solving problems, [331](#)
 sorting bus/signal names, [95](#)
 source code correlation bus,
 selecting, [280](#)
 Source Code Directories tab, [467](#)
 source code directory list, [279](#)
 source code, viewing, [277](#)
 source correlation, [468](#)
 Source display window, [412](#)
 source file line numbers, [132](#)
 source files, browsing, [278](#)
 source line numbers,
 showing/hiding, [282](#)
 source line symbol values,
 adjusting, [282](#)
 source line, go to data for, [278](#)
 source line, showing correlated, [279](#)
 SOURCE LINES, General-Purpose
 ASCII symbol file format, [518](#)
 source lines, step by, [277](#)
 Source menu, [395](#)
 source of data export, [199](#)
 Source Properties tab, [467](#)
 Source Viewer Properties dialog, [466](#)
 Source window background color, [282](#)
 Source window font size, [282](#)
 Source window properties, [281](#)
 Source window tab width, [282](#)
 Source window text color, [282](#)
 Source window, new, [396](#)
 Source window, Quick Trigger in, [145](#)
 Source window, text search, [279](#)
 specifications, [587, 611](#)
 specifications, 16800-series logic
 analyzer, [600](#)
 Specify Mapping dialog, [468](#)
 split analyzer, [72, 389](#)
 spreadsheet, importing CSV files, [198, 199, 386](#)
 spreadsheet, pasting into from clip
 board, [314](#)
 standard CSV text file, [435](#)
 standard threshold voltage level, [77](#)
 Standard toolbar, [398](#)
 START ADDRESS, General-Purpose
 ASCII symbol file format, [519](#)
 start from reset timer action, [179](#)
 start maximized, [447](#)
 start value of symbol, [458](#)
 startup options, offline, [447](#)
 state acquisition mode, [428](#)
 state analyzer, [356, 618](#)
 state analyzer, triggering, [359](#)
 state clock edges too close, [334](#)
 state measurement, [618](#)
 state mode, [51, 96, 141, 359, 490, 618](#)
 state mode sampling positions,
 understanding, [367](#)
 state mode, default storage, [167](#)
 state mode, selecting, [98](#)
 state sampling clock, [124](#)

state sampling clock mode,
 Demultiplex, [103, 115](#)
 state sampling clock mode, Dual
 Sample, [106](#)
 state sampling clock mode, Master
 Only, [100, 114](#)
 state sampling clock mode,
 Master/Slave, [101, 114](#)
 state sampling clock mode,
 selecting, [99](#)
 state sampling mode, [98](#)
 state sampling mode, memory depth
 and channel count trade-offs, [363](#)
 state sampling options, [98](#)
 state sampling setup, [399](#)
 state speed, [139](#)
 statistics, interval marker, [251, 252](#)
 status bar, [388](#)
 Status dialog, [469](#)
 status, system, [392](#)
 step to next/previous source lines, [277](#)
 steps in trigger sequence, cutting, [182](#)
 steps in trigger sequence,
 deleting, [182](#)
 steps, trigger sequence, [152, 337, 371](#)
 stop, [618](#)
 stop and reset timer action, [179](#)
 stop button, 16800-series logic
 analyzers, [527, 531](#)
 stop measurement, [392, 401](#)
 stopping self tests on fail, [343](#)
 stopping the analyzer, [196](#)
 storage control action, [152, 167, 179, 181, 424](#)
 storage qualification, [167, 375, 618](#)
 storage qualification, triggering
 capability, [370](#)
 storage qualifier when filling
 memory, [166](#)
 store action, [337, 375](#)
 store qualification, [600](#)
 store qualified (transitional) timing
 mode, [97, 545, 550, 553](#)
 store qualified (transitional) timing
 mode, default storage, [167](#)
 store sample action, [181](#)
 store trigger, [194, 389](#)
 storing favorite find patterns, [271](#)
 storing triggers, [424](#)
 strategies for setting up triggers, [376](#)
 subdirectories, searching for source
 files, [467](#)
 summary of triggering capabilities, [370](#)
 summary, module, [474](#)
 summary, system, [472](#)
 supported signal types, [595](#)
 switch endian, [91](#)
 symbol decode, [299](#)
 symbol file, ASCII format, [514](#)
 symbol file, creating ASCII, [135](#)
 symbol information, XML format, [380](#)
 symbol lookup, [132](#)

Symbol number base, [159, 232, 241](#)
 symbol reader options, changing, [136](#)
 symbol reader, object file formats
 supported, [513](#)
 symbol reader, running outside the
 application, [135](#)
 symbol search, [458](#)
 symbol selection, [458](#)
 symbolic bus/signal values, [138](#)
 symbols, [132, 387, 389, 458, 619](#)
 Symbols dialog, [471](#)
 symbols with same name, [132](#)
 symbols, displaying for bus/signal
 values, [243](#)
 symbols, line number, [277, 280](#)
 symbols, loading from file, [134](#)
 symbols, user-defined, [132](#)
 synchronous sampling, [98, 428, 619](#)
 system clock failure, [337](#)
 system details, [439](#)
 system details, viewing, [64](#)
 system in use comments, [63, 65, 459](#)
 system options, [447](#)
 system summary, [393](#)
 System Summary dialog, [472](#)
 system to use list, adding frame, [61](#)
 system to use list, deleting frame, [62](#)
 system trigger, [446](#)
 system trigger and skew, [389](#)
 system trigger and skew, setting, [312](#)

T

tab width in Source window, [282, 467](#)
 tabbed windows, [220, 235, 313, 388](#)
 target connector probing, [57](#)
 target control port, [58, 389](#)
 Target Control Port dialog, [475](#)
 target system, [619](#)
 temperature, [595, 600](#)
 terminating slave frame, [381](#)
 testing the logic analyzer, [343](#)
 text color, Source window, [282](#)
 text data, copying to clip board, [314](#)
 text file, print to, [314](#)
 text search in Source window, [279](#)
 third party tools, [390](#)
 tHold, [600](#)
 threshold (bus/signal), [83](#)
 threshold accuracy, [595](#)
 threshold level, trigger in/out, [437](#)
 Threshold Settings dialog, [77](#)
 threshold value, [334](#)
 threshold voltage, [77, 426, 619](#)
 TI COFF object file format, [513](#)
 Ticoff symbol files, [136](#)
 tile windows horizontally, [396, 398](#)
 tile windows vertically, [396, 398](#)
 time axis in Waveform window, [226](#)
 time duration in find qualifiers, [265](#)
 time interval accuracy, [600](#)

- time interval measurement, [251, 391, 400, 403](#)
- time measured, transitional timing, [365](#)
- time tag resolution, [600](#)
- time tag storage, memory requirements, [363](#)
- time tag storage, reserve pod pair for, [450](#)
- time tag storage, transitional timing, [365](#)
- time trigger occurred, [474](#)
- time, absolute, [241](#)
- time, go to in Listing window, [236](#)
- time, go to in Waveform window, [222](#)
- time, marker-relative, [242](#)
- time, position marker by, [246, 247](#)
- time, relative marker, [241](#)
- time, relative previous, [241](#)
- time/division, [221, 619](#)
- timeout, floating license server communication, [342](#)
- timer, [619](#)
- timer accuracy, [600](#)
- timer action, [179](#)
- timer event, [168, 169](#)
- timer event check without start action, [339](#)
- timer in trigger sequence, [337](#)
- timer reset latency, [153, 600](#)
- timer resolution, [600](#)
- timer status, [469](#)
- timer value range, [600](#)
- timers, [152, 374](#)
- timers, number available, [363](#)
- timers, triggering capability, [370](#)
- timers, using, [153](#)
- timing acquisition mode, [428](#)
- timing analysis sample rate, [600](#)
- timing analyzer, [356, 619](#)
- timing analyzer, triggering, [357](#)
- timing interval accuracy, [600](#)
- timing measurement, [619](#)
- timing mode, [50, 96, 141, 356, 619](#)
- timing mode sampling period, [98](#)
- timing mode trigger functions, [478](#)
- timing mode, duration or occurrence counts for events, [178](#)
- timing mode, selecting, [96](#)
- timing sampling mode, [97](#)
- timing sampling options, [97](#)
- timing zoom, [129, 553](#)
- timing zoom background color, Listing window, [239](#)
- timing zoom background color, Waveform window, [229](#)
- timing zoom sample period, [131](#)
- timing zoom trigger position, [131](#)
- timing zoom, turning on or off, [131](#)
- timing/state sampling setup, [399](#)
- timing/state setup, [389](#)
- TimingZoom Setup dialog, [476](#)
- tips for experienced users, [39](#)
- toggle target control port signals, [475](#)
- too few states between pattern1 and pattern2 trigger function, [494](#)
- too many states between pattern1 and pattern2 trigger function, [494](#)
- tool details, [469](#)
- tool properties, editing, [310](#)
- tool tip, [619](#)
- tool tip display, Waveform window, [230](#)
- tool tip, marker, [246, 254, 257](#)
- tool tips, [49, 94](#)
- tool tips, Waveform window, [227](#)
- tool, cancel processing, [469](#)
- tool, resume canceled processing, [469](#)
- toolbars, [388, 398](#)
- toolbars and mouse shortcuts, [49](#)
- toolbars, customizing, [402](#)
- Tools menu, [390](#)
- Tools menu, adding external applications, [436](#)
- tools, adding/deleting, [307](#)
- tools, help on, [397](#)
- tools, renaming, [311](#)
- tools/viewers, loading, [208](#)
- topics, help, [397](#)
- trace, [619](#)
- trace memory, [196](#)
- transferring setups to/from multi-module systems, [210](#)
- transition width in waveform tool tip, [230](#)
- transitional default storage, [167](#)
- transitional timing, [97, 365, 620](#)
- transitional timing mode, counters in, [154](#)
- transitional timing sampling mode, memory depth and channel count trade-offs, [363](#)
- Transitioning, find qualifier, [265](#)
- trigger, [146, 152, 194, 620](#)
- trigger actions, [166, 371, 600](#)
- trigger analyzer from another instrument, [192](#)
- trigger and fill memory action, [166](#)
- trigger and goto action, [166](#)
- trigger attributes, external oscilloscope, [399](#)
- trigger errors, [337](#)
- trigger function, [424, 620](#)
- trigger functions, [152, 478](#)
- trigger functions, converting to if/then, [183](#)
- trigger functions, replacing or inserting, [158](#)
- trigger functions, state mode, [490](#)
- trigger functions, timing mode, [478](#)
- trigger functions, understanding triggering, [376](#)
- trigger history, [620](#)
- trigger history depth, [194, 195, 447](#)
- trigger in, [186, 437](#)
- trigger in from external instrument, [192](#)
- Trigger In/Out and multiframe systems, [381](#)
- trigger inhibited during prestore, [340](#)
- trigger other instruments, [191](#)
- trigger out, [186, 437](#)
- trigger out to external instruments, [191](#)
- trigger point, [222, 369](#)
- trigger position, [128, 369, 428, 600, 620](#)
- trigger position, timing zoom, [131, 476](#)
- trigger position, transitional timing, [365](#)
- trigger reference point skew, [446](#)
- trigger resource conditions, [600](#)
- trigger resources, [595, 600](#)
- trigger sequence, [424, 620](#)
- trigger sequence step branching, [600](#)
- trigger sequence step, show as if/then, [182](#)
- trigger sequence steps, [152, 337, 371](#)
- trigger sequence steps, cutting, [182](#)
- trigger sequence steps, deleting, [182](#)
- trigger sequence steps, replacing or inserting trigger functions, [158](#)
- trigger sequence, clearing, [185](#)
- trigger sequences, save and document, [378](#)
- trigger setup file, [194](#)
- trigger setups, saving/recalling, [46](#)
- trigger specification file, [453](#)
- trigger specification too complex, [337](#)
- trigger step status, [469](#)
- trigger, go to, [249](#)
- trigger, go to in Listing window, [236](#)
- trigger, go to on run, [196, 447](#)
- trigger, recalling, [194, 453](#)
- trigger, send e-mail, and fill memory action, [166](#)
- trigger, system, [446](#)
- trigger, system, setting, [312](#)
- trigger, time trigger occurred, [474](#)
- triggering, [146, 152, 194](#)
- triggering after external data analysis, [206](#)
- triggering capabilities, summary of, [370](#)
- triggering the state analyzer, [359](#)
- triggering the timing analyzer, [357](#)
- triggering, understanding logic analyzer, [369](#)
- triggers, dynamic, [206](#)
- triggers, quick, [143](#)
- triggers, recalling, [194](#)
- triggers, storing, [194](#)
- triggers, strategies for setting up, [376](#)
- troubleshooting, [331](#)
- tSample accuracy, manual adjustment, [600](#)
- tSample adjustment resolution, [600](#)

tSample range, **600**
 tSetup, **600**
 turbo state mode, **98, 553**
 turbo state mode, flag actions, **157**
 turn off/on default storage action, **181**
 tutorial, **40**
 tutorial configuration file, **44**
 tWidth, **600**
 two's complement, **232, 241**
 two's complement, **159**
 type, symbol, **458**
 types, probe, **474**
 TZ (timing zoom), **129**

U

U4154A, **522, 523**
 U4154A Characteristics, **592, 596**
 U4154A clock channels, **562**
 U4154A data channels, **561**
 U4154A eyescans, **567**
 U4154A logic analyzer notes, **544, 549**
 U4154A pod, **561**
 U4154A Specifications, **592, 596**
 U4154A state clock qualifiers, **548, 552**
 U4154A state sampling clock, **548, 552**
 U4164A Logic Analyzer Product Overview, **520**
 unassigned channels, data on, transitional timing, **365**
 understanding logic analyzer triggering, **369**
 undo, **387, 398**
 undo trigger sequence step shown as if/then, **182**
 unique bus/signal names, **447**
 unsplit analyzer, **389**
 update software, **327**
 upgrade, module, **397, 425**
 upgrades, E5876A, for 16801A/16821A logic analyzer, **553**
 upgrades, E5877A, for 16802A/16822A logic analyzer, **553**
 upgrades, E5878A, for 16803A/16823A logic analyzer, **553**
 upgrades, E5879A, for 16804A logic analyzer, **553**
 upgrades, E5880A, for 16806A logic analyzer, **553**
 upgrades, licensed hardware, **139**
 user interface, **527, 531**
 user threshold value, **334**
 USER, General-Purpose ASCII symbol file format, **516**
 user-defined symbols, **132, 471**
 user-defined threshold voltage level, **77**

V

value at marker measurement, **253, 391, 400, 403**
 value at measurement, **620**
 value, position marker by, **246, 247**
 value, symbol, **458**
 variable names, **132**
 variable size, **513**
 VARIABLES, General-Purpose ASCII symbol file format, **518**
 version information, **397**
 version information, configuration file, **208**
 vibration, **595, 600**
 View As dialog, **224**
 View menu, **388**
 view operation keyboard commands, **532**
 view packet event as bits, **165**
 viewers/tools, loading, **208**
 viewing bus data as chart, **224**
 viewing captured data, **207**
 viewing tabbed windows, **313**
 virtual analyzer, **431**
 Visual Basic Editor, **401**
 Visual Basic toolbar, **401**
 voltage marker properties, changing, **262**
 voltage markers for analog signals, **257**
 voltage markers, placing, **259**
 Voltage number base for analog signal in Listing, **241**
 voltage threshold, **595**

W

wait for arm from another module, **186**
 wait for arm from another module trigger function, state mode, **499**
 wait for arm from another module trigger function, timing mode, **487**
 wait for external arm, **192**
 wait for external arm trigger function, state mode, **498**
 wait for external arm trigger function, timing mode, **486**
 wait N external clock states trigger function, **499**
 wait T seconds trigger function, **485**
 waiting in trigger step, **340**
 warning dialogs, showing/hiding, **447**
 warning messages, **339**
 warning messages in the status log, **469**
 waveform color, **231**
 waveform display scale, changing, **221**
 waveform display, changing the delay, **222**
 waveform font size, **229**
 waveform height, **232**

Waveform menu, **394**
 waveform numeric data values, showing/hiding, **232**
 waveform operation keyboard commands, **532**
 waveform tool tip display, **230**
 Waveform window, **404**
 Waveform window background color, **228**
 Waveform window, analyzing data, **220**
 Waveform window, bus/signal row properties, **230**
 Waveform window, filtered data color, **229**
 Waveform window, finding a bus/signal row, **224**
 Waveform window, go to locations in, **222**
 Waveform window, group signals into bus, **227**
 Waveform window, inserting/deleting buses/signals, **227**
 Waveform window, lock scrolling, **229**
 Waveform window, new, **396**
 Waveform window, overlaid waveform color, **228**
 Waveform window, properties, **227**
 Waveform window, Quick Trigger in, **143**
 Waveform window, showing/hiding parts of, **226**
 Waveform window, timing zoom background color, **229**
 Waveform window, viewing bus data as chart, **224**
 Waveform windows, Fast Zoom In option, **230**
 waveforms, accumulate, **229**
 waveforms, overlaying, **223**
 waveforms, re-arranging, **223**
 web sites, **351**
 what's new, **25**
 why are pods missing?, **362**
 width (bus/signal), **83**
 width of buses/signals, **426**
 width violation on pattern or pulse trigger function, **485**
 width, packet decode column, **305**
 width, tabs in Source window, **282**
 window details, **469**
 window menu, **396**
 window operation keyboard commands, **532**
 window properties, **387**
 window properties, editing, **310**
 window tabs, turning on/off, **313**
 window, cancel processing, **469**
 window, Overview, **307, 418**
 window, resume canceled processing, **469**
 windows, **404**

- windows, adding/deleting, [313](#)
- windows,
 - adding/deleting/duplicating, [307](#)
- windows, display, using, [313](#)
- windows, help on, [397](#)
- windows, renaming, [311](#)
- writing to target control port
 - signals, [475](#)

X

- XML format configuration file, [198, 208, 210](#)
- XML format configuration files,
 - automatic module mapping, [445](#)
- XML format files, offline analysis, [218](#)
- XML vs. ALA format, when to use, [380](#)

Y

- Y-Axis interval measurements, [260](#)
- Y-Axis markers, creating new, [258](#)
- Y-Axis markers, deleting, [260](#)
- Y-Axis markers, renaming, [261](#)
- Y-Axis markers, sending to back, [261](#)

Z

- zoom in on data, [45, 221](#)
- zoom out/in, [388, 398](#)
- zooming, [621](#)
- zooming fast in Waveform
 - windows, [230](#)

