Keysight Logic and Protocol Analyzer

User Guide

KEYSIGHT TECHNOLOGIES

Notices

© Keysight Technologies 2008-2014

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 06.03.0000

Edition

November 2014

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.

Restricted Rights Legend

If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as "Commercial computer software" as defined in DFAR 252.227-7014 (June 1995), or as a "commercial item" as defined in FAR 2.101(a) or as "Restricted computer software" as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause. Use, duplication or disclosure of Software is subject to Keysight Technologies' standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), 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

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. Product manuals are provided with your instrument on CD-ROM and/or in printed form. 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.

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.

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”.

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.

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

Do Not Operate in an Explosive Atmosphere Do not operate the instrument in the presence of flammable gases or fumes.

Do Not Remove the Instrument Cover 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.

Cleaning Clean the outside of the instrument with a soft, lint-free, slightly dampened cloth. Do not use detergent or chemical solvents.

Do Not Modify the Instrument 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.

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

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 Symbols

Table 1 Safety Symbol

Symbol	Description
	Direct current
	Alternating current
	Both direct and alternating current
	Three phase alternating current
	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

Symbol	Description
	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
	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 I	IEC Measurement Category I
CAT II	Measurement Category II
CAT III	Measurement Category III
CAT IV	Measurement Category IV

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
	Product With Toxic Substance 40 yr EPUP

Table 3 Environmental Information

Safety Symbol	Description
	Notice for the European Community: This product complies with the WEEE Directive (2002/96/EC) marking requirements. The affixed label indicates that you must not discard this electrical/electronic product in domestic household waste. *Product Category: With reference to the requirement types in the WEEE Directive Annex I, this product is classed as a "Monitoring and Control Instrumentation" product.* ***Do not dispose in domestic household waste.*** *To return unwanted products, contact your local Keysight office, or see www.keysight.com/environment/product/ for more information.*

Using the Keysight Logic and Protocol Analyzer

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 75) of logic analysis systems on the network, or for *offline analysis* (see page 194) of captured data.

This user guide provides the following information.

- What's New (see page 21)
- Getting Started (see page 55)
- Probing the Device Under Test (see page 73)
- Connecting to a Logic Analysis System (see page 75)
- Setting Up the Logic Analyzer (see page 87)
  - Configuring Logic Analyzer Modules (see page 88)
  - "Setting Up Probes" (in the online help)
  - Setting the Logic Analyzer Threshold Voltage (see page 91)
  - Defining Buses and Signals (see page 93)
  - Choosing the Sampling Mode (see page 106)
  - Setting Up Symbols (see page 118)
  - Installing Licensed Hardware Upgrades (see page 124)
- Capturing Data from the Device Under Test (see page 127)
  - Setting Up Quick (Draw Box) Triggers (see page 129)
  - Specifying Simple Triggers (see page 132)
  - Specifying Advanced Triggers (see page 138)
  - Triggering From, and Sending Triggers To, Other Modules/Instruments (see page 164)
  - Storing and Recalling Triggers (see page 172)
  - Running/Stopping Measurements (see page 174)
  - Saving Captured Data (and Logic Analyzer Setups) (see page 176)
  - Extending Capture Capability with COM/DCOM (see page 186)
- Analyzing the Captured Data (see page 187)
  - Offline Analysis (see page 194) (after Loading Saved Data and Setups (see page 188))
  - Analyzing Waveform Data (see page 197)
  - Analyzing Listing Data (see page 212)
  - Displaying Names (Symbols) for Bus/Signal Values (see page 220)
  - Marking, and Measuring Between, Data Points (see page 222)
  - Searching the Captured Data (see page 240)
  - Comparing Captured Data to Reference Data (see page 250)
  - Viewing Source Code Associated with Captured Data (see page 253)
  - Analyzing Packet Data (see page 259)
  - Analyzing the Same Data in Different Ways (Using the Window) (see page 283)
  - Setting the System Trigger and Skew Between Modules (see page 288)
  - Using Display Windows (see page 289)

- Printing Captured Data (see page 290)

- "Using Tools" (in the online help)
- "External Oscilloscope Time Correlation and Data Display" (in the online help)
- "Using the Pattern Generator" (in the online help)
- "Using the PCIe Gen3 Analyzer"
- Managing Software Licenses (see page 293)
- Updating Software (see page 299)
- Solving Problems (see page 303)
- Concepts (see page 325)
- Reference (see page 357)
- Glossary (see page 537)
- "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"*

Contents

Safety Summary 3

Compliance and Environmental Information 6

Using the Keysight Logic and Protocol Analyzer 7

1 What's New

Version 05.80 What's New 23

Version 05.70 What's New 25

Version 05.60 What's New 26

Version 05.50 What's New 28

Version 05.40 What's New 30

Version 05.30 What's New 31

Version 05.20 What's New 33

Version 05.00 What's New 35

Version 04.00 What's New 37

Version 03.83 What's New 38

Version 03.82 What's New 39

Version 03.80 What's New 40

Version 03.70 What's New 41

Version 03.67 What's New 42

Version 03.65 What's New 43

Version 03.60 What's New 44

Version 03.55 What's New 45

Version 03.50 What's New 46

Version 03.30 What's New 47

Version 03.20 What's New 48

Version 03.00 What's New 49

Version 02.50 What's New 50

Version 02.00 What's New 51

Version 01.40 What's New 52

Version 01.20 What's New 53

Version 01.10 What's New 54

2 Getting Started

Tutorial - Getting to know your logic analyzer 56

Turning on the logic analyzer 56
Connecting to the device under test 57
Setting up bus/signal names 58
Setting the acquisition mode 59
Setting up a simple trigger 59
Open the tutorial configuration file 60
Using markers 60
Zooming in on the data 61
Loading and saving configuration files 62
Saving and recalling trigger setups 62
Quick marker measurements 63
Searching data 64
Toolbars, tool tips, and mouse shortcuts 65

Measurement Examples 66

Making a timing analyzer measurement 66
Making a state analyzer measurement 67
To trigger on one of multiple edges or glitches 68
To trigger on ranges 68

Demo Center 71

3 Probing the Device Under Test

To control signals in the device under test 74

4 Connecting to a Logic Analysis System

Connecting to the U4154A/B Logic Analyzer Module 75

To add a logic analysis system to the list 77

To delete a logic analysis system from the list 78

To refresh the logic analysis system list 79

To view logic analysis system details 80

To enter your "System In Use" comments 81

To select a logic analysis system for auto-connect 82

To chat with another logic analysis system user 83

Setting Up Passwords for Remote Access 84

5 Setting Up the Logic Analyzer

Configuring Logic Analyzer Modules 88

To disable and enable modules 88

Setting the Logic Analyzer Threshold Voltage 91

Defining Buses and Signals 93

To add a new bus or signal 94
To delete a bus or signal 95
To rename a bus or signal 95
To rename the bits of a bus 96
To assign channels in the default bit order 98
To assign channels, selecting the bit order 99
To use clock channels as extra data channels 99
To define buses and signals by importing netlist files 100
To reorder bits by editing the Channels Assigned string 101
To set the default number base 103
To set polarity 104
To add user comments 104
To add a folder 105
To alias a bus/signal name 105
To sort bus/signal names 105

Choosing the Sampling Mode 106

Selecting the Timing Mode (Asynchronous Sampling) 106
Selecting the State Mode (Synchronous Sampling) 108
In Either Timing Mode or State Mode 115
Using Timing Zoom 116

Setting Up Symbols 118

To create user-defined symbols 118
To load symbols from a file 119
To run the symbol reader outside the application 120
To create an ASCII symbol file 121
To change symbol reader options 121
To enter symbolic bus/signal values 123

Installing Licensed Hardware Upgrades 124

6 Capturing Data from the Device Under Test

Setting Up Quick (Draw Box) Triggers 129

To set a Quick Trigger in the Waveform window 129
To set a Quick Trigger in the Listing window 130
To set a Quick Trigger in the Source window 131

Specifying Simple Triggers 132
To specify bus patterns in a simple trigger 133
To specify signal levels in a simple trigger 134
To set a bus/signal edge in a simple trigger 134
Specifying Advanced Triggers 138
Reading Event and Action Statements 139
Using Timers 139
Using Counters 140
Using Flags 141
To replace or insert trigger functions into trigger sequence steps 142
To specify bus/signal patterns 142
To set a bus/signal edge in an advanced trigger 144
To specify packet events (in "Find a packet" trigger function) 148
To specify a trigger sequence step's goto or trigger action 150
To specify default storage 151
To insert or delete events 152
To negate events 154
To change the evaluation order of AND/OR'ed events 155
To choose between a duration or occurrence count for events (timing mode) 156
To insert or delete actions (in a trigger sequence step) 156
To cut, copy, and paste trigger sequence steps 159
To delete trigger sequence steps 160
To show a trigger sequence step as Advanced If/Then trigger functions 160
To convert a trigger sequence step to Advanced If/Then trigger functions 160
To display or hide "If" clause comments 161
To clear the trigger sequence 163
Triggering From, and Sending Triggers To, Other Modules/Instruments 164
To arm one module with another module's trigger 164
To trigger other instruments - Trigger Out 169
To trigger analyzer from another instrument - Trigger In 170
Storing and Recalling Triggers 172
To store a trigger 172
To recall a trigger 172
To set the trigger history depth 173
Running/Stopping Measurements 174
To change the "Go to Trigger on Run" option 174

Saving Captured Data (and Logic Analyzer Setups) 176
To save a configuration file 176
To export data to standard CSV format files 177
To export data to module CSV format files 179
To export data to module binary (ALB) format files 181
To export data to 16700 ASCII format files 183

Extending Capture Capability with COM/DCOM 186

7 Analyzing the Captured Data

Loading Saved Data and Setups 188
To open a configuration file 188
To recall a recently used configuration file 190
To transfer module setups to/from multi-module systems 190
To set up multiple-modules with XML-format configurations 191

Offline Analysis 194
Offline Analysis on Logic Analyzers 194
Offline Analysis on Personal Computers 195
Offline File Formats 195

Analyzing Waveform Data 197
To change the display scale (time/division) 198
To go to different locations in the captured data 199
To re-arrange waveforms 200
To overlay waveforms 200
To find a bus/signal row 201
To view bus data as a chart 201
To show/hide parts of the waveform display 203
To insert or delete buses/signals 204
To group signals into a bus 204
To expand/collapse buses 204
To insert separator rows 204
Changing Waveform Window Properties 204
Changing Bus/Signal Row Properties 207
Changing Analog Signal Row Properties 209

Analyzing Listing Data 212
To go to different locations in the captured data 213
To re-arrange bus/signal columns 214
To find a bus/signal column 214
To show/hide parts of the listing display 214
To insert or delete buses/signals 215
Changing Listing Window Properties 215
Changing Bus/Signal Column Properties 217

Displaying Names (Symbols) for Bus/Signal Values 220

Marking, and Measuring Between, Data Points 222

To read the markers display and overview bars 222
To create new markers 223
To place markers in data 224
To go to a marker 226
To center the display about a marker pair 226
To change a marker's snap to edge setting 227
To delete a marker 227
To create a new time interval measurement 228
To create a new sample interval measurement 229
To create a new value at measurement 230
To rename a marker 231
To send a marker to the back 231
Changing Marker Properties 232
Using Voltage Markers for Analog Signals (in the Waveform Display) 234

Searching the Captured Data 240

To quickly find bus/signal patterns 240
To find bus/signal patterns in the captured data 241
To find packet patterns in the captured data 243
To find complex patterns in the captured data 246
To store, recall, or delete favorite find patterns 247
To specify "found" marker placement 248

Comparing Captured Data to Reference Data 250

To copy data to the reference buffer 250
To find differences in the compared data 250
To compare only a range of samples 251
To offset the reference data 251
To run until a number of compare differences 251
To set Compare window properties 252

Viewing Source Code Associated with Captured Data 253

To step through captured data by source lines 253
To go to captured data associated with a source line 254
To browse source files 254
To search for text in source files 255
To go to the source line associated with the listing center rectangle 255
To edit the source code directory list 255
To select the correlation bus 256
Changing Source Window Properties 257

Analyzing Packet Data 259
Viewing the Packet Summary 261
Viewing a Selected Packet 269
Changing Packet Summary Event Colors 277
Changing Protocol Viewer Window Properties 278
Changing Packet Summary Column Properties 280

Analyzing the Same Data in Different Ways (Using the Overview Window) 283
To open or display the Overview window 283
To add, duplicate, or delete windows and tools 283
To edit window or tool properties 286
To rename windows, tools, and modules 287
To redraw the Overview window 287
To delete the Overview window 287

Setting the System Trigger and Skew Between Modules 288

Using Display Windows 289
To add or delete display windows 289
To turn window tabs on/off 289

Printing Captured Data 290
To print captured data 290
To copy text to the clip board 290
To copy a screen to the clip board 291
To install a printer 291
To connect a LAN 291

8 Managing Software Licenses

To view active software license information 294

To activate software licenses 295

To access floating license servers 296

To borrow floating licenses and return them early 298

9 Updating Software

10 Solving Problems

If starting in offline mode is unexpected 304

If an ALA format configuration file won't open 305

Interpreting Error Messages 306
Error Messages 306
Warning Messages 311
Informational Messages 312

License Problems 314
License Not Available 314
Floating License Server Communication Timeout 314
Running Self Tests 315
Accessing Japanese Online Help 317
Network Troubleshooting Guide 320
Remote Desktop Set Up 322
For More Information 323

11 Concepts

When should you use an oscilloscope? 326
When should you use a logic analyzer? 327
What is a logic analyzer? 328
Sampling clock in the timing analyzer 328
Sampling in the timing acquisition mode 328
Triggering the timing analyzer 329
Sampling clock in the state analyzer 330
Sampling in the state acquisition mode 331
Triggering the state analyzer 331
Pod and Channel Naming Conventions 333
Why Are Pods Missing? 334
Memory Depth and Channel Count Trade-offs 335
Transitional Timing 337
Understanding State Mode Sampling Positions 339
Understanding Logic Analyzer Triggering 341
The Conveyor Belt Analogy 341
Summary of Triggering Capabilities 342
Sequence Steps 343
Boolean Expressions 344
Branches 345
Edges 345
Ranges 345
Flags 345
Occurrence Counters and Global Counters 345
Timers 346
Storage Qualification 347
Strategies for Setting Up Triggers 348
Conclusions 351

ALA vs. XML, When to Use Each Format 352

Multiframe Logic Analysis Systems 353

12 Reference

Menus 358

File Menu 358
Edit Menu 359
View Menu 360
Setup Menu 361
Tools Menu 362
Markers Menu 364
Run/Stop Menu 365
Overview Menu 366
Listing Menu 366
Waveform Menu 366
Compare Menu 367
Source Menu 368
Protocol Viewer Menu 368
Window Menu 368
Help Menu 369

Toolbars 371

Standard Toolbar 371
Pattern Generator Toolbar 372
Analyzer Setup Toolbar 372
External Oscilloscope Setup Toolbar 373
Probes Toolbar 373
Markers Toolbar 373
Run/Stop Toolbar 374
Visual Basic Toolbar 374
To customize toolbars 375

Marker Measurement Display Bar 376

Windows 377

Waveform Display Window 377
Listing Display Window 382
Compare Display Window 384
Source Display Window 385
Protocol Viewer Display Window 386
Overview Window 389

Dialogs 391
Advanced Clocking Setup Dialog 391
Advanced Trigger Dialog 392
Keysight Logic Analyzer Upgrade Dialog 393
Analyzer Setup Dialog 394
Chat Dialog 397
Chat Select Destination Dialog 398
Choose a Protocol Family and Bus Dialog 398
Create a New Configuration Dialog 399
E-mail Dialog 399
Event Editor Dialog 401
Export Dialog 401
Export File Selection Dialog 402
External Application Setup Dialog 403
External Trigger Dialog 404
Find Dialog 406
Frame/Module Information Dialog 407
Module Mapping Dialog 409
Module Skew and System Trigger Dialog 410
Netlist Import Dialog 411
Offline Startup Options Dialog 411
Options Dialog 412
Pod Assignment Dialog 414
Printing Data Dialog 415
Properties Dialog 415
Range Properties Dialog 416
Recall Trigger Dialog 417
Run Properties Dialog 418
Select Symbol Dialog 419
Select System to Use Dialog 421
Software Licensing Dialog 423
Source Viewer Properties Dialog 427
Specify Mapping Dialog 429
Split Analyzer Setup Dialog 429
Status Dialog 430
Symbols Dialog 433
System Summary Dialog 434
Target Control Port Dialog 437
TimingZoom Setup Dialog 438

Trigger Functions 440
Timing Mode Trigger Functions 440
State Mode Trigger Functions 452

Data Formats 466
ALA Format 466
Standard CSV Format 466
Module CSV Format 466
Module Binary (ALB) Format 473

Object File Formats Supported by the Symbol Reader 475

General-Purpose ASCII (GPA) Symbol File Format 476
General-Purpose ASCII (GPA) Record Format Summary 476
SECTIONS 477
FUNCTIONS 478
USER 478
VARIABLES 480
SOURCE LINES 480
START ADDRESS 481
Comments 481

Product Overviews 482
U4154A Logic Analyzer Product Overview 482
U4154B Logic Analyzer Product Overview 484
16850-Series Logic Analyzer Product Overview 485
Keysight Logic and Protocol Analyzer Application Product Overview 488

Logic Analyzer Notes 492
U4154A Logic Analyzer Notes 492
U4154B Logic Analyzer Notes 494
16850-Series Logic Analyzer Notes 526

Specifications and Characteristics 528
U4154A Logic Analyzer Specifications and Characteristics 528
U4154B Logic Analyzer Specifications and Characteristics 531
16850-Series Logic Analyzer Specifications and Characteristics 531
What is a Specification? 536
What is a Characteristic? 536

13 Glossary

Index

1 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 MPCIe 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 MPCIe 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 MPCIe 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.

See Also

- "Version 05.80 What's New" on page 23
- "Version 05.70 What's New" on page 25
- "Version 05.60 What's New" on page 26
- "Version 05.50 What's New" on page 28
- "Version 05.40 What's New" on page 30
- "Version 05.30 What's New" on page 31
- "Version 05.20 What's New" on page 33
- "Version 05.00 What's New" on page 35
- "Version 04.00 What's New" on page 37
- "Version 03.83 What's New" on page 38
- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 05.80 What's New

In this release, version 05.80, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- U4431A MIPI M-PHY Analyzer module - The Keysight Logic and Protocol Analyzer application now adds support for a new AXIe-based module introduced with this release to capture and decode M-PHY, UniPro, UFS, SSIC, and CSI-3 data. This module is installed in an Keysight AXIe chassis and configured and controlled using the Keysight Logic and Protocol Analyzer application. Refer to the *U4431 M-PHY Analyzer Online Help* to know how to configure and use with module.
- U4301 PCIe Gen3 analyzer module -
  - You can now decode AHCI transactions as well in addition to NVMe and PCIe transactions from the data captured by the U4301 module. Refer to the topic *Computing and Viewing Decoded Transactions* in the *U4301 PCIe Gen3 Analyzer Online Help* for more information.
  - A new feature, *Flow Control* is now available in the PCIe 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 PCIe traffic. Refer to the topic *Viewing Offline Performance Summary* in the *U4301 PCIe Gen3 Analyzer Online Help* for more information.
- DDR/LPDDR tools enhancements -
  - A new tool named *DDR/LPDDR Custom Configuration Creator* has been added to the *Keysight DDR Setup Assistant and Eyefinder* software package. This tool 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. Refer to the tool's online help to know more. The online help is accessible from the tool's GUI and also installed at *<logic and protocol analyzer install location> \help*.
  - The *B4623 LPDDR Decoder* now supports decoding and displaying the entry to and exit from Self Refresh and Power Down events. Refer to the topic *Decoding Self Refresh and Power Down Events* in the *LPDDR Bus Decoder Online Help*.
  - Five new LPDDR2/3 tests have been added to the DDR Post Process Compliance tool to handle compliance parameters related to power down and self refresh modes. Three of these tests have also been added to the Real Time Compliance tool.
  - Support for DDR4 running at > 2.5GHz added.
- Trigger on edge - For 16850-series and U4154A logic analyzer modules, you can now set a trigger on Rising, Falling, or Either edge in the Timing - Asynchronous sampling mode as well as State –Synchronous sampling mode. Refer to the topics *Specifying Simple Triggers* and *Specifying Advanced Triggers* in the *Logic and Protocol Analyzer* online help.

See Also

- "Version 05.70 What's New" on page 25
- "Version 05.60 What's New" on page 26
- "Version 05.50 What's New" on page 28
- "Version 05.40 What's New" on page 30
- "Version 05.30 What's New" on page 31
- "Version 05.20 What's New" on page 33
- "Version 05.00 What's New" on page 35
- "Version 04.00 What's New" on page 37
- "Version 03.83 What's New" on page 38
- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43

- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 05.70 What's New

In this release, version 05.70, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **16850 series portable logic analyzers** - With this release, the Keysight Logic and Protocol Analyzer application can also be installed and used with the newly introduced 16850 series portable logic analyzers. Refer to the topic *16850-Series Logic Analyzer Notes* to know more about this new series of logic analyzers.
- **Transaction decoding** - You can now compute and view decoded transactions from the captured PCIe data using the newly added Transaction Decode tab in the Protocol Viewer. Currently, the support for decoding NVMe transactions is added. The transaction decoding feature is a licensed option. Refer to the help book *Viewing Decoded Transactions* in the *U4301 PCIe Gen3 Analyzer online help* to know more.
- **Offline performance summary computation** - You can now compute and view offline performance summary from the captured PCIe data using the newly added PCIe Performance Overview tab in the Protocol Viewer. This tab presents statistics for various performance parameters in tabular as well as charts form. The offline performance summary feature is a licensed option. Refer to the help book *Viewing Offline Performance Summary* in the *U4301 PCIe Gen3 Analyzer online help* to know more.

See Also

- "Version 05.60 What's New" on page 26
- "Version 05.50 What's New" on page 28
- "Version 05.40 What's New" on page 30
- "Version 05.30 What's New" on page 31
- "Version 05.20 What's New" on page 33
- "Version 05.00 What's New" on page 35
- "Version 04.00 What's New" on page 37
- "Version 03.83 What's New" on page 38
- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 05.60 What's New

In this release, version 05.60, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **Windows 8 and Windows Server 2008 support added** - With this release, you can install the Keysight Logic and Protocol Analyzer application on a host with Windows 8 or Windows Server 2008 operating system.
- **Enhancements have been made to the Eyescan feature -** The eyescan feature now supports scaling the voltage axis for an entire bus to a selected value in an eyescan diagram. To know more, refer to the topic "Setting up and Running Eyescans in U4154A/B Logic Analyzer" on page 508.
- **B462X software packages** - Both A and B versions of the licenses for B4621/2/3 software packages are now supported in the Logic and Protocol Analyzer application. B version is an upgrade to A version and therefore provides additional capabilities and features. To know more about the feature difference, refer to the *Online help* of these software packages.
- **Repetitive run feature in the DDR Post Process Compliance tool** - You can now run the selected tests in a repetitive mode. To accomplish this, the tool runs the logic analyzer repetitively for data acquisition and then run the tests repetitively on the newly acquired data. To know more, refer to the *DDR Post Process Compliance Tool Online Help*.
- **BMC Firmware Upgrade utility** - A new utility named BMC Firmware Upgrade has been provided. This utility upgrades the BMC firmware of a module to the latest version provided with the Logic and Protocol Analyzer software. To know more, refer to the *Keysight AXIe Based Logic Analysis and Protocol Test Module Installation Guide*.
- **Support for five independent modules in a chassis** - You can now install and use five independent modules in an Keysight five-slot AXIe chassis. In earlier releases, a maximum of three module sets were supported. (A module set can be either a single module or a group of modules cabled together to operate as a module.)
- **"Or-ing" ARM In support added in triggering** - You can now set the trigger configurations of an AXIe based module to ARM In from multiple modules in the same chassis or another connected chassis/ logic analysis frame (in a multi-frame setup). An AXIe module can now receive ARM IN signals from an "OR'ed" combination of trigger out signals from multiple modules. Refer to the topic "To arm one module with another module's trigger" on page 164 to know more.

See Also

- "Version 05.50 What's New" on page 28
- "Version 05.40 What's New" on page 30
- "Version 05.30 What's New" on page 31
- "Version 05.20 What's New" on page 33
- "Version 05.00 What's New" on page 35
- "Version 04.00 What's New" on page 37
- "Version 03.83 What's New" on page 38
- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48

- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 05.50 What's New

In this release, version 05.50, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- **U4421A MIPI D-PHY Analyzer and Exerciser module** - The Keysight Logic and Protocol Analyzer application now adds support for a new module introduced with this release to send D-PHY stimulus and capture and decode D-PHY data. This module is installed in Keysight AXIe chassis and configured and controlled using the Keysight Logic and Protocol Analyzer application. Refer to the *U4421 D-PHY Analyzer and Exerciser Online Help* to know how to configure and use with module.
- **U4301A PCIe Gen3 Analyzer module** - You can now use the Keysight Logic and Protocol Analyzer application to create a physical layer tuning (.ptu) file for tuning the U4301A analyzer for a DUT. The creation and fine tuning of a ptu file is now GUI based. For more information, refer to the topic *Tuning the Analyzer for a Specific DUT* in the *U4301A PCIe Gen3 Analyzer Online Help*.
- **USB connectivity between AXIe chassis and host computer** - You can now connect the host computer to the AXIe chassis via PCIe or USB. The USB connectivity support has been added as a licensed option. Refer to the *AXIe based Logic Analysis & Protocol Test Modules Installation Guide* to know more.
- **B4621B, B4622B, and B4623B software packages** - These packages replace the B4621A, B4622A, and B4623A software packages and have the following new additions/updates -
  - **B4621B** - Decoder support for DDR4 added.
  - **B4623B** - Decoder support for LPDDR3 added.
  - **B4622B** -
    - The three tools contained in this toolset now support DDR, DDR2, DDR3, DDR4, LPDDR, LPDDR2, and LPDDR3. Several new tests have been added for DDR3/4 and LPDDR2/3.
    - A new tool *Real-Time Compliance Tool* has been added that evaluates DDR/LPDDR data in real- time to detect and report violations to compliance limits specified in DDR/LPDDR specifications. Refer to the *Real-Time Compliance Tool Online Help* to know more.
    - The DDR Validation Tool has been renamed to DDR Post Process Compliance Tool.
- DDR Setup Assistant and DDR Eyescan have been enhanced to include support for LPDDR3 and DDR4. These have also been updated to handle finding the chip select eyes in a separate measurement from the clock. Refer to the *DDR Setup Assistant online help* and *Modifying General or Target-specific Scan Qualifications* in this user guide to know more.

See Also

- "Version 05.40 What's New" on page 30
- "Version 05.30 What's New" on page 31
- "Version 05.20 What's New" on page 33
- "Version 05.00 What's New" on page 35
- "Version 04.00 What's New" on page 37
- "Version 03.83 What's New" on page 38
- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47

- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 05.40 What's New

In this release, version 05.40, of the Keysight Logic and Protocol Analyzer application, the following changes have been made:

- U4301A PCIe Gen3 Analyzer module
  - **LTSSM Overview pane added in the Protocol Viewer window** - You can now view the LTSSM states and their transitions for the PCIe data that you captured using the U4301 module. A new pane called LTSSM Overview has been introduced in the Protocol Viewer window to display these states and transitions. To know about this feature, refer to the topic *Viewing LTSSM States and Transitions* in the U4301 PCIe Gen3 Analyzer Online Help.
- U4154A Logic Analyzer module
  - **Eyescan enhancements** - The eyescan feature of the U4154A module has been enhanced to allow you to modify the general as well as DDR-specific scan qualification. To know about these enhancements, refer to the topic *Modifying General or Target-specific Scan Qualification* in this Online Help.

See Also

- "Version 05.30 What's New" on page 31
- "Version 05.20 What's New" on page 33
- "Version 05.00 What's New" on page 35
- "Version 04.00 What's New" on page 37
- "Version 03.83 What's New" on page 38
- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 05.30 What's New

In this release, version 05.30, of the Keysight Logic Analyzer application, the following changes have been made:

- U4301A PCIe Gen3 Analyzer module
  - **x8 bidirectional configurations supported on a single U4301A module** – With this release, a single U4301A module can support a bidirectional (upto x8) configuration. You can use the same U4301A module to probe and capture data in both upstream as well as downstream directions. For detailed information on how to use the U4301A module in bidirectional configurations, refer to the *PCI Express Gen3 Hardware and Probing Guide*. This guide is available on www.keysight.com and also installed with the Logic Analyzer software at <*Logic Analyzer Install location*>*help**pdfs*.
  - **U4324A PCIe Gen3 Flying Lead probe support added** - Besides the existing probing options, you can now use the new U4324A flying lead probes with the U4301A Analyzer module. For detailed information on this probe and its setup, refer to the PCI Express Gen3 Hardware and Probing Guide. This guide is available on www.keysight.com and also installed with the Logic Analyzer software at <*Logic Analyzer Install location*>*help**pdfs*.
- U4154A Logic Analyzer module
  - **A new license option** - 01G has been introduced for the U4154A module. You now have two license options, 01G and 02G available for the U4154A module. Based on the license option that you purchase, the state speed for the U4154A module is set. A "01G" license sets maximum state speed of U4154A to 1.4Gbs. A "02G option sets it to 2.5Gbs. These license options do not however control and set the timing speed of the module. Besides setting the state speed to 2.5Gbs, the 02G option also provides High Resolution (4.8 ps of time resolution) in eyescans which is not available with the 01G option.
  - **Eyescan enhancements** - The eyescan feature of the U4154A module has been enhanced to allow you to modify the unit interval settings, adjusting the sample position markers, enabling/disabling the marker placements to the center of the eye and setting sample positions to their suggested sample positions in a single click. To know about these enhancements, refer to the topic "Setting up and Running Eyescans in U4154A/B Logic Analyzer" on page 508 in this user guide.
- U4998A HDMI Protocol/Audio/Video Analyzer and Generator module
  - **MHL testing support added** - Besides supporting HDMI 1.4a and 1.4b, the module now supports the MHL protocol as well. You can now use this module to test MHL source and sink DUTs as well. To know more, refer to the *U4998A HDMI and MHL Protocol/Audio/Video Analyzer and Generator Online Help*.
  - The Protocol Viewer window has replaced the legacy Packet Viewer window. You can now use Protocol Viewer to display the data captured by the U4301A PCIe Gen3 Analyzer module. To know more about Protocol Viewer, refer to the topic Protocol Viewer Window in this user guide.

See Also

- "Version 05.20 What's New" on page 33
- "Version 05.00 What's New" on page 35
- "Version 04.00 What's New" on page 37
- "Version 03.83 What's New" on page 38
- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44

- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 05.20 What's New

In this release, version 05.20, of the Keysight Logic Analyzer application:

- You can now use the Keysight AXIe M9536A Embedded Controller module as the host computer with the AXIe based instrument modules installed in AXIe chassis. The instrument modules that support this controller module include U4998A HDMI Analyzer, U4301A PCIe Gen3 Analyzer, and U4154A Logic Analyzer modules. The Embedded Controller module is a one-slot module that needs to be installed in slot 1 of the AXIe chassis.
- Windows 7 support has been added for the 16800 and 16900 series of Logic Analyzers with serial number MY51420101 or higher. With the 05.20.0000 release of the Keysight Logic Analyzer application, 16800-series logic analyzers and 16900-series logic analysis systems are shipped from the factory with Windows 7 operating system. Refer to the 16800 and 16900 Install guides to know about the Windows 7 specific procedures, firewall settings and the new recovery procedure of 16800 and 16900 series of Logic Analyzers for Windows 7. These guides are installed with the Logic Analyzer software at <Logic Analyzer Install location>\help\pdfs.
- U4301 PCIe Analyzer module - The following changes have been made to the U4301A PCIe Gen3 Analyzer module.
  - The Protocol Viewer window now has an additional tab – Traffic Overview" in its lower pane. This tab displays the traffic statistics for the captured PCIe packets displayed in the upper pane of Protocol Viewer. It displays the count of captured packets based on packet types and the link direction (upstream or downstream). Refer to the topic Viewing PCIe Gen3 Packets in the U4301 PCIe Gen3 Analyzer online help to know more.
  - A new feature "CSV Export" has been added to the Protocol Viewer window. This feature allows you to export the displayed PCIe packet data from Protocol Viewer to a specified .csv file. Refer to the topic "Viewing PCIe Gen3 Packets" in the U4301 PCIe Gen3 Analyzer online help to know more.
- U4154A Logic Analyzer module - The following changes have been made to the U4154A module features.
  - A new license option - 01G has been introduced for the U4154A module. You now have two license options .01G and 02G available for the U4154A module. Based on the license option that you purchase, the state speed for the U4154A module is set. A "01G" license sets maximum state speed of U4154A to 1.4Gbs. A "02G option sets it to 2.5Gbs. These license options do not however control and set the timing speed of the module. Besides setting the state speed to 2.5Gbs, the 02G option also provides High Resolution (4.8 PS of time resolution) in eyescans which is not available with the 01G option.
  - The eyescans that you set up and run using the U4154A module now include a scan resolution selection feature. If you have the 02G license of the U4154A module, you can either choose a Normal or a High resolution for eyescans. To know more, refer to the topic "Setting up and Running Eyescans in U4154A/B Logic Analyzer" on page 508 to know more about the eyescan resolutions.
  - You can now export eyescan data from the Eye Scan - Sample Position and Threshold Settings dialog box to a specified .csv file. To know more, refer to the topic "Setting up and Running Eyescans in U4154A/B Logic Analyzer" on page 508.
  - The Transitional / Store Qualified Half Channel 5.0 GHz mode is now available as a Timing mode sampling option for the U4154A module. For details on this mode, refer to "U4154A Logic Analyzer Notes" on page 492.

See Also

- "Version 05.00 What's New" on page 35
- "Version 04.00 What's New" on page 37
- "Version 03.83 What's New" on page 38
- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40

- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 05.00 What's New

In this release, version 05.00, of the *Keysight Logic Analyzer* application:

- You can now use the Keysight Logic Analyzer application with the new U4154A Logic Analyzer module hosted in the new Keysight AXIe chassis (for example the M9502A portable 2-slot chassis). U4154A is a 136 channel AXIe based high speed state and timing logic analyzer.
- Trigger capabilities have been enhanced with more levels and deeper bursts (burst levels increased from 4 to 8). New event counter added for specifying the trigger action.
- 12.5 GHz Timing zoom support added for the U4154A Logic Analyzer module. The support has been added for state as well as timing sampling modes.
- The eye scan feature has been added for U4154A to determine optimal threshold voltage and sample position settings for individual channels and automatic adjustment of these settings. Eye scan triggers added to add trigger conditions for the samples that U4154A takes during the eye scan.
- One state clock provided to generate state clock signals for the U4154A logic analyzer module. Four state clock qualifiers have been added to allow you to add conditions to the state clock using qualifiers.
- The Force Prestore option added in the sampling trigger position.
- Advanced probe settings enabled for U4154A and 16962 logic analyzers. You can enable or disable the peaking at the channel/pod/module level for the probing system used for these logic analyzers.
- Two full channel modes (Full channel at 2.5 GHz and Transitional / Store qualified Full channel at 2.5 GHz) and one half channel mode at 5.0 GHz supported for U4154A. Quarter channel timing mode and Transitional / Store qualified Half channel mode not available for the U4154A Logic Analyzer module.
- DDR Setup Assistant tool enhanced to support the set up of the U4154A Logic Analyzer using this tool with automated steps for determining and setting up the sample positions for command, address, and data read and write signals.
- Support for Keysight U4998A HDMI Protocol/Audio/Video capture and generate (PAG) module added. You can now use the Keysight Logic Analyzer application with this module hosted in the Keysight AXIe or AMP chassis.
- Data storage qualifiers, ASPM (power management) support, and lane viewer added in the U4301A PCIe 8 Gbps protocol analyzer.

See Also

- "Version 04.00 What's New" on page 37
- "Version 03.83 What's New" on page 38
- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51

- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 04.00 What's New

- The *Keysight Logic Analyzer* application adds support for the new Keysight Digital Test Console chassis (for example the U4002A portable 2-slot chassis) and the U4301 PCIe Gen3 analyzer blade. See "U4301 PCIe Gen3 Analyzer—At a Glance" (in the online help).
- The DDR Setup Assistant has been added. The DDR Setup Assistant helps you properly set up a logic analysis system given the type of logic analyzer, DDR bus, and probing solution. Many of the set up steps are now automated. See "DDR Setup Assistant—At a Glance" (in the online help).
- The "LPDDR Bus Decoder" (in the online help) software is now available; it lets you view transactions, commands, and data captured on LPDDR buses.

See Also

- "Version 03.83 What's New" on page 38
- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.83 What's New

In version 03.83, of the *Keysight Logic Analyzer* application:

- Colorized DDR2/DDR3 eye scan measurements, using the 16962A logic analyzer module, were added. These measurements provide new DDR measurement capabilities:
  - Place sample points far more accurately than ever before to get the most reliable data capture.
  - View event qualified eye diagrams to assess both functional and parametric behavior of the device under test. With this new capability, you can rapidly view eye characteristics and identify design defects like inter-symbol interference and driver contention.
  - Gain signal integrity insight and identify problem signals quickly by simultaneously scanning all of the signals in your system.
- The N4851B MIPI D-PHY acquisition probe and N4861B MIPI D-PHY stimulus probe were supported. Also, the capabilities of existing N4851A and N4861A MIPI D-PHY probes are enhanced (as described below with the purchase of the N4851U-004).

  MIPI D-PHY analysis additions:
  - 3 and 4 lane analysis support at up to 940 Mb/s for the N4851B.
  - 3 and 4 lane analysis support at up to 800 Mb/s for the N4851A (requires purchase of the N4851U-004 to access this capability).

  Stimulus capabilities:
  - 3 lane stimulus support at up to 1 GB/s for the N4861B.
  - 3 lane stimulus support at up to 500 Mb/s for the N4861A (requires purchase of the N4851U-004 to access this capability).

  For more information, see "N4851A/B, N4861A/B MIPI D-PHY Acquisition/Stimulus Probes" (in the online help).
- The Add External Oscilloscope wizard was updated to support the latest Keysight oscilloscopes. For the complete list of supported oscilloscopes, see "External Oscilloscope Time Correlation and Data Display" (in the online help).
- The Add or Remove Keysight Logic Analyzer Software tool was updated to let you manage releases, that is, to let you remove old releases and free disk space. See Chapter 9, “Updating Software,” starting on page 299.

See Also

- "Version 03.82 What's New" on page 39
- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.82 What's New

In version 03.82, of the *Keysight Logic Analyzer* application:

- The new 16962A logic analyzer card was supported in the 16900-series logic analysis system. See "U4154A Logic Analyzer Notes" on page 492.

See Also

- "Version 03.80 What's New" on page 40
- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.80 What's New

In version 03.80 of the *Keysight Logic Analyzer* application:

- Timing analysis modes and additional triggering capabilities were added to the 16960A logic analyzer.
- The *Keysight Logic Analyzer* application supported the Windows® Vista operating system in addition to the Windows® XP operating system.
- An option in the Filter/Colorize tool to specify how the operation is applied across multiple data sources was added. See "To change the "across all data" option" (in the online help).
- An option to accumulate waveforms in the Waveform window was added. See "To accumulate waveforms" on page 206.
- There were additions to the Signal Extractor tool to allow it to convert timing analysis captures of serial buses into samples that can be decoded and displayed using the Packet Decoder and Packet Viewer. See "What's New in the Signal Extractor Tool" (in the online help).

See Also

- "Version 03.70 What's New" on page 41
- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.70 What's New

In version 03.70 of the *Keysight Logic Analyzer* application:

- Support for the new 16902B logic analysis system was added.
- The N4850A DigRF v3 acquisition probe for decoding and displaying DigRF v3 signals and the N4860A DigRF v3 stimulus probe for generating digital signals to emulate baseband IC or RF IC signals was supported. See "N4850/60A DigRF v3 Acquisition/Stimulus Probes" (in the online help).
- The Run Properties dialog that lets you save captured data after each run and stop after a certain number of repetitive runs was added. See Run Properties Dialog (see page 418).
- The Protocol Development Kit (PDK) editor added intelligent context-sensitive popup menus that show valid XML elements and attributes at the cursor location. These popup menus make editing protocol definition files easier. There have also been some minor improvements to the protocol description file XML elements and attributes, the Packet Decoder tool, and the Packet Viewer window. See "Protocol Development Kit (PDK)" (in the online help).
- The COM Connection Tool for testing the COM Automation connection between a controller PC and a logic analysis system was added. See "Test your Distributed COM connection" (in the online help).
- The "Signal Extractor" (in the online help) tool was enhanced with new commands and debugging features to help extract data from one input bus/signal and place it on multiple output buses/signals.
- The *Keysight Logic Analyzer* application required the Windows® XP operating system and no longer supported the Windows 2000 operating systems.

See Also

- "Version 03.67 What's New" on page 42
- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.67 What's New

In version 03.67 of the *Keysight Logic Analyzer* application:

- The Add or Remove Keysight Logic Analyzer Software tool was added to help you manage your logic analyzer software and keep it up to date. See Updating Software (see page 299).
- The "FPGA Dynamic Probe for Xilinx FPGAs" (in the online help) supported connecting to JTAG cables on remote PCs or logic analysis systems.
- The pattern generator was enhanced to: let you find instructions or vectors in the sequence, let you use macro, loop, and comment instructions when importing vectors from CSV format files (exporting to CSV format files still gives compiled sequences), and let you change the colors associated with instructions and macros. See "Using the Pattern Generator" (in the online help).
- The Add External Oscilloscope wizard was updated to set up an external oscilloscope module that can import digital channel data from mixed-signal oscilloscopes as well as analog channel data. Also, the update allowed external oscilloscope modules to be correlated with split logic analyzer modules. See "External Oscilloscope Time Correlation and Data Display" (in the online help).
- The external protocol analyzer correlation software supported the N5319A interconnect cable that provides more flexible cross-triggering (via flags). See "External Protocol Analyzer Time Correlation" (in the online help).
- The Status dialog was updated to show more information in the columns of the display and lets you choose which columns are displayed. See Status Dialog (see page 430).

See Also

- "Version 03.65 What's New" on page 43
- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.65 What's New

In version 03.65 of the *Keysight Logic Analyzer* application:

- The new 16951B logic analyzer card was supported in the 16900-series logic analysis system.
- The new "Protocol Development Kit (PDK)" (in the online help) lets you edit and create protocol description files in order to decode, display, and trigger on customized packet data.

See Also

- "Version 03.60 What's New" on page 44
- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.60 What's New

In version 03.60 of the *Keysight Logic Analyzer* application:

- You could control the new 16901A logic analysis system.
- The new 16950B logic analyzer card was supported in the 16900-series logic analysis system.
- The "FPGA Dynamic Probe for Altera FPGAs" (in the online help) add-in software was introduced; it lets you probe signals internal to FPGAs.
- You could set up an external protocol analyzer and a logic analyzer to cross-trigger and make time-correlated measurements (using markers). See "External Protocol Analyzer Time Correlation" (in the online help).
- You could open ALA format configuration files "read-only" in any *Keysight Logic Analyzer* application instance (without licenses for tools, windows, etc. that may be in the configuration). See ALA vs. XML, When to Use Each Format (see page 352).

See Also

- "Version 03.55 What's New" on page 45
- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.55 What's New

In version 03.55 of the *Keysight Logic Analyzer* application:

- The E9524A MicroBlaze trace toolset updated the "inverse assembler" (in the online help) to support MicroBlaze version 5.

See Also

- "Version 03.50 What's New" on page 46
- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.50 What's New

In version 03.50 of the *Keysight Logic Analyzer* application:

- You could control the new 16800-series logic analyzers.
- You could view decoded packet data in the new Packet Viewer window. See Analyzing Packet Data (see page 259).
- The "Xilinx FPGA Dynamic Probe" (in the online help) added support for MicroBlaze trace cores (MTC) and there was a new E9524A MicroBlaze trace toolset that provides "inverse assembly" (in the online help).
- You could use the new Demo Center (see page 71) application to view logic analysis system feature demonstrations.
- You could use the new "Signal Extractor" (in the online help) tool to extract data from one input bus/signal and place it on multiple output buses/signals. This is useful for extracting I and Q data from simple serial protocols or remultiplexing high-speed digital data that has been demultiplexed onto additional logic analyzer channels.
- You could connect to another logic analysis system without having to go *offline* first. See Connecting to a Logic Analysis System (see page 75).
- There were improvements for multiframe logic analysis systems (see page 353), especially for *performance multiframe configurations*.
- You could rename the bits of a bus and view them when the bus is expanded in the Waveform display window. See To rename the bits of a bus (see page 96).
- You could add separator rows to the Waveform display window. See To insert separator rows (see page 204).
- A graphical trigger was added when viewing module details in the Status dialog (see page 430).
- You could import external data into the logic analysis system from module binary (ALB) format files as well as module CSV format text files. See Using Data Import Module.
- There was a "Force Prestore" option to ensure that the specified percentage of pre-trigger memory is filled before the logic analyzer begins looking for a trigger. Timing mode used to behave this way while state mode did not. With the "Force Prestore" option, you can choose the desired behavior for both sampling modes. See To specify the trigger position (see page 115).
- You could use the logic analysis system's 10 MHz CLOCK IN input to keep a logic analyzer and an external oscilloscope in sync over long acquisitions. See " Correlation Drift Over Long Acquisitions" (in the online help).
- You could export Listing display window data to 16700 ASCII format files. See To export data to 16700 ASCII format files (see page 183).

See Also

- "Version 03.30 What's New" on page 47
- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.30 What's New

In version 03.30 of the *Keysight Logic Analyzer* application:

- You could add an external oscilloscope without the time correlation fixture. Default (typical) deskew values for your logic analyzer and oscilloscope models were used. You could still use the time correlation fixture to get the most accurate deskew values for the actual instruments being used. See " External Oscilloscope Time Correlation and Data Display" (in the online help).
- You could import external oscilloscope data and view it faster with import and waveform drawing performance improvements.
- You could activate software licenses, access floating license servers, and borrow floating licenses much easier using the new license manager interface. See Managing Software Licenses (see page 293).
- You could import external data into the logic analysis system and analyze it just like data acquired by logic analyzer modules. See Using Data Import Modules.
- You could use these new VBA (Microsoft Visual Basic for Applications) macros and VbaView windows:
  - *SendToExcel* VBA macro that sends captured data to Microsoft Excel.
  - *SendToPatternGeneratorModule* VBA macro that sends captured data to a pattern generator module.
  - *Timing Compare* VbaView window that compares timing analyzer data with a specified tolerance (plus or minus a number of samples).

  See Tools Menu (see page 362), Windows Menu (see page 368), and the online help included with the macro or VbaView window.
- You could use MatLab in conjunction with the logic analysis system more easily with the new MatLab Connectivity and Analysis package. This licensed package contains these VBA macros and VbaView windows:
  - *SendToMatLab* VBA macro that sends logic analyzer data to MatLab.
  - *MatLab Analysis* VbaView window that sends logic analyzer data to MatLab for processing and displays the results in an XY scattergram chart.
  - *FFT* VbaView window that performs a Fast Fourier Transform on logic analyzer data and displays the results in a line chart.

  See the online help included with the VBA macro and VbaView windows.
- You could use additional remote programming commands with the updated Remote Programming Interface (RPI) compatibility package. See "Using the Remote Programming Interface (RPI)" (in the online help).

See Also

- "Version 03.20 What's New" on page 48
- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.20 What's New

In version 03.20 of the *Keysight Logic Analyzer* application:

- The 16760A logic analyzer card was supported in the 16900-series logic analysis system.
- You could import data from external oscilloscopes. The Keysight 6000-series oscilloscopes were added to the list of supported oscilloscopes. Also, a new VbaView window let you use the oscilloscope's web interface from within the *Keysight Logic Analyzer* application. See " External Oscilloscope Time Correlation and Data Display" (in the online help).
- The "Serial To Parallel" (in the online help) tool for converting serial data streams into parallel bus data was introduced.
- The "Xilinx FPGA Dynamic Probe" (in the online help) add-in software was updated to support automatic set up features.
- Pattern generator compiled vectors were saved in XML format configuration files as well as ALA format configuration files. See "Using the Pattern Generator" (in the online help).
- Microsoft Visual Basic for Applications (VBA) macros and VbaView windows were saved in XML format configuration files as well as ALA format files. See "Distributing VBA Code" (in the online help).
- You could disable multiple modules at the same time and choose whether or not to remove their connections to tools and display windows. When you leave connections, buses/signals associated with disabled modules remain in display windows, but they have no data. See To disable and enable modules (see page 88).

See Also

- "Version 03.00 What's New" on page 49
- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 03.00 What's New

In version 03.00 of the *Keysight Logic Analyzer* application:

- The 16720A pattern generator card was supported in the 16900-series logic analysis system. See "Using the Pattern Generator" (in the online help).
- The *eye finder* feature for automatically adjusting state mode sampling positions was enhanced to automatically determine optimal threshold voltage settings as well. See Understanding State Mode Sampling Positions (see page 339).
- Microsoft Visual Basic for Applications (VBA) was been integrated into the *Keysight Logic Analyzer* application to let you easily automate measurements and add custom control and data visualization tools.

See Also

- "Version 02.50 What's New" on page 50
- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 02.50 What's New

In version 02.50 of the *Keysight Logic Analyzer* application:

- The *eye finder* feature for automatically adjusting sampling positions in state mode was made available for 1680- and 1690-series logic analyzers (this feature was previously only available for logic analyzers in the 16900-series logic analysis system).
- The "Xilinx FPGA Dynamic Probe" (in the online help) add-in software was introduced; it lets you probe signals internal to FPGAs.
- The "General Purpose Probe" (in the online help) add-in software was added; it lets you set up probe definitions in order to:
  - Cause device under test pin/pad numbers to appear in the Buses/Signals setup tab.
  - Document logic analyzer pod connections.
  - Prepare for importing netlist files to automatically define buses/signals.
- The "PCI Express Analysis Probe" (in the online help) add-in software (for the N4220A/B PCI Express analysis probe) was introduced, and the new Packet Decoder tool was used to decode PCI Express bus data.
- In the window, a separate column for probes (general purpose, FPGA dynamic, PCI Express, etc.) was added, and you were able to access probe properties from there.

See Also

- "Version 02.00 What's New" on page 51
- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 02.00 What's New

In version 02.00 of the *Keysight Logic Analyzer* application:

- You could control the new 16900-series logic analysis system and access the new features associated with it. The same *Keysight Logic Analyzer* application controls the 1680- and 1690-series logic analyzers and can be used by itself on Windows XP/2000 computers for *remote access* (see page 75) or *offline analysis* (see page 194).
- The ability to split logic analyzer modules was added. This is useful for analyzing buses with different clocks.

  With multiple logic analyzer modules, the ability to identify which is the system trigger and specify the skew between the modules was added.
- The *eye finder* feature for automatically adjusting sampling positions in state mode was added for logic analyzers in the 16900-series logic analysis system.
- The ability to load program symbols from compiler-generated output files was added.
- The ability to display the high-level language source code associated with captured data and set up triggers based on source code locations was added in the new Source display window.
- A new dual-sample state sampling clock mode, often used to capture DDR memory bus activity, was added.
- The Netlist Import dialog was reorganized and a separate dialog for defining probes was added.
- New System Summary and Status dialogs for displaying information about the logic analysis system were added.
- Performance improvements were made: waveform drawing became much faster, and the *Keysight Logic Analyzer* application was made into a multi-threaded application.

See Also

- "Version 01.40 What's New" on page 52
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 01.40 What's New

Version 01.40 of the *Keysight Logic Analyzer* application:

- Let you display the same data in multiple windows by using the window (which replaced the previous Tool dialog); for example, you could display the same data filtered in one Listing window and unfiltered in another Listing window.
- In the Buses/Signals dialog, you could reorder bits assigned to a bus/signal name. This is useful in cases where buses in the device under test are not probed by consecutive logic analyzer channels.
- Also in the Buses/Signals dialog, you could import bus/signal names and assignments from a netlist file.
- Let you time-correlate data captured by the logic analyzer to data captured by external Infiniium oscilloscopes using the E5850A time-correlation fixture.
- Let you control logic analyzers with remote programs that communicate with the Instrument COM Automation Server, included with the *Keysight Logic Analyzer* application.
- Let you save more logic analyzer configuration information to generic XML configuration files. You could save trigger specifications in XML format and transfer them between different logic analyzers. Also, you could use portions of generic XML configuration files with several of the COM automation commands.
- Let you show statistics in interval markers and stop a repetitive measurement or send an e-mail message on particular statistic values.
- Let you use the Compare window's properties, and the Difference Properties tab, to specify the run until options. (The Compare window's Run Until button was removed.)
- Let you use the *quick pick* bus/signal name selection feature to quickly select from recently used names. This feature was added to the Filter/Colorize tool properties dialog and to the advanced trigger setup dialog.

- When exporting data to comma-separated value (CSV) files, you could choose to export data from selected buses/signals. Also, you could select the number base of the data that is exported.
- In the advanced trigger setup dialog, you could re-group events.
- Let you lock the scrolling on display windows (for example, Waveform, Listing, or Compare) so that when one window is scrolled, all locked windows are scrolled as well. You can find the *Lock* property with other windows property settings.
- Let you set the delay between repetitive measurements so that you could look at the captured data before deciding whether to stop the measurement.

See Also
- "Version 01.20 What's New" on page 53
- "Version 01.10 What's New" on page 54

Version 01.20 What's New

Version 01.20 of the *Keysight Logic Analyzer* application:

- Let you, in the timing (asynchronous) sampling mode, trigger on multiple glitches/edges.
- Let you perform a simple compare of captured data sets.
- Added the ability to save/load user-defined symbols to/from XML format logic analyzer configuration files.

  This meant you could use text processing tools to re-format symbol information from software development tools, insert them into an XML format configuration file, and load them into the *Keysight Logic Analyzer* application.
- Contained an API and documentation to enable inverse assembler and analysis tool development.
- Extended its offline analysis capability to 16700-series logic analyzers.

See Also

- "Version 01.10 What's New" on page 54

Version 01.10 What's New

Version 01.10 of the *Keysight Logic Analyzer* application:

- Added a Japanese version of the online help.

2 Getting Started

Tutorial - Getting to know your logic analyzer / 56
Measurement Examples / 66
Demo Center / 71

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 326)
- When should you use a logic analyzer? (see page 327)
- What is a logic analyzer? (see page 328)

  Timing analyzer:
  - Sampling clock (see page 328)
  - Sampling (see page 328)
  - Triggering (see page 329)

  State analyzer:
  - Sampling clock (see page 330)
  - Sampling (see page 331)
  - Triggering (see page 331)

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 56)
- Connecting to the device under test (see page 57)
- Setting up bus/signal names (see page 58)
- Setting the acquisition mode (see page 59)
- Setting up a simple trigger (see page 59)
- Open the tutorial configuration file (see page 60)
- Using markers (see page 60)
- Zooming in on the data (see page 61)

Time saving tasks

- Loading and saving configuration files (see page 62)
- Saving and recalling trigger setups (see page 62)
- Quick marker measurements (see page 63)
- Searching data (see page 64)
- Toolbars, tool tips, and mouse shortcuts (see page 65)

See Also

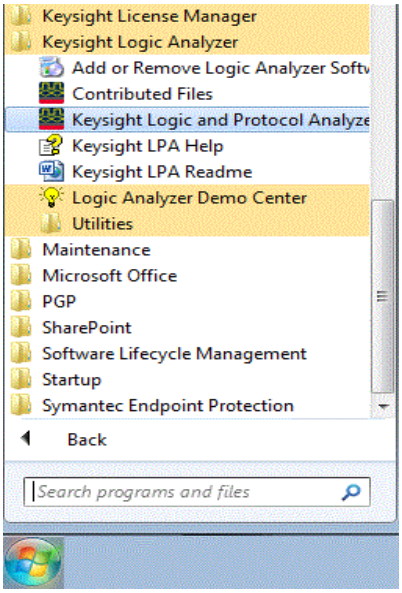
- Product Overviews (see page 482)

Turning on the logic analyzer

[Tutorial Home (see page 56)] [Next Topic (see page 57)] [Previous Topic (see page 331)]

1 Plug in the power cable and press the front-panel On/Off button.

2 From the Windows Start bar, click **Start>Programs>Keysight Logic Analyzer>Keysight Logic Analyzer**.



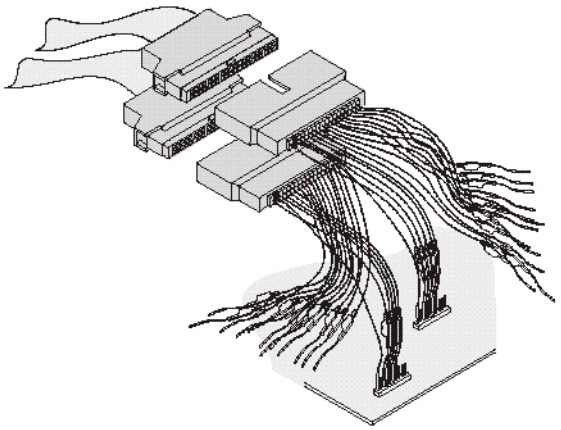
NOTE Optional: If you have a logic analyzer shortcut icon on screen, double-click the icon.

Connecting to the device under test

[Tutorial Home (see page 56)] [Next Topic (see page 58)] [Previous Topic (see page 56)]

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 73).

Setting up bus/signal names

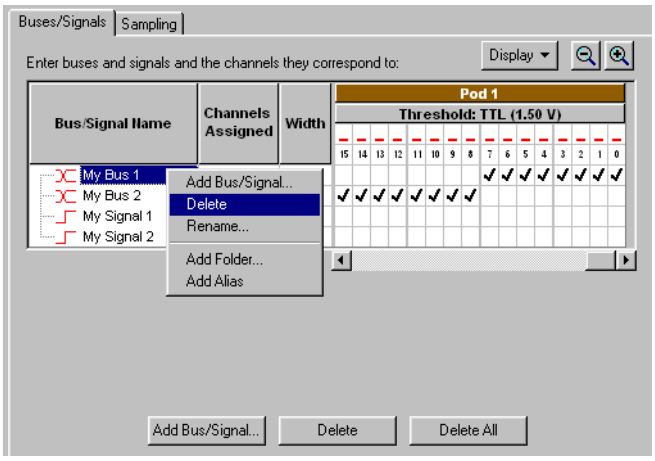
[Tutorial Home (see page 56)] [Next Topic (see page 59)] [Previous Topic (see page 57)]

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

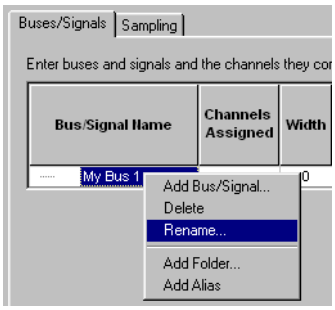
1. In the menu bar click **Setup>(Logic Analyzer Module)>Bus/Signal...**.
2. In the Analyzer Setup dialog that appears, right-click on **My Bus 1**, then select **Delete**. Repeat until all bus signal names are deleted. After the last bus/signal is deleted, "My Bus 1" appears again as a default name.

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

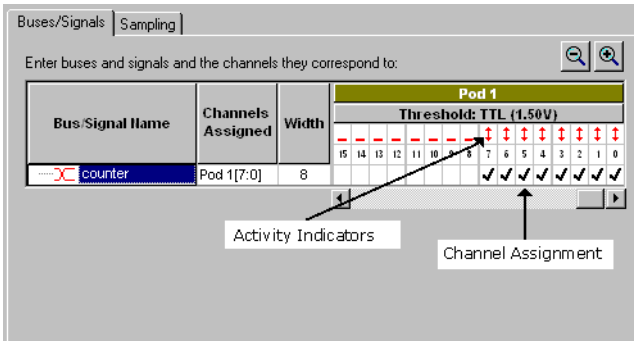
The logic analyzer must be told which probed signals from the device under test are to be included in the measurement, and how you want them grouped. In this exercise, you assign channels 0 - 7 on Pod 1 under the name "counter". Notice that when more than one channel is assigned to "counter" it becomes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



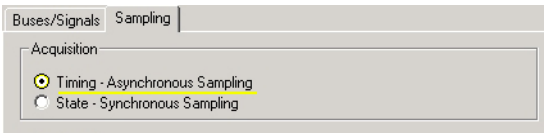
Setting the acquisition mode

[Tutorial Home (see page 56)] [Next Topic (see page 59)] [Previous Topic (see page 58)]

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**.

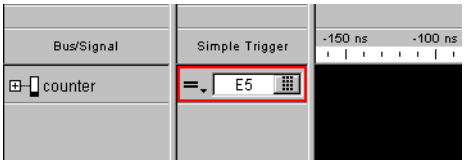
Setting up a simple trigger

[Tutorial Home (see page 56)] [Next Topic (see page 60)] [Previous Topic (see page 59)]

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 56)] [Next Topic (see page 60)] [Previous Topic (see page 59)]

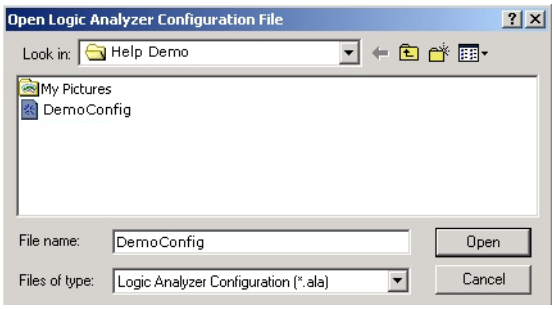
At this point in a measurement, you would normally run the logic analyzer. However, because you are not connected to a device under test, you cannot capture real data. You will have to load a configuration file that contains this data.

Load the configuration file

1 Select **File>Open**.

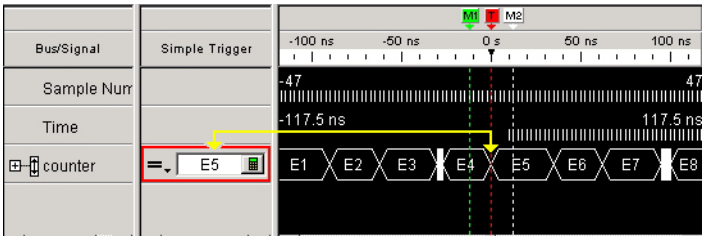
2 From the file manager dialog, select the file named **DemoConfig.ala** from the following directory: **C:\Documents and Settings\All Users\Documents\Keysight Technologies\Logic Analyzer\Default Configs\Keysight\Help Demo**

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 56)] [Next Topic (see page 61)] [Previous Topic (see page 60)]

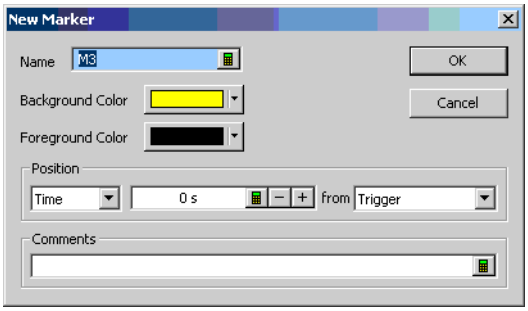
Markers are used for creating reference points in data. Once markers are placed in data, you can use them to quickly see what time, sample, or data value the marker is set on.

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

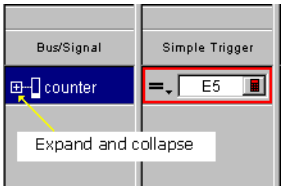
[Tutorial Home (see page 56)] [Next Topic (see page 62)] [Previous Topic (see page 60)]

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.

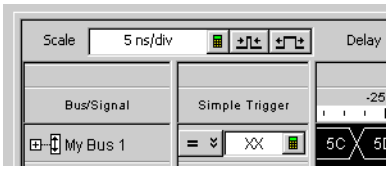
Expand a bus

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 56)] [Next Topic (see page 62)] [Previous Topic (see page 61)]

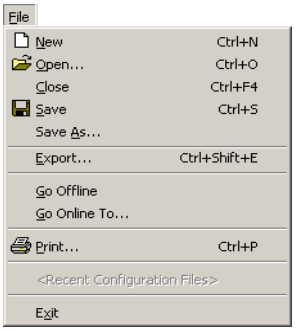
Many times it is quicker to open an existing configuration file with a similar setup than to create a new configuration from scratch. You simply open a similar file, make the appropriate changes to the setup, then save the file as a new filename.

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 56)] [Next Topic (see page 63)] [Previous Topic (see page 62)]

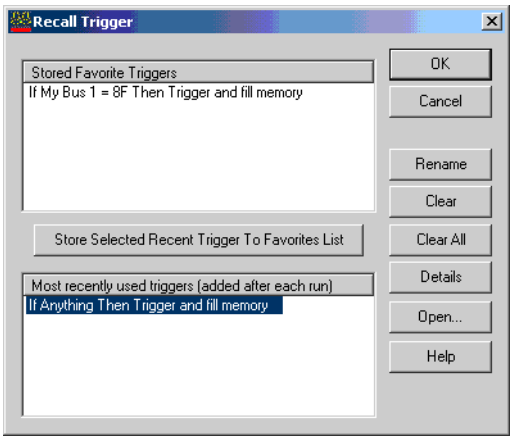
Each time you set up a new trigger and run the measurement, the trigger setup is saved in the logic analyzer. It is quicker to recall a trigger setup rather than re-configure the trigger setup each time.

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.

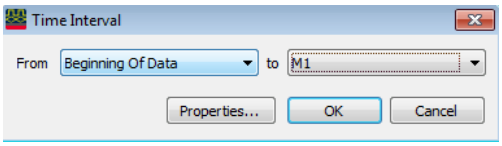


Quick marker measurements

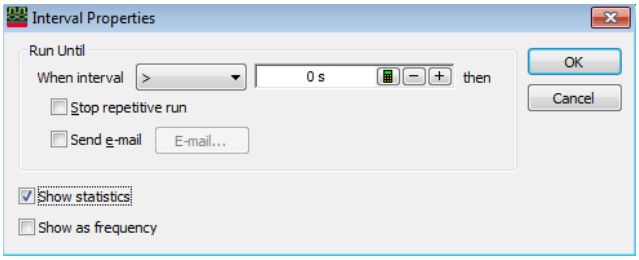
[Tutorial Home (see page 56)] [Next Topic (see page 64)] [Previous Topic (see page 62)]

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**.

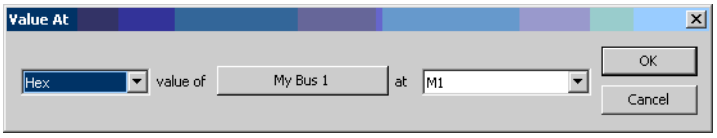


Click **OK** to close the Interval Properties dialog, and click **OK** to close the Time Interval dialog.

After a repetitive run, the result of the time interval measurement is displayed in the marker measurements display bar.

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

is displayed in the marker measurement display bar.

Searching data

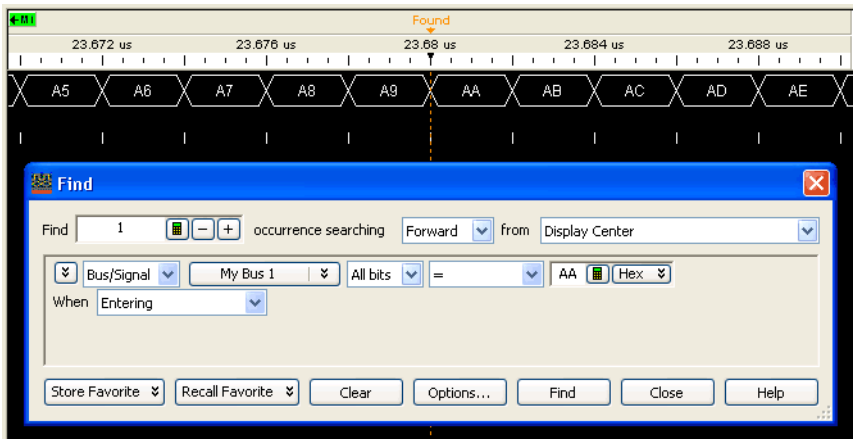
[Tutorial Home (see page 56)] [Next Topic (see page 65)] [Previous Topic (see page 63)]

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 56)] [Previous Topic (see page 64)]

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

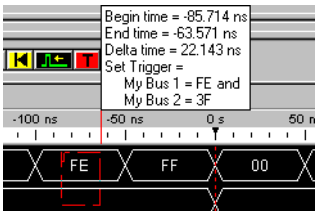
Below the menu bar are groups of icons that represent shortcuts to many dialogs and features. For more information refer to Toolbars (see page 371) in the main help.

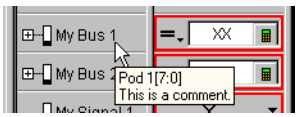
Mouse Shortcuts

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 104), 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 66)
- Making a state analyzer measurement (see page 67)
- To trigger on one of multiple edges or glitches (see page 68)
- To trigger on ranges (see page 68)

See Also

- Tutorial - Getting to know your logic analyzer (see page 56)
- Timing Mode Trigger Functions (see page 440)
- State Mode Trigger Functions (see page 452)

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 56)" for a quick tutorial on logic analysis concepts and measurements.**

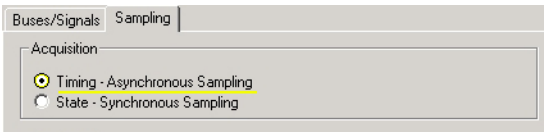
1. Connect the probing to the device under test (see Probing the Device Under Test (see page 73) for more information).
2. Turn on the logic analyzer.

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 95) existing names, or deleting (see page 95) and creating (see page 94) new names.
3. From the Buses/Signals tab, define buses and signals (see page 93) 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 107).
4. Set the timing mode Sampling Period (see page 108).

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 132).
   - From the Advanced Trigger dialog (see page 392), set up a timing mode advanced trigger (see page 440) function.

Run the measurement

1 Run (see page 174) the measurement.

See Also

- To specify the trigger position (see page 115)
- To set acquisition memory depth (see page 115)

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 56)" 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 73) 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.**

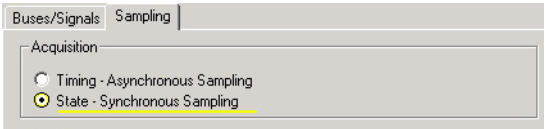
2 Turn on the logic analyzer.

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 95) existing names, or deleting (see page 95) and creating (see page 94) new names.
3 From the Buses/Signals tab, define buses and signals (see page 93) 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 109).
4 Set the state sampling clock (see page 113).

5 If necessary, set the advanced state clocking (see page 114).

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 132).
   - From the Advanced Trigger dialog (see page 392), set up an advanced trigger (see page 452) function.

Run the measurement

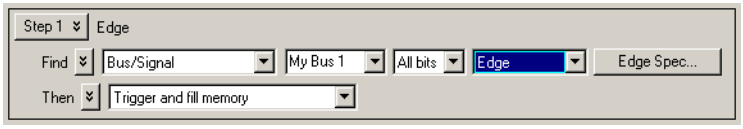
1 Run (see page 174) the measurement.

See Also

- To specify the trigger position (see page 115)
- To set acquisition memory depth (see page 115)

To trigger on one of multiple edges or glitches

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 or glitches.
3 Select **All bits** in the bus.
4 Select **Edge**.



5 Click **Edge Spec...**.
6 In the Set Edge/Glitch dialog, specify edges or glitches 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/Glitch dialog.
8 Click **OK** to close the Advanced Trigger dialog.

NOTE **Glitches are not drawn on the screen. You need an oscilloscope to further troubleshoot glitches and find out when they occur.**

To trigger on ranges

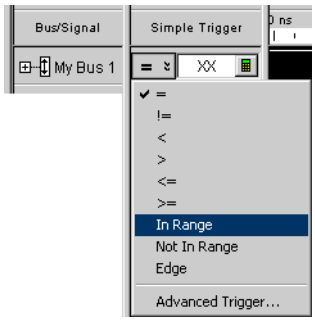
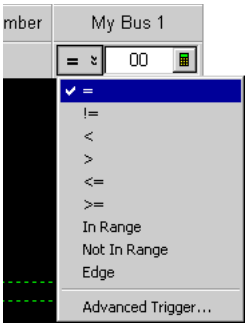
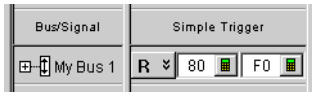
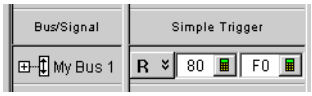
NOTE

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 132)

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

	Waveform Display Window	Listing/Compare/Source Display Window
Bus Range Trigger		
		

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 419).

3 From the menu bar, choose **Run/Stop>Run**, or click the ▶ icon from the run/stop toolbar (see page 374).

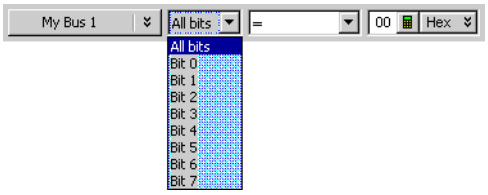
When setting up advanced triggers (see page 138)

1 Click Trig in the analyzer setup toolbar, or choose **Setup>(Logic Analyzer Module)>Advanced Trigger...** from the menu bar.

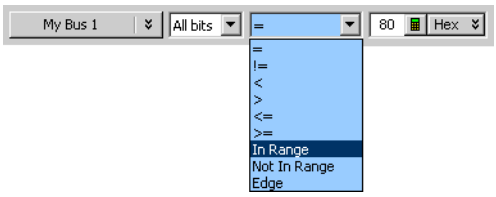
2 In the Advanced Trigger dialog, select the bus.

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 419).

7 From the menu bar, choose **Run/Stop>Run**, or click the icon from the run/stop toolbar (see page 374).

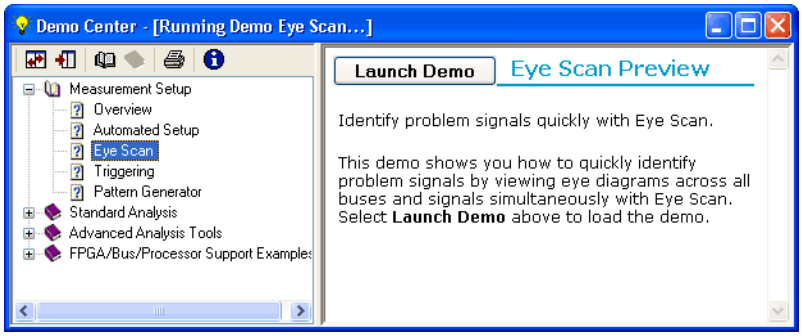
Demo Center

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 74)

To control signals in the device under test

The 16850 series logic analysis system *frames* (see page 539) 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 437), 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 421), 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 84)), 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 77)
- To delete a logic analysis system from the list (see page 78)
- To refresh the logic analysis system list (see page 79)
- To view logic analysis system details (see page 80)
- To enter your "System In Use" comments (see page 81)
- To select a logic analysis system for auto-connect (see page 82)
- To chat with another logic analysis system user (see page 83)

See Also

- Offline Analysis (see page 194)

To add a logic analysis system to the list

1 In the Select System to Use dialog (see page 421) (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 75)

To delete a logic analysis system from the list

1 In the Select System to Use dialog (see page 421) (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 75)

To refresh the logic analysis system list

1 In the Select System to Use dialog (see page 421) (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 *Keysight Logic Analyzer* 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 75)

To view logic analysis system details

1 In the Select System to Use dialog (see page 421) (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 407).

See Also

- Returning to Online Analysis (see page 75)

To enter your "System In Use" comments

1. In the Select System to Use dialog (see page 421) (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 75)

To select a logic analysis system for auto-connect

1. In the Select System to Use dialog (see page 421) (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 75)

To chat with another logic analysis system user

1. In the Select System to Use dialog (see page 421) (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 398), 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 397), 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 75)

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 421).)

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 75)

5 Setting Up the Logic Analyzer

Configuring Logic Analyzer Modules / 88
Setting Up Probes (in the online help)
Setting the Logic Analyzer Threshold Voltage / 91
Defining Buses and Signals / 93
Choosing the Sampling Mode / 106
Setting Up Symbols / 118
Installing Licensed Hardware Upgrades / 124

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 538) or several cards, and a single card or several card module can be split into two modules. Modules give you the flexibility to:

- Increase logic analyzer channel count by using more than one card.
- Probe buses that have different time domains by splitting a single card into two modules.

Combining cards to increase channel count is done when installing cards into a *frame/chassis* (see page 539 and page 538). Cables are connected between the cards. For more information, see the *"AXIe based Logic and Protocol Analysis – Installation Guide"*.

CAUTION **You must power down the AXIe chassis before inserting, replacing, or removing the U4154A/B Logic Analyzer module. The enclosure surface of the U4154A/B 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.**

Splitting a single card (or cards that were combined during installation) is done from the Setup menu or in the window. To split logic analyzer channels into two modules:

1 Choose **Setup>(Logic Analyzer Module)>Split Analyzer...**.

2 In the Split Analyzer Setup dialog (see page 429), drag the channel assignment slider to specify the number of channels in each analyzer.

If you'd like to assign particular *pods* (see page 540) or *pod pairs* (see page 540) to a module, check **Advanced Pod Assignment Mode**.

3 Click **OK**.

See Also
- To disable and enable modules (see page 88)
- Memory Depth and Channel Count Trade-offs (see page 335)

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.

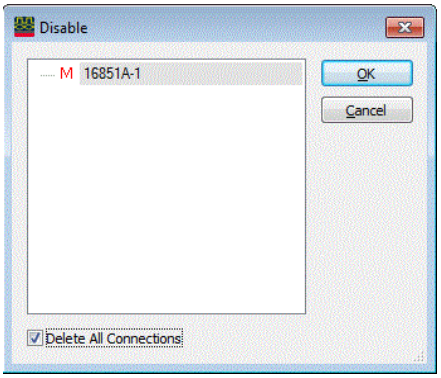
NOTE **Disabling a module in a split analyzer causes both modules of the split analyzer to be disabled.**

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.

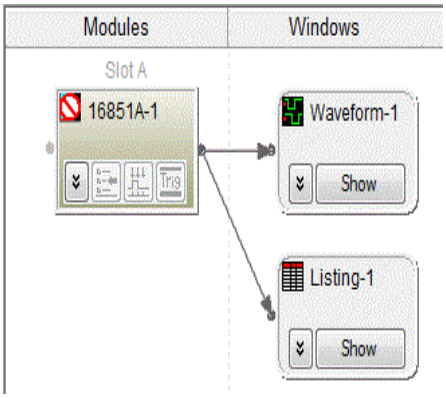
Uncheck **Delete All Connections** if you want to leave module connections to tools and windows. The disabled modules' buses and signals are hidden from tools and windows, but they will reappear when the modules are re-enabled.

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 widow, disabled modules have the ⃠ icon:

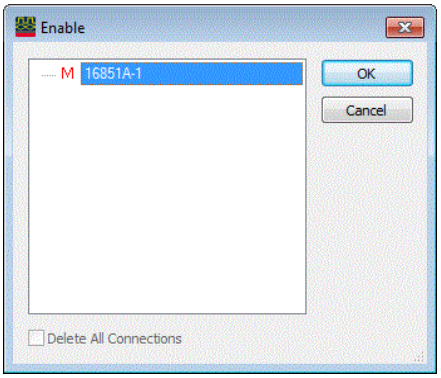


To enable modules

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

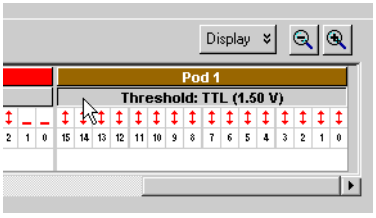
- Configuring Logic Analyzer Modules (see page 88)

Setting the Logic Analyzer Threshold Voltage

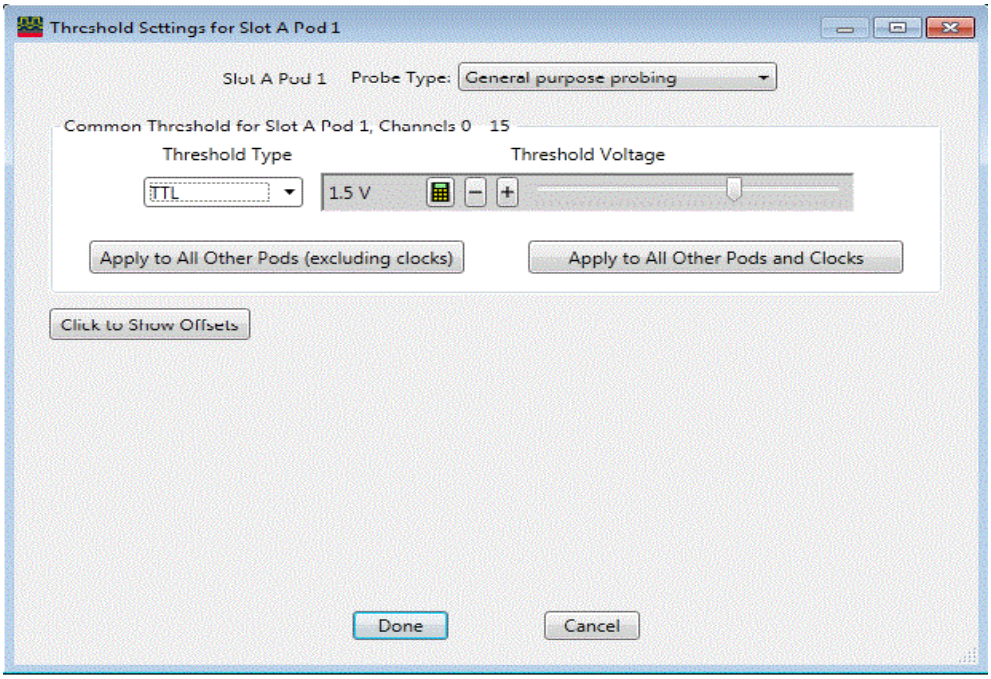
It is very important that you specify a threshold voltage that matches what your device under test is using. Incorrectly specified threshold voltages result in incorrect data.

1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**.

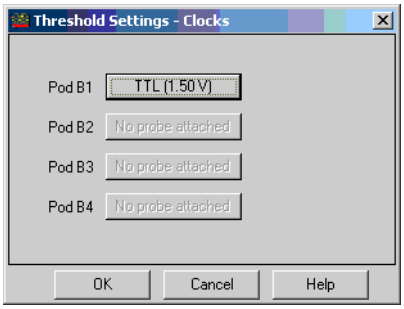
2 In the Buses/Signals Setup dialog, click any **Threshold** button. The Threshold buttons are located under the Pod or Clocks label.

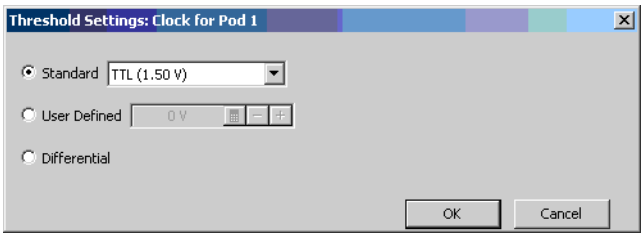


3 In the Threshold Settings dialog:



Some logic analyzers let 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.





a Check **Apply settings to all pods** if you want the settings to apply to all pods; otherwise, the settings apply only to the selected pod.

b Check **Apply settings to clock bits** if you want the settings to apply to clock bits; otherwise, the settings apply only to the data channels.

c Select the **Probe** type; this may affect the settings that can be selected.

d Specify the threshold level. Choices are:

- Standard – you can select from a list of pre-defined threshold levels (which are shown in parentheses).
- User Defined – you can enter a value from -6.00 to 6.00 V.
- Differential.
- External Reference – some probes have external reference inputs that can define the threshold setting.

See Also
- "Pod and Channel Naming Conventions" on page 333
- "Pod and Channel Naming Conventions in U4154A/B Logic Analyzer" on page 497

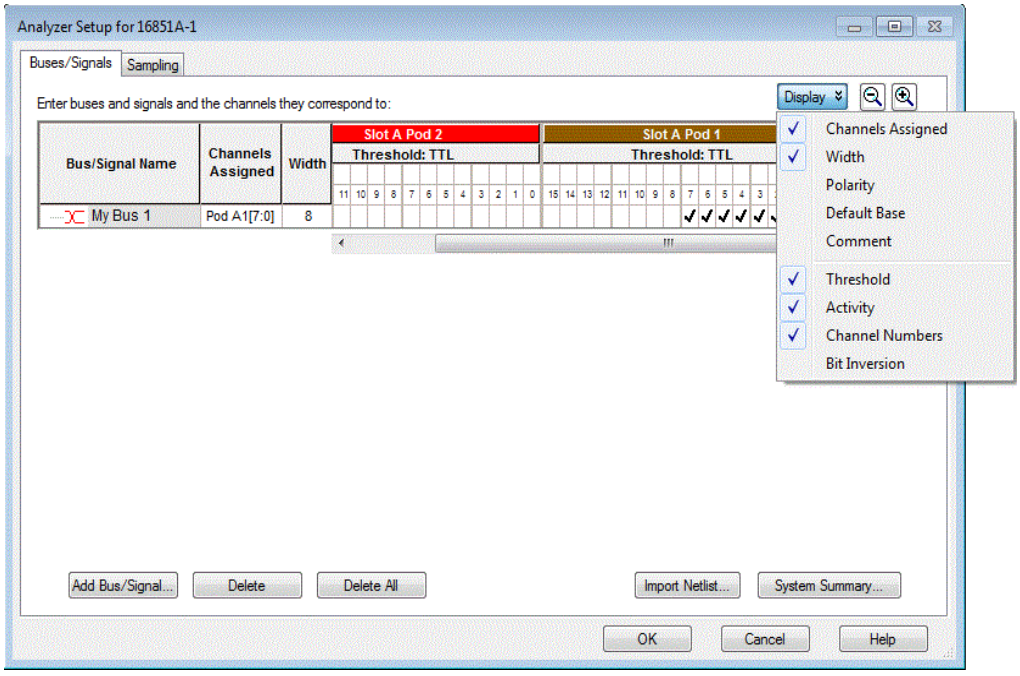
Defining Buses and Signals

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 395).



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

- To add a new bus or signal (see page 94)
- To delete a bus or signal (see page 95)
- To rename a bus or signal (see page 95)
- To rename the bits of a bus (see page 96)
- To assign channels in the default bit order (see page 98)
- To assign channels, selecting the bit order (see page 99)
- To use clock channels as extra data channels (see page 99)
- To define buses and signals by importing netlist files (see page 100)
- To reorder bits by editing the Channels Assigned string (see page 101)
- To set the default number base (see page 103)
- To set polarity (see page 104)
- To add user comments (see page 104)
- To add a folder (see page 105)
- To alias a bus/signal name (see page 105)
- To sort bus/signal names (see page 105)

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)).

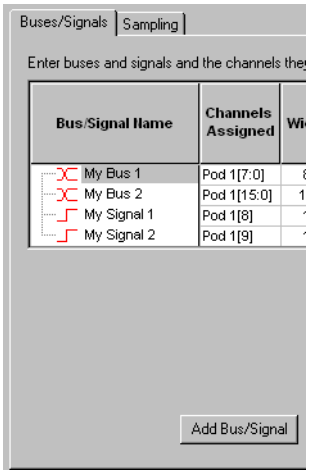
See Also
- Setting the Logic Analyzer Threshold Voltage (see page 91)
- Why Are Pods Missing? (see page 334)

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 95) 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 95)
- To rename a bus or signal (see page 95)
- To assign channels in the default bit order (see page 98)
- To assign channels, selecting the bit order (see page 99)
- Defining Buses and Signals (see page 93)

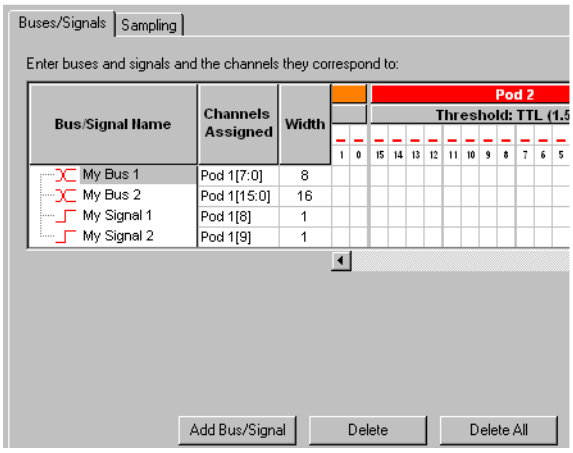
To delete a bus or signal

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 95)
- To delete all buses and signals (see page 95)

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.

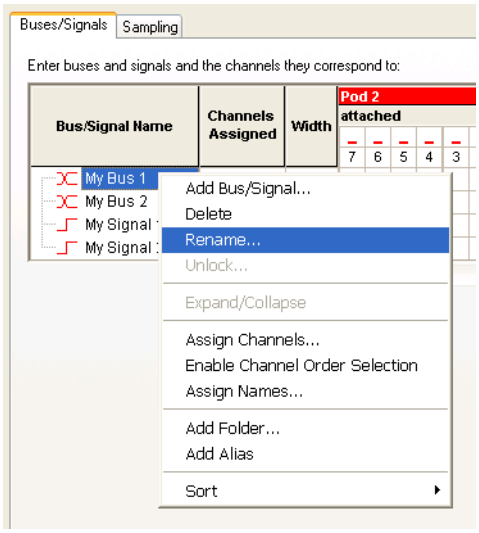
See Also

- Defining Buses and Signals (see page 93)

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 372).
2 Right-click the bus or signal name and choose **Rename...**



3 Enter the new bus or signal name.

4 Select **OK**.

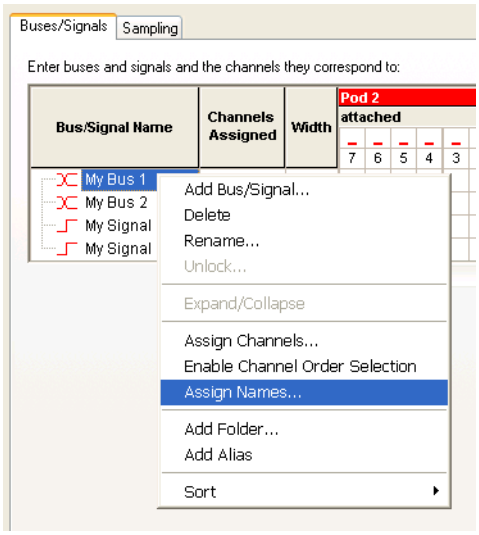
See Also
- To add a new bus or signal (see page 94)
- To delete a bus or signal (see page 95)
- To rename the bits of a bus (see page 96)
- Defining Buses and Signals (see page 93)

To rename the bits of a bus

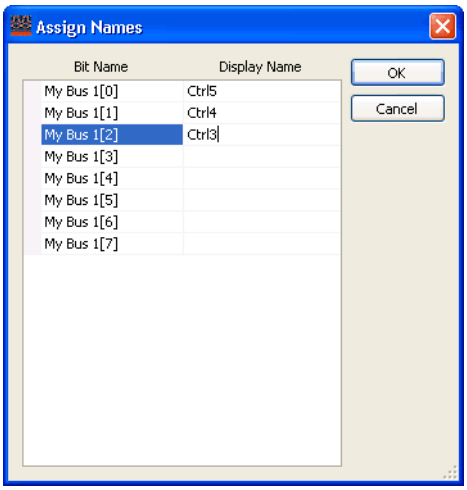
When a bus is expanded in the Waveform display window, the names of the signals within the bus are like Bus[0], Bus[1], etc., by default. However, you can assign your own names to the bits within a bus.

1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**, or click the icon in the setup toolbar (see page 372).

2 Right-click the bus name and choose **Assign Names...**.

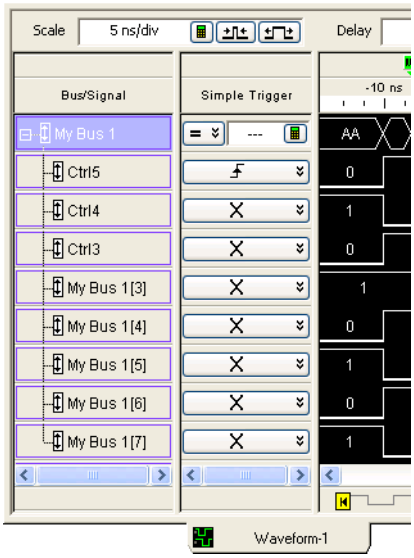


3 In the Assign Names dialog, enter the new names of bits within the bus.



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 95)
- To add a new bus or signal (see page 94)
- Defining Buses and Signals (see page 93)

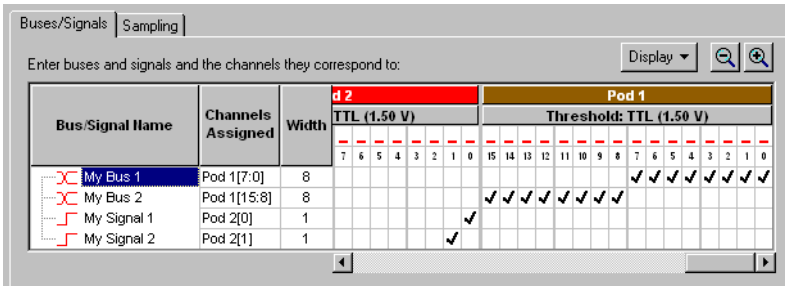
To assign channels in the default bit order

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.



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 100)
- To reorder bits by editing the Channels Assigned string (see page 101)
- Pod and Channel Naming Conventions (see page 333)

To assign channels, selecting the bit order

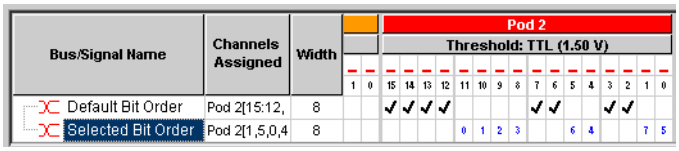
In cases where buses in the device under test haven't been probed with consecutive logic analyzer channels, you can assign channels to a bus name in a selected bit order.

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.

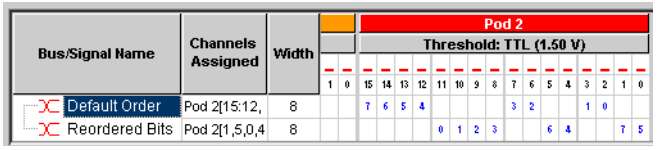
The bit numbers are displayed as you select squares.



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).



To reset to the default bit order:
- Right-click the bus name, and uncheck **Enable Channel Order Selection**.

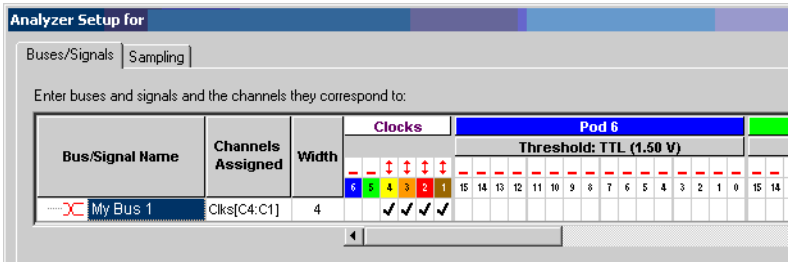
See Also
- To define buses and signals by importing netlist files (see page 100)
- To reorder bits by editing the Channels Assigned string (see page 101)
- Pod and Channel Naming Conventions (see page 333)

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 98) or To assign channels, selecting the bit order (see page 99)).

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.

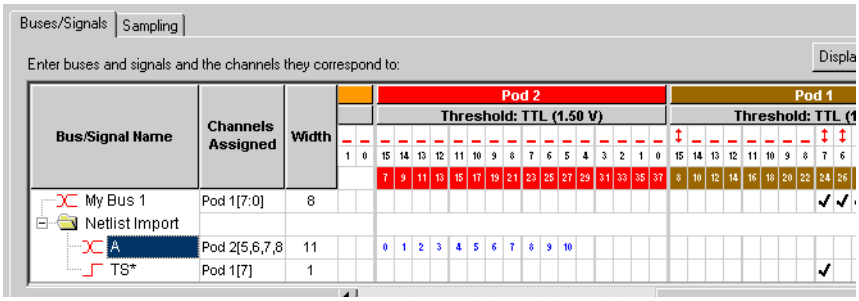
See Also
- To reorder bits by editing the Channels Assigned string (see page 101)
- Pod and Channel Naming Conventions (see page 333)

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.



Imported bus/signal definitions are placed in the "Netlist Import" folder.

CAUTION **The "Netlist Import" folder is deleted and re-created on each import. If you want to keep definitions from the "Netlist Import" folder, either rename the folder or move bus/signal definitions out of the folder before the next netlist import.**

Example Line from Netlist File:

Netlist files created by Electronic Design Automation (EDA) tools have lines in the following format:

NET '/Bus1(3)' J1-7

Where:

- **Bus1** = Four bit bus.
- **(3)** = Bit 3.
- **J1-7** = Connector J1, pin 7.

See Also

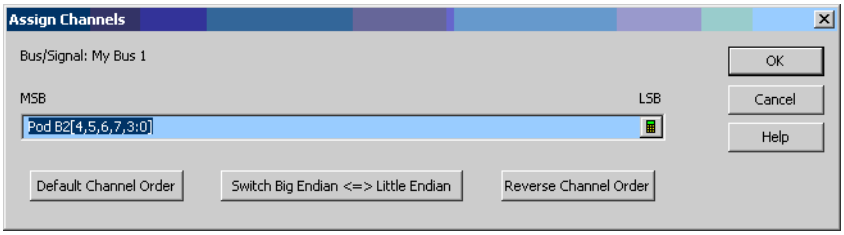
- To assign channels in the default bit order (see page 98)
- To assign channels, selecting the bit order (see page 99)
- To reorder bits by editing the Channels Assigned string (see page 101)
- Pod and Channel Naming Conventions (see page 333)

To reorder bits by editing the Channels Assigned string

You can change the order of the bits in a bus name (assigned in either the default order (see page 98) or a selected order (see page 99)) by editing the **Channels Assigned** text string.

1 In the Buses/Signals tab of the Analyzer Setup dialog, click the **Channels Assigned** to the bus name.

2 In the Assign Channels dialog, enter the appropriate order of bits in the bus.



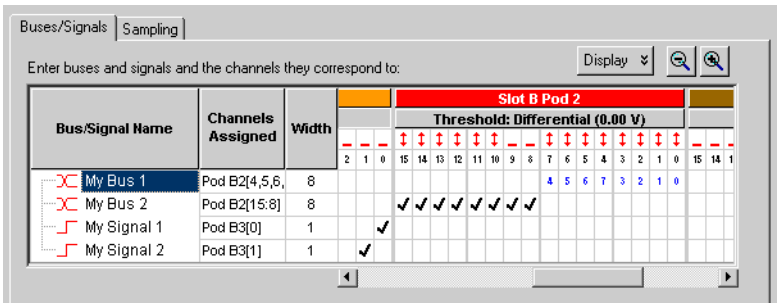
Example	Description
Pod B1[0]	Signal consisting of the first channel in the first pod of slot B.
Pod B1[15:0]	Bus consisting of all sixteen channels in Pod B1 in default order.
Pod B1[7:0], Pod B2[15:8]	Bus with seven channels from first pod in slot B followed by seven channels from second pod in slot 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 333) 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**.

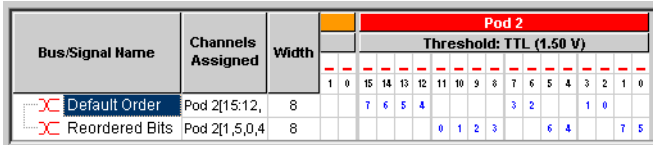
Channel numbers are displayed for reordered bits in the Bus/Signal Setup dialog.



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).



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 98)
- To assign channels, selecting the bit order (see page 99)
- Pod and Channel Naming Conventions (see page 333)

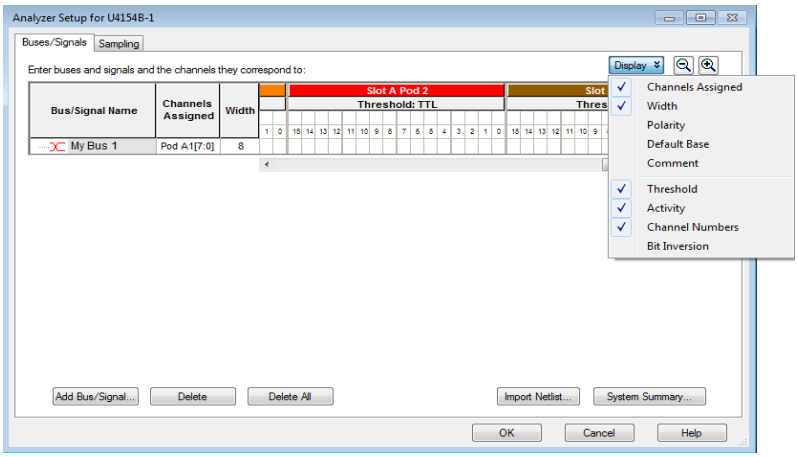
To set the default number base

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 218) and waveform (see page 209) 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.

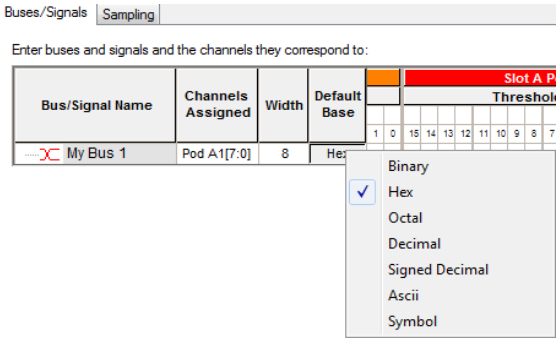
1 From the menu bar, select **Setup>(Logic Analyzer Module)>Bus/Signal...**.

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

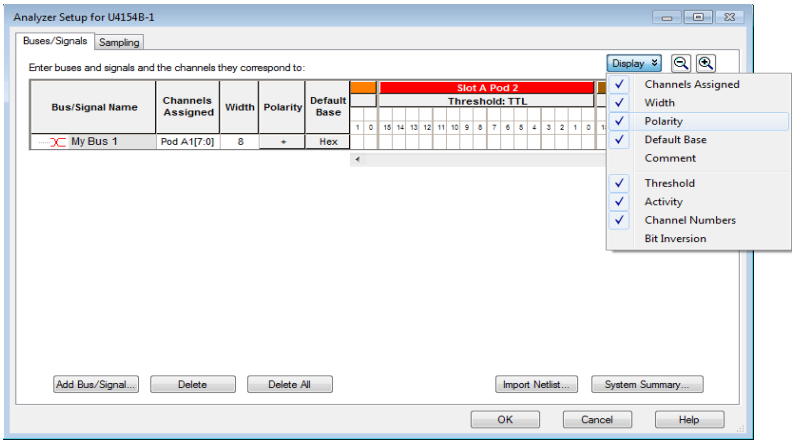
You can define buses and signals to display with negative or positive polarity. This affects the display of values and waveforms. When a bus or signal is set to negative polarity, an incoming high voltage will be shown with a low waveform and a logical value of 0. The polarity is reflected in all places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

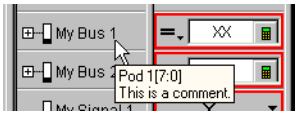
2 In the bus/signal setup dialog, select **Display**.

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 95) the new folder in the same way you would a bus/signal name.

See Also
- To alias a bus/signal name (see page 105)

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 105) 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 95), however, the new name will also be applied to the original name.

See Also
- To add a folder (see page 105)

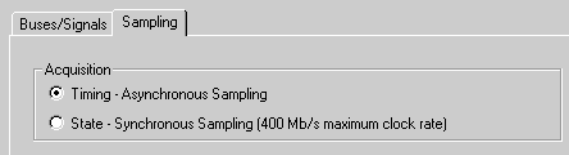
To sort bus/signal names

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 105)

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 106)
  - To select the timing acquisition mode (see page 107)
  - To select the timing sampling option (see page 107)
  - To set the timing mode sampling period (see page 108)
- Selecting the State Mode (Synchronous Sampling) (see page 108)
  - To select the state acquisition mode (see page 108)
  - To select the state sampling option (see page 108)
  - Selecting the State Sampling Clock Mode (see page 109)
  - To set up the state sampling clock (see page 113)
  - To automatically adjust state sampling positions and threshold voltages (see page 114)
  - To manually adjust state sampling positions (see page 114)
- In Either Timing Mode or State Mode (see page 115)
  - To specify the trigger position (see page 115)
  - To set acquisition memory depth (see page 115)
- Using Timing Zoom (see page 116)
  - To turn timing zoom on or off (see page 117)
  - To specify the timing zoom sample period (on some logic analyzers) (see page 117)
  - To specify the timing zoom trigger position (see page 117)
  - To align timing zoom in a split analyzer (see page 117)

See Also
- Memory Depth and Channel Count Trade-offs (see page 335)

Selecting the Timing Mode (Asynchronous Sampling)

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

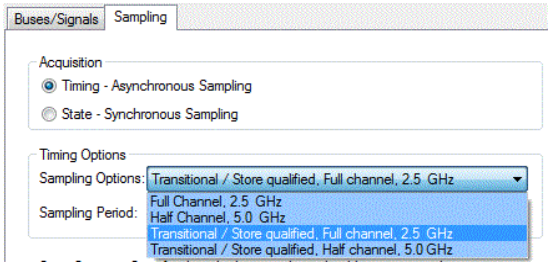
- To select the timing acquisition mode (see page 107)
- To select the timing sampling option (see page 107)
- To set the timing mode sampling period (see page 108)

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 372).

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
Full Channel Timing Mode (see page 492)	Default. All channels are available.
Half Channel Timing Mode (see page 492)	Uses one pod from each pod pair.
Transitional / Store Qualified Timing Mode (see page 492)	Provides maximum duration of acquisition because data is only stored when a change from the last value is detected. See transitional timing (see page 337).

See Also

- Logic Analyzer Notes, Timing Mode Sampling Options/Period (see page 492)

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 372).

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 492)

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.

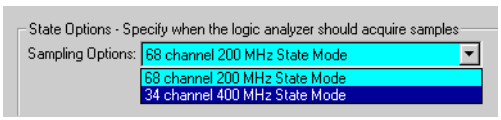
- To select the state acquisition mode (see page 108)
- To select the state sampling option (see page 108)
- Selecting the State Sampling Clock Mode (see page 109)
- To set up the state sampling clock (see page 113)
- To automatically adjust state sampling positions and threshold voltages (see page 114)
- To manually adjust state sampling positions (see page 114)

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 372).

2 In the Acquisition area of the Sampling setup dialog, select the **State - Synchronous Sampling** option.

To select the state sampling option



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.

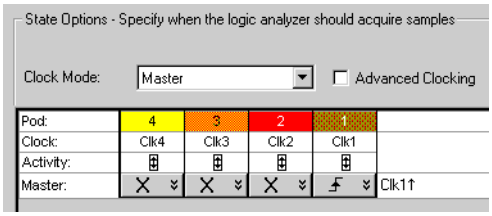
3 Select the sampling option you prefer.

Mode	Description
General State Mode (see page 492)	Default.
Turbo State Mode (see page 492)	Faster state mode sampling speeds are supported, but triggering and clocking are restricted.

See Also
- Logic Analyzer Notes, State Mode Sampling Options (see page 492)

Selecting the State Sampling Clock Mode

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



The state sampling clock mode specifies how the clock inputs are used for sampling. For the U4154A/B module, there are two state sampling clock modes to choose from:

- Master Only (see page 109) – all pods sampled on one master clock.
- Dual Sample (see page 110) – one pod of pair sampled on one master clock but with different delays.

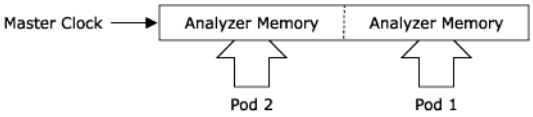
For instructions on setting up these state sampling clock modes, see:

- "To set up the master only sampling clock mode" on page 111
- "To set up the dual sample sampling clock mode" on page 112

See Also
- "To set up the state sampling clock" on page 113
- "To set up advanced clocking" on page 114
- "Pod and Channel Naming Conventions" on page 333

Master Only Sampling Clock Mode

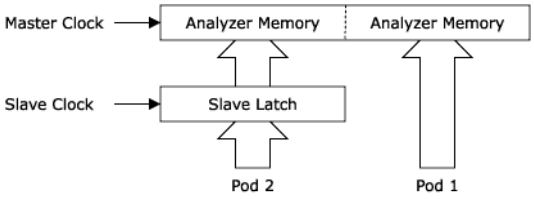
In the Master only state sampling clock mode, there is one sampling clock signal. When a clock edge occurs, data is captured and saved into one sample of logic analyzer memory.



See Also
- To set up the master only sampling clock mode (see page 111)

Master/Slave Sampling Clock Mode

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

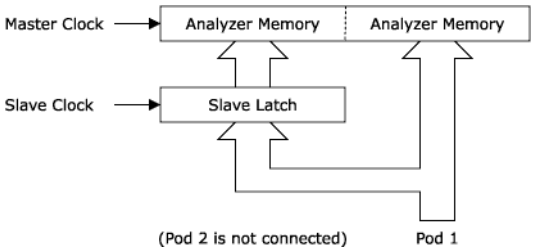


When the slave clock occurs, data captured on the pods that use the slave clock is saved in a slave latch. Then, when the master clock occurs, data captured on the pods that use the master clock, as well as the slave latch data, are saved into logic analyzer memory. If multiple slave clocks occur before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

See Also
- To set up the master/slave sampling clock mode (see page 111)

Demultiplex Sampling Clock Mode

In the Demultiplex state sampling clock mode, you can demultiplex data being probed by one pod into the logic analyzer memory that is normally used for two pods. Demultiplex mode uses the master and slave clocks to demultiplex the data.

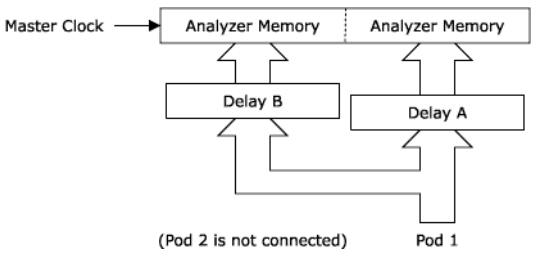


When the slave clock occurs, data captured on the pod is saved into the slave latch for the other pod in the pod pair. Then, when the master clock occurs, data captured on the pod, as well as the slave latch data, are saved in logic analyzer memory. As with master/slave mode, if multiple slave clocks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

See Also
- To set up the demultiplex sampling clock mode (see page 111)

Dual Sample Sampling Clock Mode

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.



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 Thresholds and Sample Positions dialog.

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 112

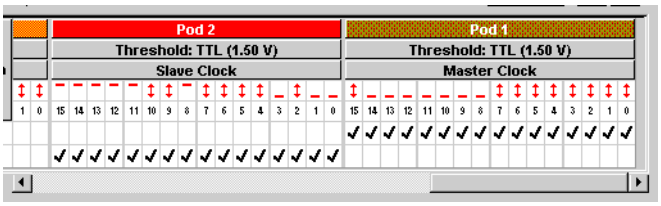
To set up the master only sampling clock mode

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 113)).

See Also • Master Only Sampling Clock Mode (see page 109)

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 In the State Options, change Clock Mode to **Master/Slave/Demux**.
4 Set up your master and slave clocks (see To set up the state sampling clock (see page 113)).
5 Select the **Buses/Signals** tab.
6 For each pod, click the clock button under the pod heading and choose either **Master Clock** or **Slave Clock**.

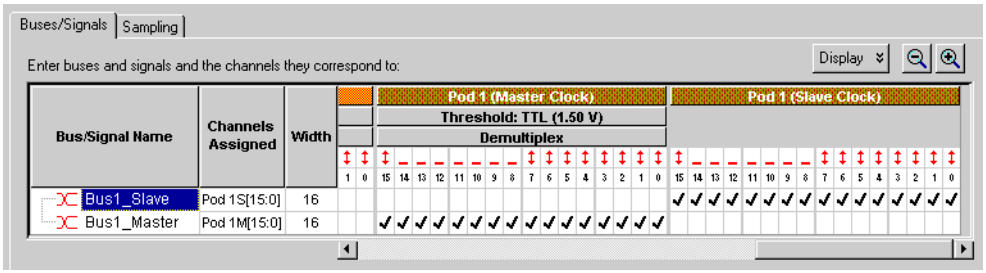


See Also • Master/Slave Sampling Clock Mode (see page 109)

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 In the State Options, change Clock Mode to **Master/Slave/Demux**.
4 Set up your master and slave clocks (see To set up the state sampling clock (see page 113)).
5 Select the **Buses/Signals** tab.
6 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).

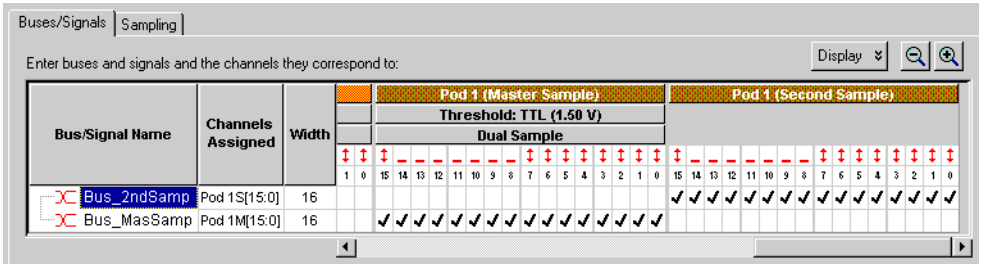
Note that you can adjust sampling positions relative to the master clock and to the slave clock by assigning master and slave bus/signal names. (Note the "M" and "S" suffixes on pods in the **Channels Assigned** column to indicate "master clock" and "slave clock".) For more information on adjusting sampling positions, see To automatically adjust state sampling positions and threshold voltages (see page 114) or To manually adjust state sampling positions (see page 114).

See Also
- Demultiplex Sampling Clock Mode (see page 110)

To set up the dual sample sampling clock mode

NOTE To capture dual sample data, use only one pod of a pod pair.

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 **Dual Sample**.
4 Set up your master clock (see To set up the state sampling clock (see page 113)).
5 Select the **Buses/Signals** tab.
6 Click the clock button under the pod heading, and choose **Dual Sample**.



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

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

7 Assign bus/signal names to the channels in the master sample and to the channels the second sample.

Bus/signal names assigned to the master sample can have different sampling positions than the bus/signal names assigned to the second sample.

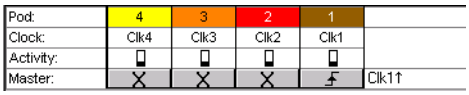
8 Adjust the state sampling positions differently for the bus/signals associated with the master sample and the buses/signals associated with the second sample (see To automatically adjust state sampling positions and threshold voltages (see page 114) or To manually adjust state sampling positions (see page 114)).

Note the "M" and "S" suffixes on pods in the **Channels Assigned** column to indicate "master sample" and "second sample".

See Also
- "Dual Sample Sampling Clock Mode" on page 110

To set up the state sampling clock

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



In the picture above, the clocks refer to the clock signal lines on pods 1 through 4. 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.

1 Attach logic analyzer pods to your device under test. Clock signals must be connected to the clock lines on pods 1 through 4.

2 From the menu bar, select **Setup>(Logic Analyzer Module)>Timing/State (Sampling)...**.

3 Select **State - Synchronous Sampling**. The State Options area becomes active.

4 Select the state clock mode (see page 109). Most measurements use only the master clock.

5 Set up a clock description to match the clock(s) in your device under test.

Symbols	Description
X	Don't care. Clock line not used in this clock.
↑	Rising edge.
↓	Falling edge.
⇕	Both edges.
1	Qualifier - high.
0	Qualifier - low.

A clock description must have at least one edge.

See Also
- "Selecting the State Sampling Clock Mode" on page 109
- "To automatically adjust state sampling positions and threshold voltages" on page 114
- "To manually adjust state sampling positions" on page 114
- "To set up advanced clocking" on page 114

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.

1 From the menu bar, select **Setup>(Logic Analyzer Module)>State/Timing (Sampling)...**.
2 Select **State** mode. The state options become selectable.
3 Next to the clock mode, select **Advanced Clocking**. The clock controls are replaced by a button.
4 Select **Master Clock...** or **Slave Clock...** as appropriate.
5 In the Advanced Clocking Setup Dialog (see page 391), choose settings as appropriate.
Clock channels can be used both as primary clocks and as clock qualifiers.
6 Click **OK** to close the dialog. The clock description in the Analyzer Setup window updates.

NOTE If you un-check advanced clocking, all qualifiers are erased.

See Also
- Pod and Channel Naming Conventions (see page 333)

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 U4154A/B Logic Analyzer" on page 508 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 176)).

See Also
- Understanding State Mode Sampling Positions (see page 339)
- To automatically adjust state sampling positions and threshold voltages (see page 114)

In Either Timing Mode or State Mode

- To specify the trigger position (see page 115)
- To set acquisition memory depth (see page 115)

To specify the trigger position

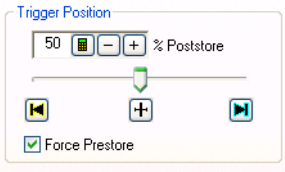
The trigger position specifies the amount of trace memory used for samples captured after the trigger. For example, when 10% poststore is selected, 90% of trace memory is used for samples captured before the trigger. When 90% poststore is selected, 10 % of trace memory is used for samples captured before 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 372).

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 341)
- To set the acquisition memory depth (see page 115)

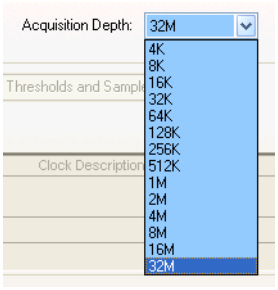
To set acquisition memory depth

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 335)
- Logic Analyzer Notes, Channels and Memory Depth (see page 492)
- Understanding Logic Analyzer Triggering, The Conveyor Belt Analogy (see page 341)
- To specify the trigger position (see page 115)

Using Timing Zoom

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

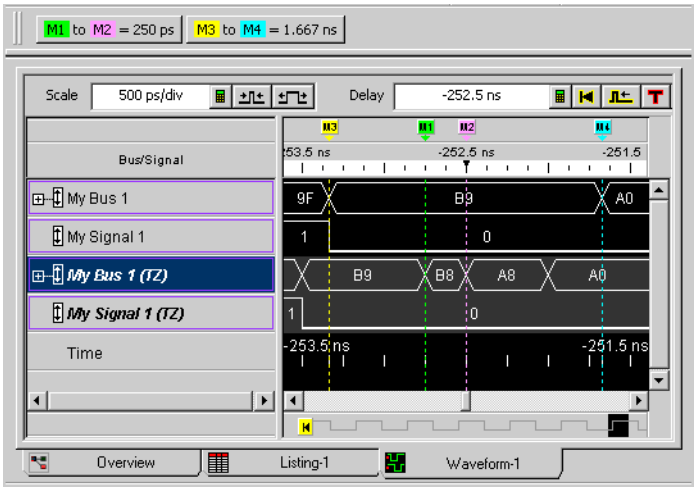
The timing zoom settings are accessed through the TimingZoom box in the Sampling tab.

- To turn timing zoom on or off (see page 117)
- To specify the timing zoom sample period (on some logic analyzers) (see page 117)
- To specify the timing zoom trigger position (see page 117)
- To align timing zoom in a split analyzer (see page 117)

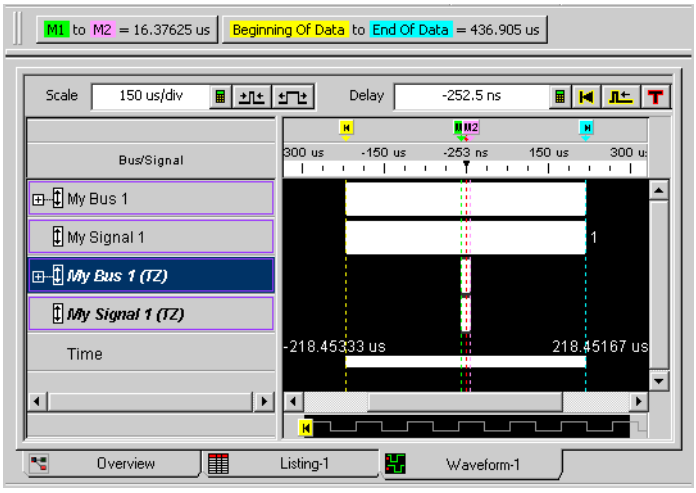
What is Timing Zoom?

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 492)

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 492)), 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 438), select the **Sampling Period**.

See Also
- Logic Analyzer Notes, Timing Zoom (see page 492)

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 438), 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 492)

To align timing zoom in a split analyzer

On some older logic analyzer modules, if you have split the analyzer, you need to specify which analyzer's trigger timing zoom should be aligned with.

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 438), select the logic analyzer to **Align Trigger With**.

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 user-defined symbols (see page 118)
- To load symbols from a file (see page 119)
- To run the symbol reader outside the application (see page 120)
- To create an ASCII symbol file (see page 121)
- To change symbol reader options (see page 121)

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).

See Also

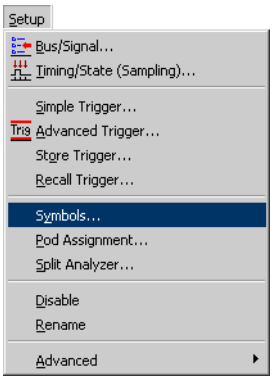
- To enter symbolic bus/signal values (see page 123)

To create 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...**.

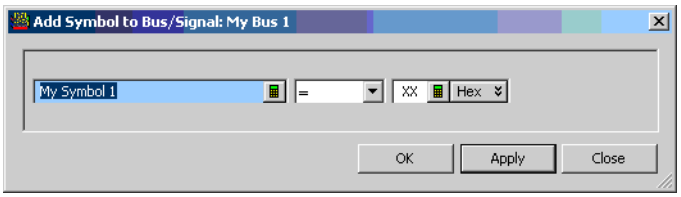


2 In the Symbols dialog (see page 433), select the bus or signal for which the new symbol should be displayed.

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.

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 218) for the bus/signal to Symbols.

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**.

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 176). 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 220)

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 475)).
   - Generate an Keysight Symbol Reader ".sym" file by running the symbol reader outside of the *Keysight Logic Analyzer* application (see page 120); 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 121).

2 From the *Keysight Logic Analyzer* application's main menu bar, choose **Setup>(Logic Analyzer Module)>Symbols...**.

3 In the Symbols dialog (see page 433), 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 433), 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 433), 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 475)

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 119)
- To change symbol reader options (see page 121)

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 476)

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

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 419), 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 419)

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\`

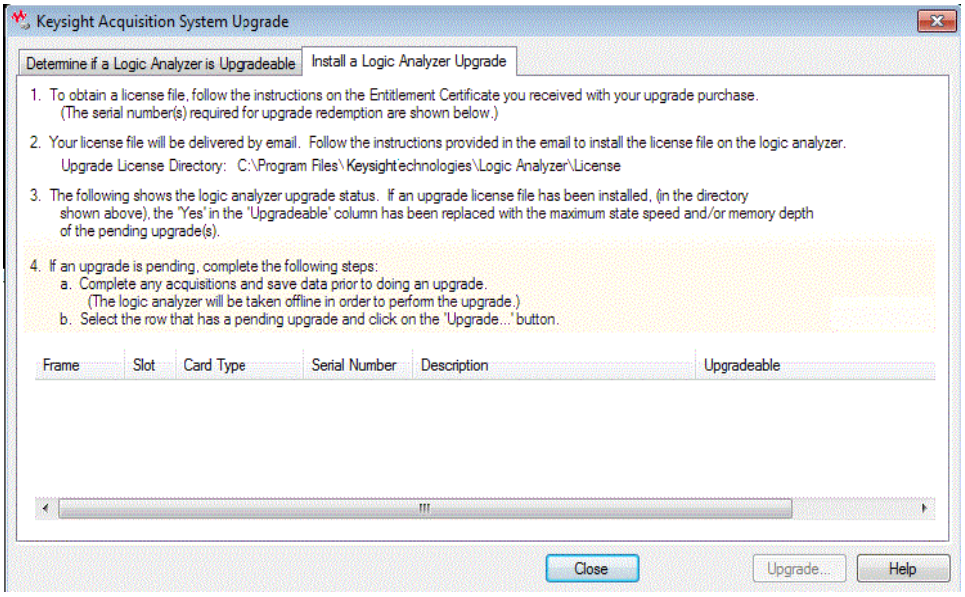
If upgrade options were ordered for several cards at the same time, there will be one license file for all submitted serial numbers.

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**.

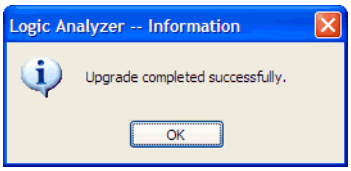
3 In the Keysight Logic Analyzer Upgrade dialog's "Install a Logic Analyzer Upgrade" tab, select the card that has an upgrade pending.



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 73) and set up the logic analyzer (see page 87) (by defining buses and signals (see page 93) and choosing the sampling mode (see page 106)), 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 440)*.

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 187) and save it (along with the logic analyzer setup).

- Setting Up Quick (Draw Box) Triggers (see page 129)
  - To set a Quick Trigger in the Waveform window (see page 129)
  - To set a Quick Trigger in the Listing window (see page 130)
  - To set a Quick Trigger in the Source window (see page 131)
- Specifying Simple Triggers (see page 132)
  - To specify bus patterns in a simple trigger (see page 133)
  - To specify signal levels in a simple trigger (see page 134)
  - To set a bus/signal edge in a simple trigger (see page 134)
- Specifying Advanced Triggers (see page 138)
  - To replace or insert trigger functions into trigger sequence steps (see page 142)
  - To specify bus/signal patterns (see page 142)
  - To set a bus/signal edge in an advanced trigger (see page 144)
  - To specify packet events (in "Find a packet" trigger function) (see page 148)
  - To specify a trigger sequence step's goto or trigger action (see page 150)
  - To specify default storage (see page 151)
  - To insert or delete events (see page 152)
  - To negate events (see page 154)
  - To change the evaluation order of AND/OR'ed events (see page 155)
  - To choose between a duration or occurrence count for events (timing mode) (see page 156)
  - To insert or delete actions (in a trigger sequence step) (see page 156)
  - To cut, copy, and paste trigger sequence steps (see page 159)
  - To delete trigger sequence steps (see page 160)
  - To show a trigger sequence step as Advanced If/Then trigger functions (see page 160)
  - To convert a trigger sequence step to Advanced If/Then trigger functions (see page 160)

  - To display or hide "If" clause comments (see page 161)
  - To clear the trigger sequence (see page 163)
- **Triggering From, and Sending Triggers To, Other Modules/Instruments** (see page 164)
- **Storing and Recalling Triggers** (see page 172)
  - To store a trigger (see page 172)
  - To recall a trigger (see page 172)
  - To set the trigger history depth (see page 173)
- **Running/Stopping Measurements** (see page 174)
- **Saving Captured Data (and Logic Analyzer Setups)** (see page 176)
  - To save a configuration file (see page 176)
  - To export data to standard CSV format files (see page 177)
  - To export data to module CSV format files (see page 179)
  - To export data to module binary (ALB) format files (see page 181)
  - To export data to 16700 ASCII format files (see page 183)
- **Extending Capture Capability with COM/DCOM (see page 186)**

See Also

- Probing the Device Under Test (see page 73)
- Setting Up the Logic Analyzer (see page 87)
- Defining Buses and Signals (see page 93)
- Choosing the Sampling Mode (see page 106)
- Analyzing the Captured Data (see page 187)

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 172) at any time.

- To set a Quick Trigger in the Waveform window (see page 129)
- To set a Quick Trigger in the Listing window (see page 130)
- To set a Quick Trigger in the Source window (see page 131)

To set a Quick Trigger in the Waveform window

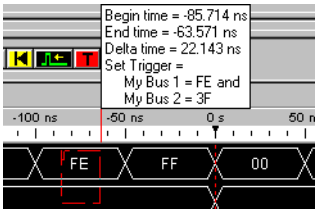
In the Waveform window, you can quickly set up a simple trigger by drawing a rectangle with the mouse.

1 Make sure the Waveform window's Fast Zoom In (see page 207) 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:

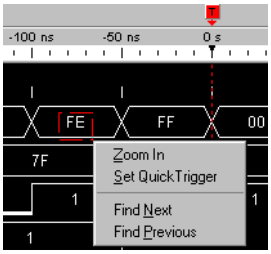
a If drawing starts on a bus, none of its expanded signals can be included.

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.

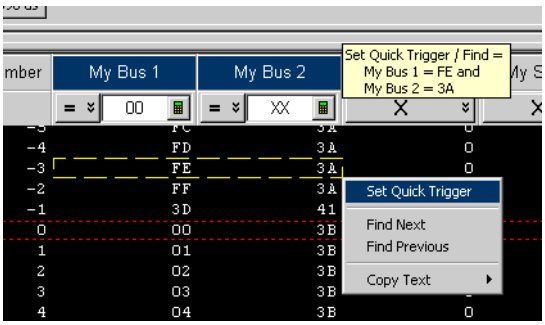
1 Using the mouse, point to the sample that you want to use as the quick trigger.

2 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.

3 Select **Set Quick Trigger**.



General Guidelines

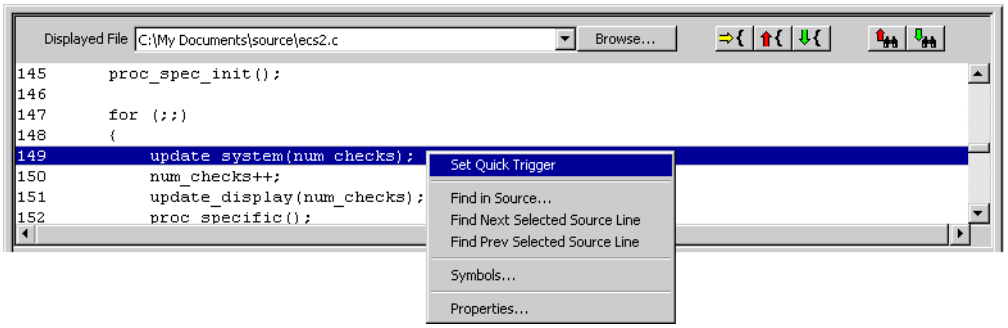
- Any bus/signals with overlapping bits are not included within the trigger specification.

  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

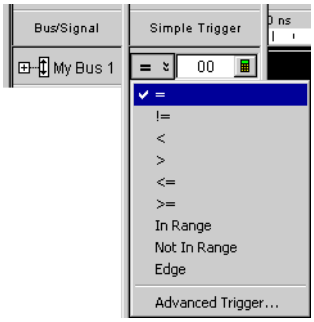
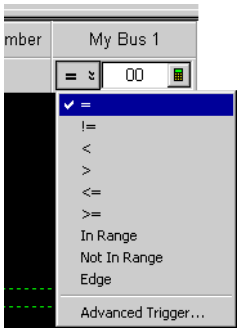
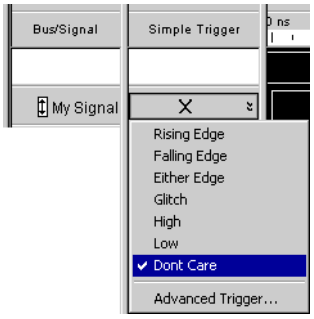
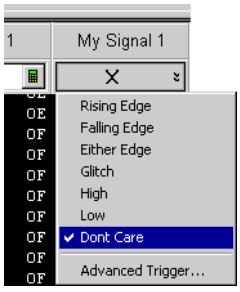
1 In the Source window's source pane, right-click the source line you want to set a Quick Trigger on, and choose **Set Quick Trigger**.



See Also
- To change the "Set Quick Trigger" alignment (see page 258)
- Running/Stopping Measurements (see page 174)
- Viewing Source Code Associated with Captured Data (see page 253)

Specifying Simple Triggers

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

	Waveform Display Window	Listing/Compare/Source Display Window
Bus Trigger		
Signal Trigger		

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

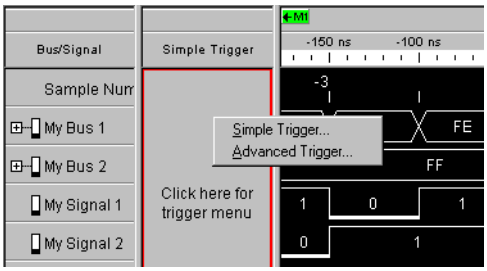
Signals are compared to edges (**Rising Edge**, **Falling Edge**, **Either Edge**, or a logic level pattern (**High**, **Low**). Edge options are available in timing mode only. For 16850 series and U4154A/B Logic Analyzer modules, edge options are also available in the state mode.

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 138). 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.



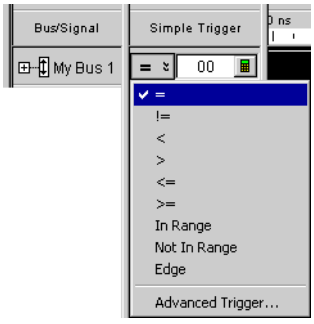
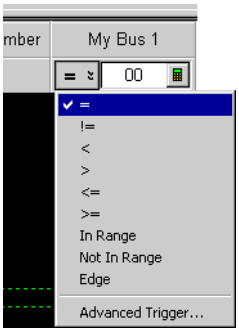
- To specify bus patterns in a simple trigger (see page 133)
- To specify signal levels in a simple trigger (see page 134)
- To set a bus/signal edge in a simple trigger (see page 134)

See Also

- Setting Up Quick Triggers (see page 129)
- Specifying Advanced Triggers (see page 138)
- To store a trigger (see page 172)
- To recall a trigger (see page 417)

To specify bus patterns in a simple trigger

When specifying simple triggers (see page 132), you can specify bus patterns to trigger on.

	Waveform Display Window	Listing/Compare/Source Display Window
Bus Trigger		

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)

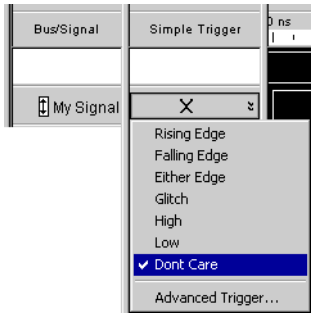
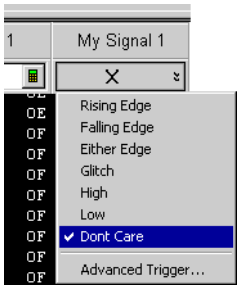
- In Range
- Not In Range

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 XX, enter the bus pattern value to compare.

To specify signal levels in a simple trigger

When specifying simple triggers (see page 132), you can specify signal levels to trigger on.

	Waveform Display Window	Listing/Compare/Source Display Window
Signal Trigger		

To specify a signal level

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

See Also
- To specify bus patterns in a simple trigger (see page 133)

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.

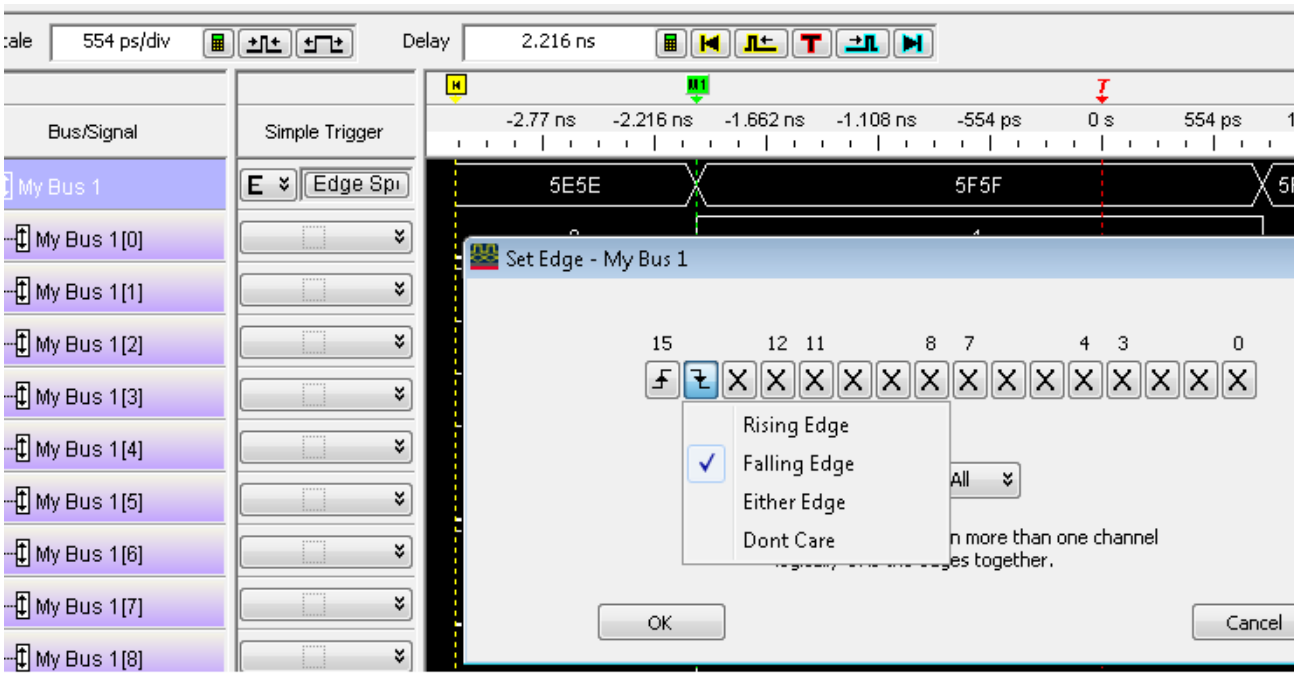


Figure 1 Figure - Trigger on edge set from Waveform viewer for a bus

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".

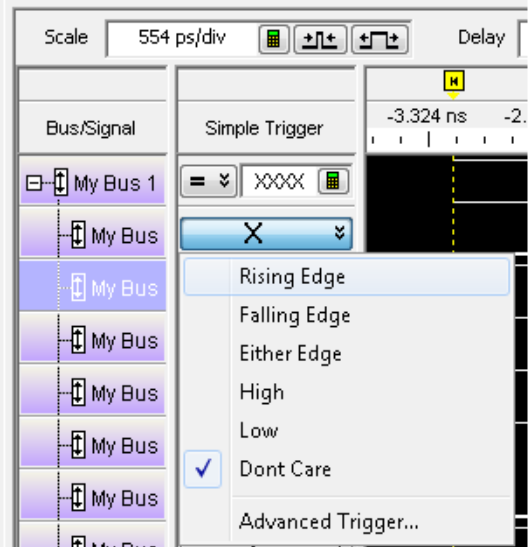


Figure 2 Trigger on edge set from Waveform viewer for a signal

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.

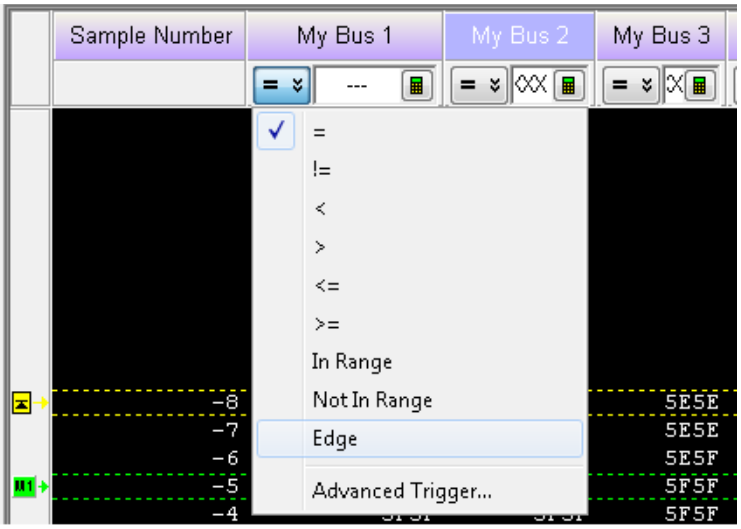


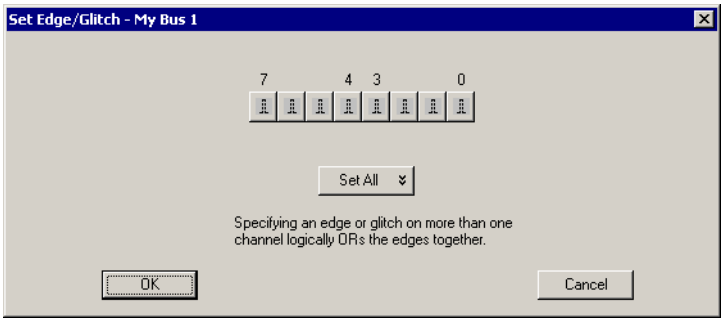
Figure 3 Trigger on edge set from Listing viewer for a bus

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 X edge/level button; then, choose from:

- **Rising Edge**
- **Falling Edge**
- **Either Edge**

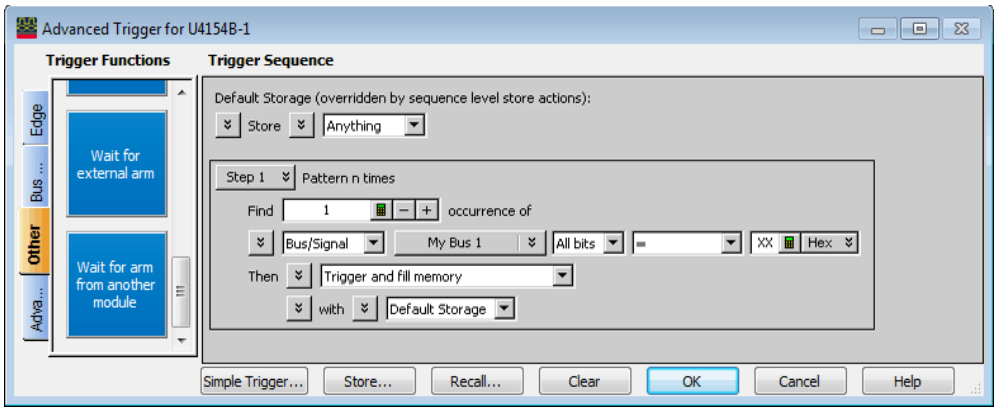
See Also

- To specify signal levels in a simple trigger (see page 134)
- Understanding Logic Analyzer Triggering, Edges (see page 341)

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 440)* 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 139)* 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 139)* (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 142)
- To specify bus/signal patterns (see page 142)
- To set a bus/signal edge in an advanced trigger (see page 144)
- To specify packet events (in "Find a packet" trigger function) (see page 148)
- To specify a trigger sequence step's goto or trigger action (see page 150)
- To specify default storage (see page 151)
- To insert or delete events (see page 152)
  - To insert a timer event (see page 153) (see also Using Timers (see page 139))
  - To insert a counter event (see page 153) (see also Using Counters (see page 140))
  - To insert a flag event (see page 154) (see also Using Flags (see page 141))
  - To insert an "Arm in from" event (see page 154)
- To negate events (see page 154)
- To change the evaluation order of AND/OR'ed events (see page 155)

- To choose between a duration or occurrence count for events (timing mode) (see page 156)
- To insert or delete actions (in a trigger sequence step) (see page 156)
  - To insert a timer action (see page 157) (see also Using Timers (see page 139))
  - To insert a counter action (see page 157) (see also Using Counters (see page 140))
  - To insert a reset occurrence counter action (see page 157)
  - To insert a flag action (see page 158) (see also Using Flags (see page 141))
  - To insert a storage control action (see page 159)
- To cut, copy, and paste trigger sequence steps (see page 159)
- To delete trigger sequence steps (see page 160)
- To show a trigger sequence step as Advanced If/Then trigger functions (see page 160)
- To convert a trigger sequence step to Advanced If/Then trigger functions (see page 160)
- To display or hide "If" clause comments (see page 161)
- To clear the trigger sequence (see page 163)

See Also
- Understanding Logic Analyzer Triggering (see page 341)
- State Mode Trigger Functions (see page 452)
- Timing Mode Trigger Functions (see page 440)
- Specifying Simple Triggers (see page 132)

Reading Event and Action Statements

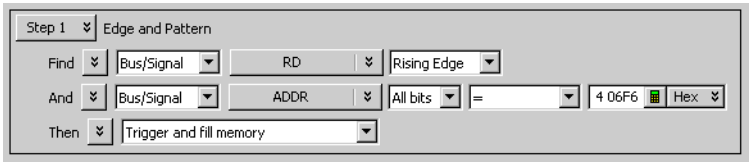
Event and action statements in trigger sequence steps read from left-to-right like:

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 138)
- Understanding Logic Analyzer Triggering, Sequence Steps (see page 343)

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**.

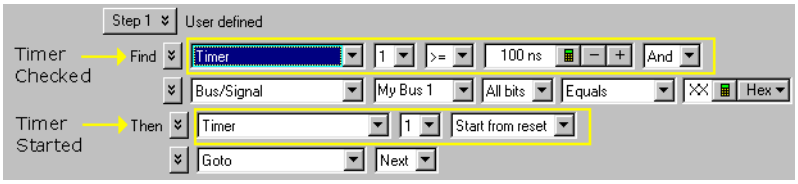
Timer considerations:

- 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 528).
- 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)	*General State Mode*	# pod pairs not reserved for time tag storage
	Turbo State Mode	0
Timing mode (asynchronous sampling)	all	# pod pairs not reserved for time tag storage

  Refer to your logic analyzer characteristics (see page 528) 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 334).
- 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 157)
- To insert a timer event (see page 153)
- Understanding Logic Analyzer Triggering, Timers (see page 346)

Using Counters

Counters are available in both Event and Action statements, and like other events, they evaluate to either true or false. Use counters to create a user-defined delay, or to create a standard against which valid data duration is evaluated. Once configured, a counter persists throughout all the steps of the trigger sequence.

Counter considerations:

- Maximum counters available is 2.
- When using counters in the transitional timing mode, one counter is used internally so only one counter is available in a sequence step.
- Once a counter is configured, you can reuse the counter by selecting its identification number. Each use of the counter must check it for the same value.

NOTE The logic analyzer also has occurrence counters, and a **reset occurrence counter** action. Occurrence counters only exist within steps that contain the "occurs" phrase and are not affected by the other counter actions described on this page.

See Also
- To insert a counter action (see page 157)
- To insert a counter event (see page 153)
- Understanding Logic Analyzer Triggering, Counters (see page 345)

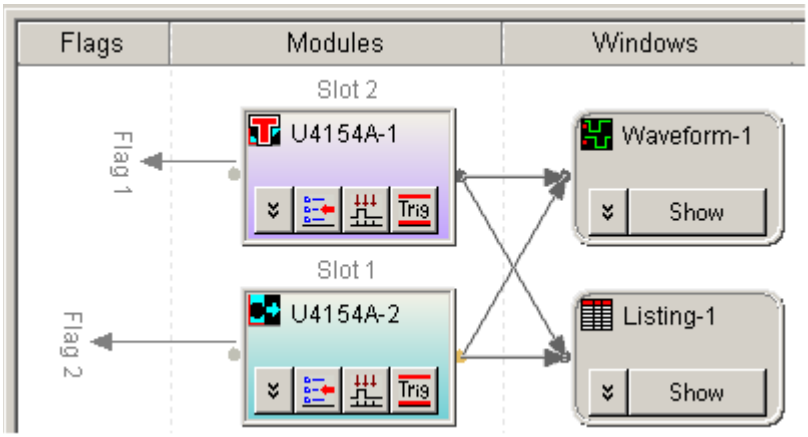
Using Flags

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.

NOTE Flag actions are not available in *Turbo State Mode*; however, you can still check flags with flag events.

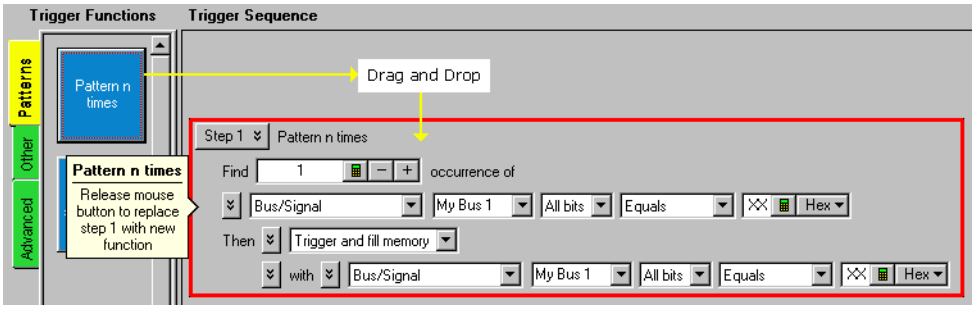
Flags can also be used to drive the logic analysis system's Port Out signal.

See Also
- To insert a flag action (see page 158)
- To insert a flag event (see page 154)
- Understanding Logic Analyzer Triggering, Flags (see page 345)

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 440) 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.

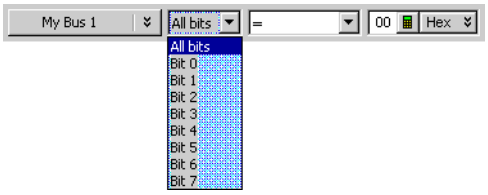
See Also
- Understanding Logic Analyzer Triggering, Sequence Steps (see page 343)
- To delete trigger sequence steps (see page 160)
- State Mode Trigger Functions (see page 452)
- Timing Mode Trigger Functions (see page 440)

To specify bus/signal patterns

1 In the Advanced Trigger dialog, select the bus or signal.

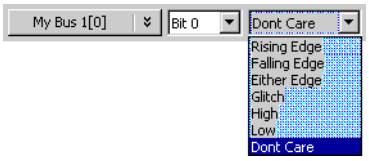
Clicking lets you select from recently used bus/signal names. Clicking elsewhere on a bus/signal name button opens a Select dialog for selecting a different name.

2 If a bus has been selected, either select **All bits** on the bus or select an individual bit.



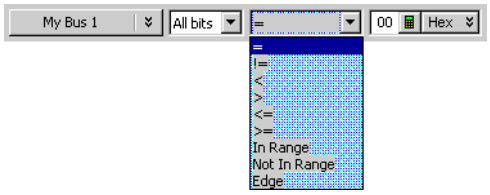
3 Specify the bus/signal value:

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 419) to specify the pattern values.

See Also
- To specify default storage (see page 151)
- To insert or delete events (see page 152)
- Understanding Logic Analyzer Triggering, Ranges (see page 345)

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.

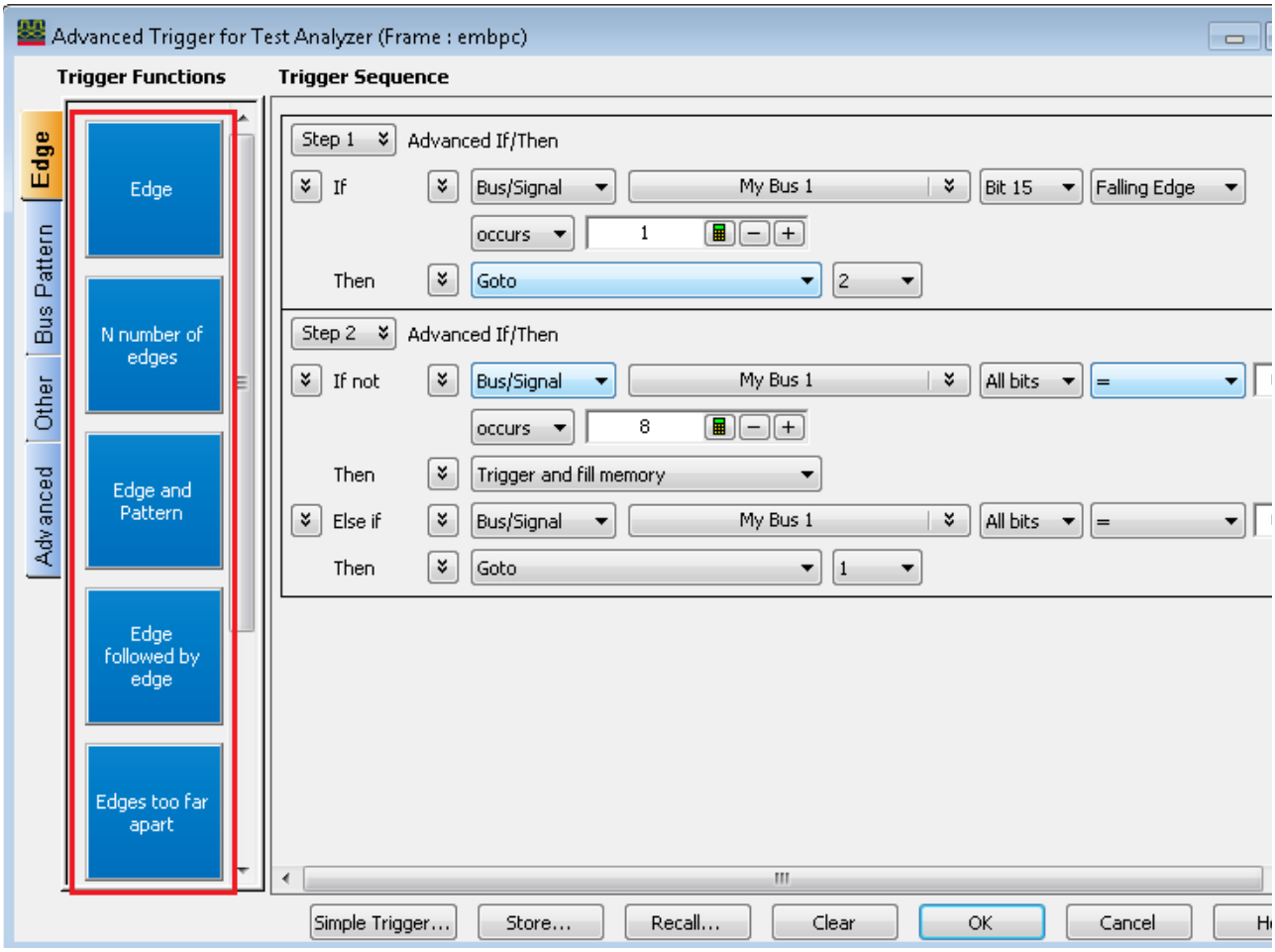
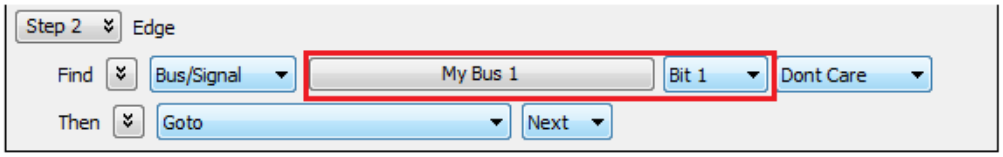


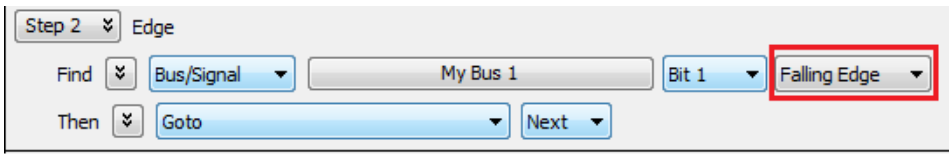
Figure 4 Trigger on edge options for the timing mode

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.

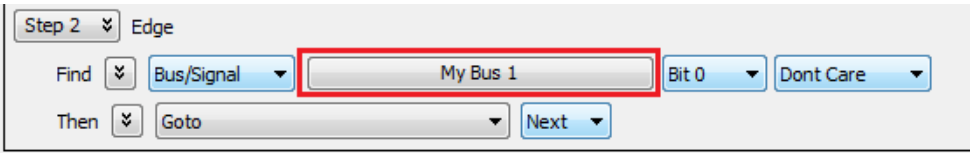


2 Select the edge value (**Rising Edge**, **Falling Edge**, or **Either Edge**) for the signal.

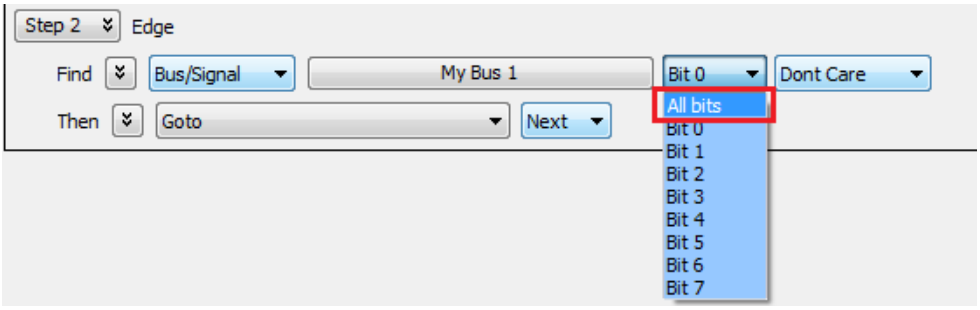


To specify bus/signal edges in advanced trigger

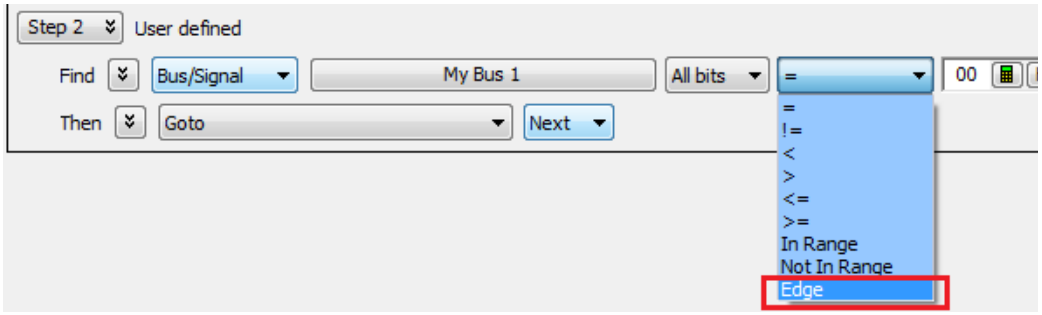
1 In the Advanced Trigger dialog, select the bus on which you want to set trigger on edge.



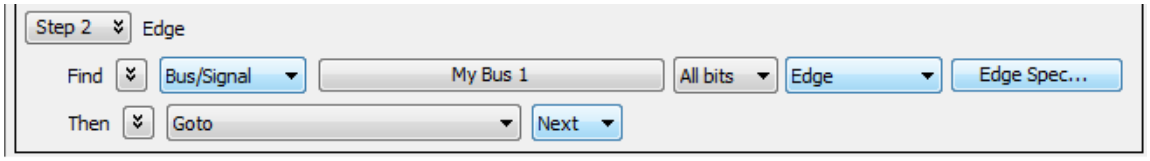
2 Select the **All bits** option for the bus.



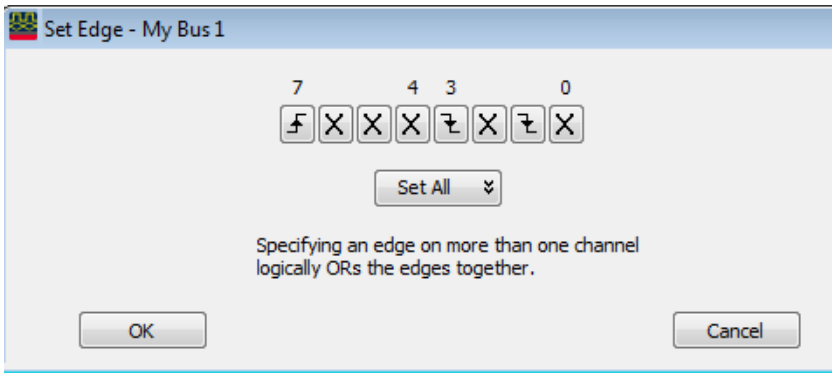
3 Select the **Edge** option.



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.

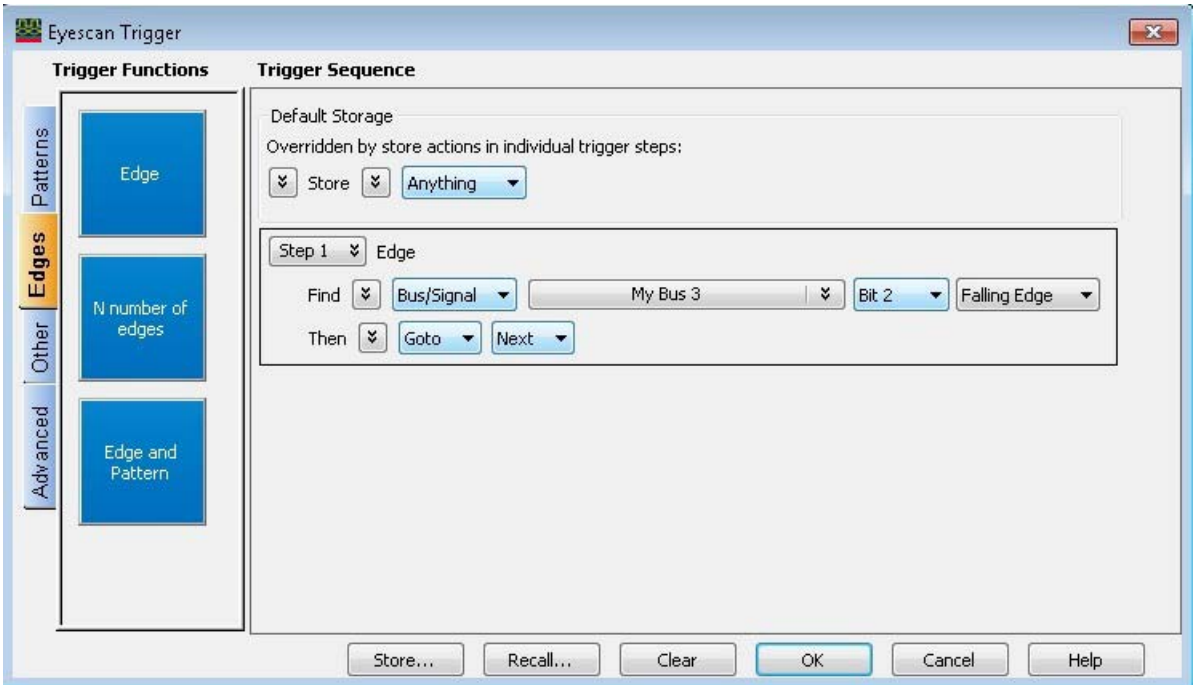


Figure 5 Trigger on edge in an eyescan trigger

See Also

- To specify default storage (see page 151)
- To insert or delete events (see page 152)
- Understanding Logic Analyzer Triggering, Edges (see page 345)

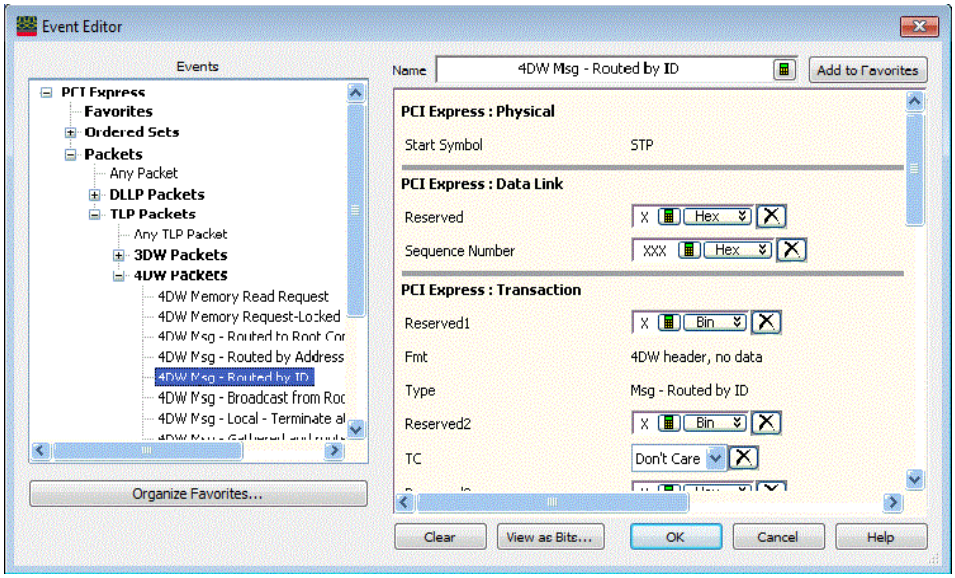
To specify packet events (in "Find a packet" trigger function)

1 In the Advanced Trigger dialog, replace or insert the Find a packet (see page 458) trigger function into the trigger sequence (see To replace or insert trigger functions into trigger sequence steps (see page 142)).
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 398), 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 401), 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 148)

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:

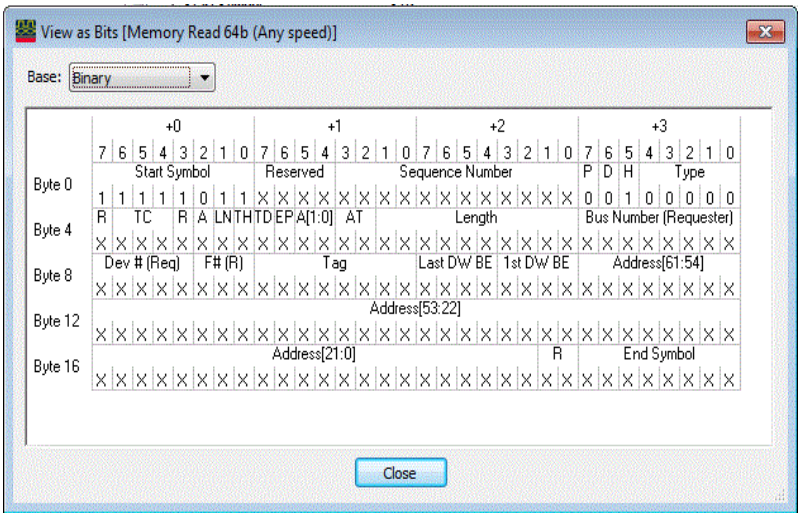
- To view a packet event as bits (see page 149)
- To save favorite packet events (see page 149)
- To organize favorite packet events (see page 149)

See Also
- Find a packet (see page 458) trigger function

To view a packet event as bits

1 While specifying packet events using the Event Editor Dialog (see page 401), 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 148)

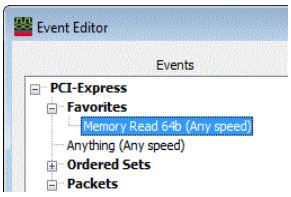
To save favorite packet events

While specifying packet events using the Event Editor Dialog (see page 401), 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

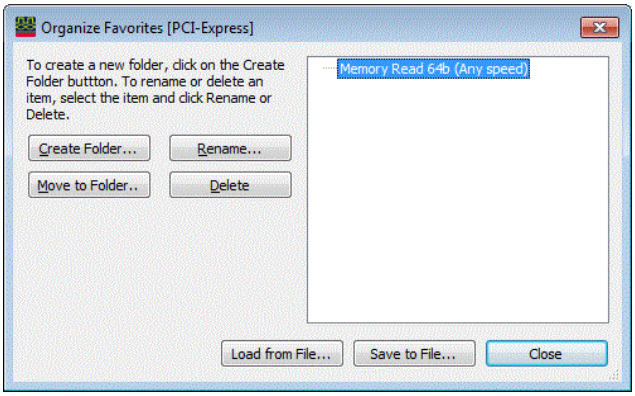
- To organize favorite packet events (see page 149)
- To specify packet events (in "Find a packet" trigger function) (see page 148)

To organize favorite packet events

While specifying packet events using the Event Editor Dialog (see page 401), you can organize saved packet event favorites.

1 Click **Organize Favorites...**.

2 In the Organize Favorites dialog, you can:



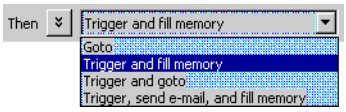
- Create folders, move selected events to folders, and rename or delete selected events.
- Save favorites to a file, or load saved favorites from a file.

3 When you are done organizing packet event favorites, click **Close**.

See Also
- To save favorite packet events (see page 149)
- To specify packet events (in "Find a packet" trigger function) (see page 148)

To specify a trigger sequence step's goto or trigger action

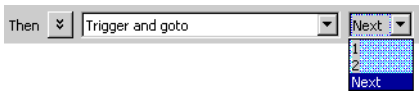
1 In the Advanced Trigger dialog, within a sequence step, select the desired trigger action:



You can select:

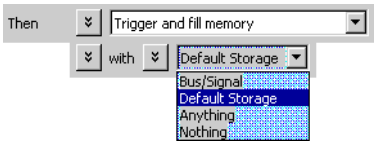
- **Goto**—To go to another trigger sequence step.

- **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 399) 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 152)
- To specify default storage (see page 151)
- To insert or delete actions (in a trigger sequence step) (see page 156)

To specify default storage

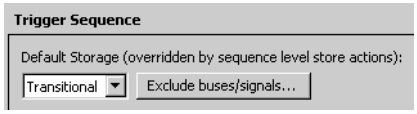
Storage qualifiers are used to specify which samples (captured from the device under test) are stored in logic analyzer memory. By storing only the samples you are interested in, you can make better use of the available memory and capture activity for a greater amount of time.

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.

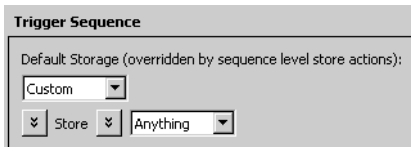
The default storage qualifier is available in state sampling mode and in the store qualified (transitional) timing mode.

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.

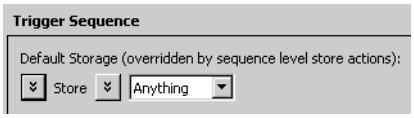


If you want to exclude certain bus/signal transitions from being stored, click **Exclude buses/signals...**, and specify which buses/signals should be excluded.
- When **Custom** is selected, edit or insert events that should be stored (or not stored) in logic analyzer memory.



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 152)
- To negate events (see page 154)
- To change the evaluation order of AND/OR'ed events (see page 155)
- To insert a storage control action (see page 159) (in a trigger sequence step)
- To specify a trigger sequence step's goto or trigger action (see page 150)
- Understanding Logic Analyzer Triggering, Storage Qualification (see page 347)

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 142).
- **Anything**—Any sample.
- **Nothing**—No sample.
- **Timer**—A timer value, see To insert a timer event (see page 153).
- **Counter**—A counter value, see To insert a counter event (see page 153).
- **Flag**—A flag value, see To insert a flag event (see page 154).
- **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 154).

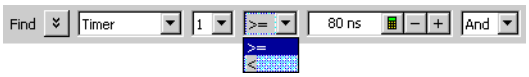
3 If inserting an event, specify whether the event should be **And**'ed or **Or**'ed with the other events.

See Also
- To negate events (see page 154)
- To change the evaluation order of AND/OR'ed events (see page 155)
- Understanding Logic Analyzer Triggering, Boolean Expressions (see page 344)

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 139).

See Also
- To insert or delete events (see page 152)

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.

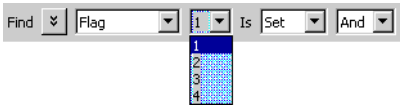
3 Enter the counter value to compare against.

For more information on counters, see Using Counters (see page 140).

See Also
- To insert or delete events (see page 152)

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.

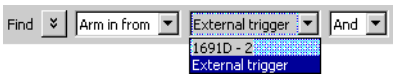
Multiple modules can insert flag events for the same flag to test whether the flag is set or clear.

For more information on flags, see Using Flags (see page 141).

See Also
- To insert or delete events (see page 152)

To insert an "Arm in from" event

1 Specify the source of the arming signal by selecting another module or **External trigger**.



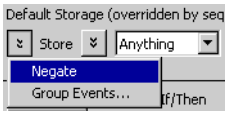
For more information on triggering on signals from other logic analyzer modules or external instruments, see Triggering From, and Sending Triggers To, Other Modules/Instruments (see page 164).

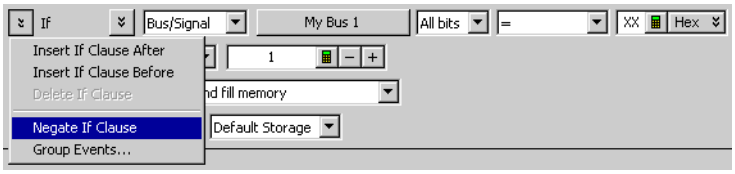
See Also
- To insert or delete events (see page 152)

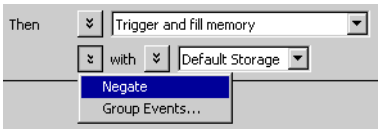
To negate events

Everywhere in the Advanced Trigger dialog where you can edit or insert events, you can also negate the events.

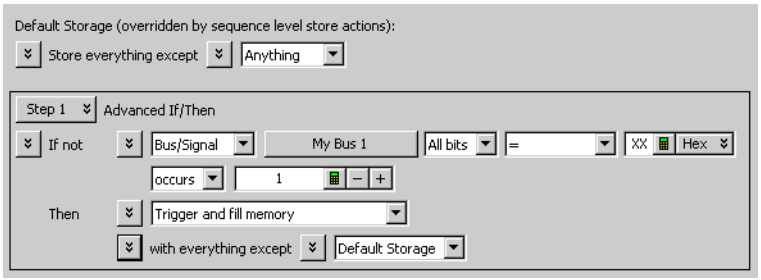
1 Click the ⁞ button associated with the events, and choose **Negate**.







Text in the dialog changes to indicate that events are negated.



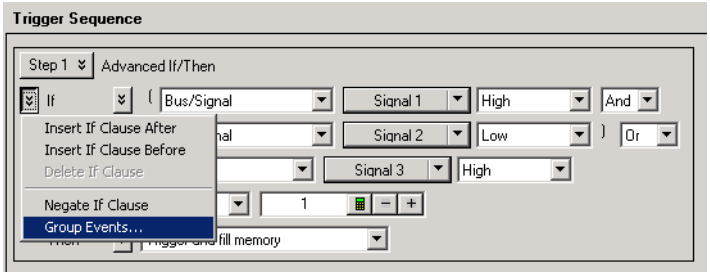
NOTE **Negate** is not available for storage qualifier events in the *Turbo State Mode* (see page 108).

See Also
- To insert or delete events (see page 152)

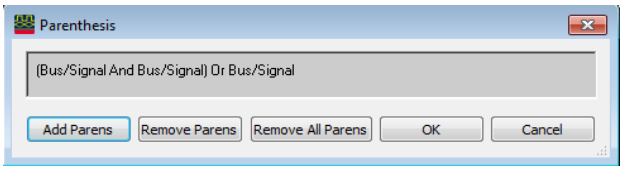
To change the evaluation order of AND/OR'ed events

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.



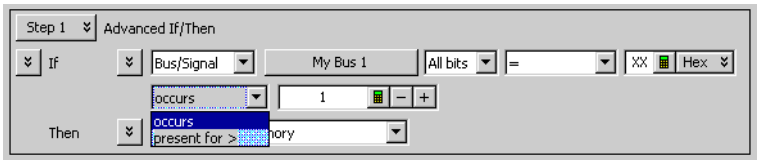
3 When you have finished grouping events, click **OK**.

See Also
- To insert or delete events (see page 152)

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.

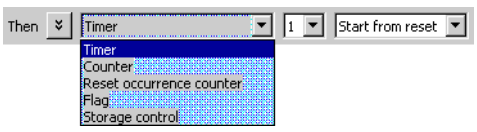


See Also
- To convert a trigger sequence step to Advanced If/Then trigger functions (see page 160)

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 157).
- **Counter**—For incrementing or resetting a counter, see To insert a counter action (see page 157).

- **Reset occurrence counter**—For resetting the occurrence counter, see To insert a reset occurrence counter action (see page 157).
- **Flag**—For setting or clearing a flag, see To insert a flag action (see page 158).
- **Storage control**—For storing samples or not or for turning on/off default storage, see To insert a storage control action (see page 159).

See Also
- To insert or delete events (see page 152)

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 139).

See Also
- To insert or delete actions (in a trigger sequence step) (see page 156)

To insert a counter action

1 Select the counter that you want to specify an action for.

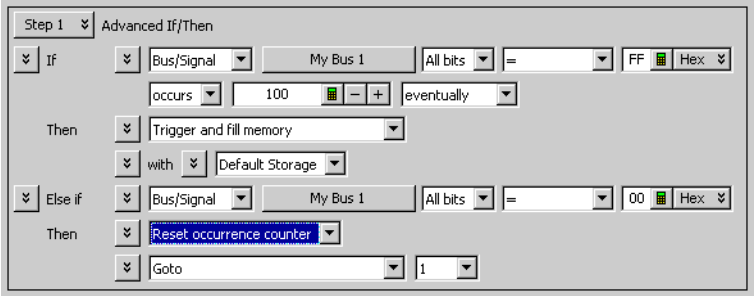
2 Specify the counter action by selecting either **Increment** or **Reset**.

For more information on counters, see Using Counters (see page 140).

See Also
- To insert or delete actions (in a trigger sequence step) (see page 156)

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 156)
- Understanding Logic Analyzer Triggering, Occurrence Counters (see page 345)

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 141).

See Also
- To insert or delete actions (in a trigger sequence step) (see page 156)

To insert a storage control action

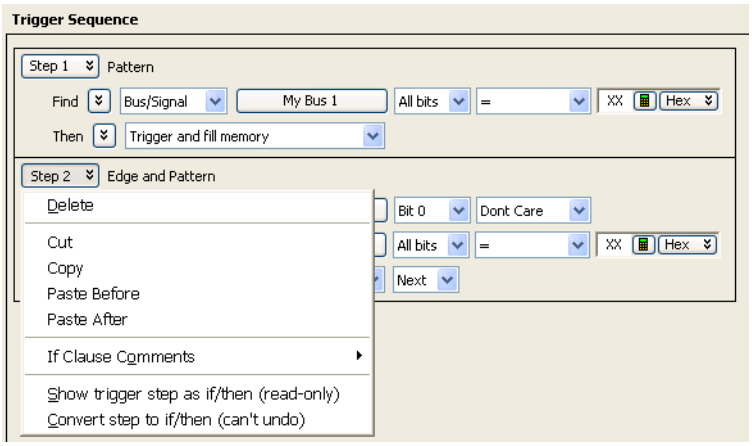
1 Specify the storage control action by selecting either **Store sample**, **Don't store sample**, **Turn on default storage**, or **Turn off default storage**.

For more information on sequence step storage and storage control actions, see Understanding Logic Analyzer Triggering, Storage Qualification (see page 347).

See Also
- To insert or delete actions (in a trigger sequence step) (see page 156)

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**.

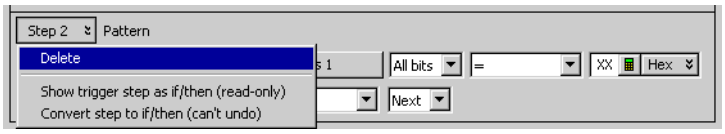


There must be at least one step in the trigger sequence.

See Also
- To delete trigger sequence steps (see page 160)

To delete trigger sequence steps

1 In a trigger sequence step, click **Step N** and choose **Delete**.

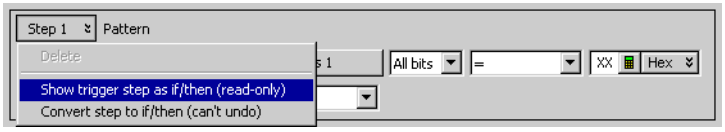


There must be at least one trigger sequence step.

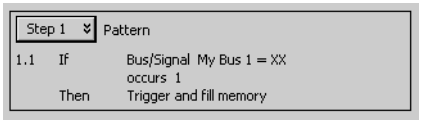
To show a trigger sequence step as Advanced If/Then trigger functions

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)**.

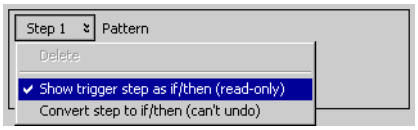


The trigger sequence step will be shown as the equivalent Advanced If/Then trigger functions in read-only form.



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

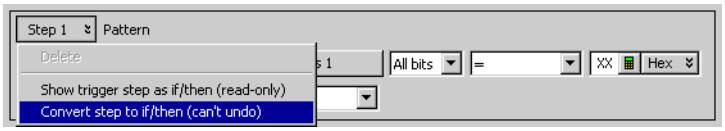
- To convert a trigger sequence step to Advanced If/Then trigger functions (see page 160)

To convert a trigger sequence step to Advanced If/Then trigger functions

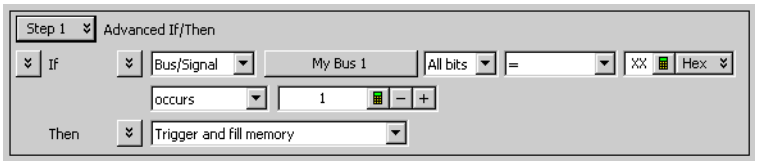
If the trigger function you need doesn't exist, start with a trigger function that is close, convert the trigger sequence step to advanced If/Then trigger functions, and edit the If/Then trigger functions.

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.



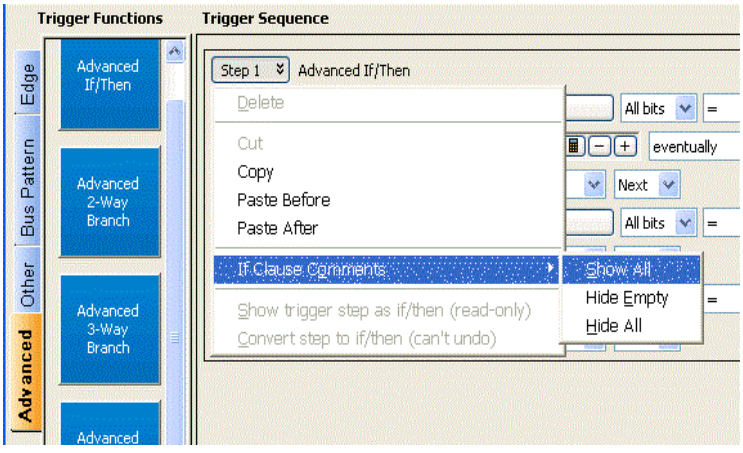
See Also
- To show a trigger sequence step as Advanced If/Then trigger functions (see page 160)
- Understanding Logic Analyzer Triggering, Branches (see page 345)

To display or hide "If" clause comments

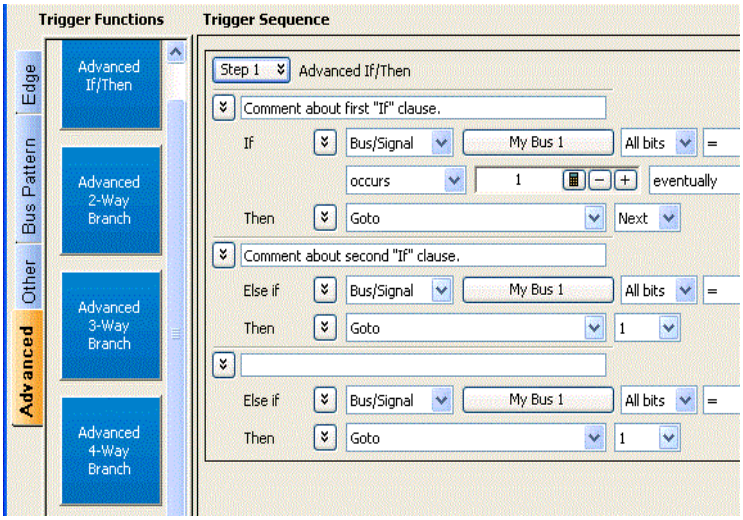
When using Advanced If/Then trigger functions, you can include comments with the "If" clauses in a trigger sequence step. It is useful to have descriptions in complex trigger functions.

To show all "If" clause comments

1 In a trigger sequence step, click **Step N** and choose **If Clause Comments>Show All**.

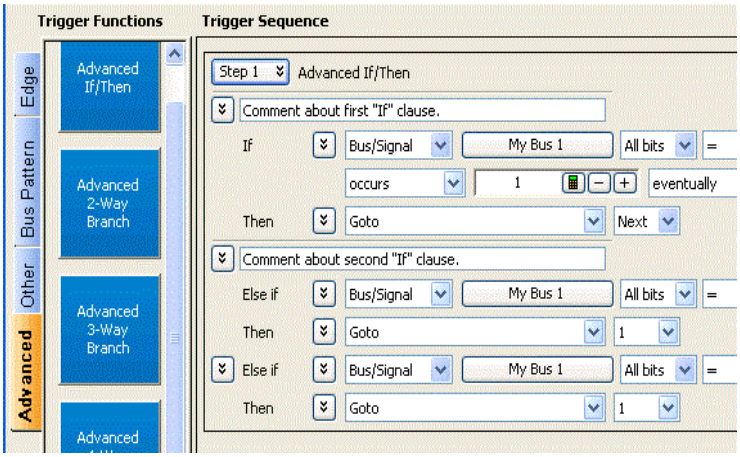


All "If" clause comments are show, including empty comments.



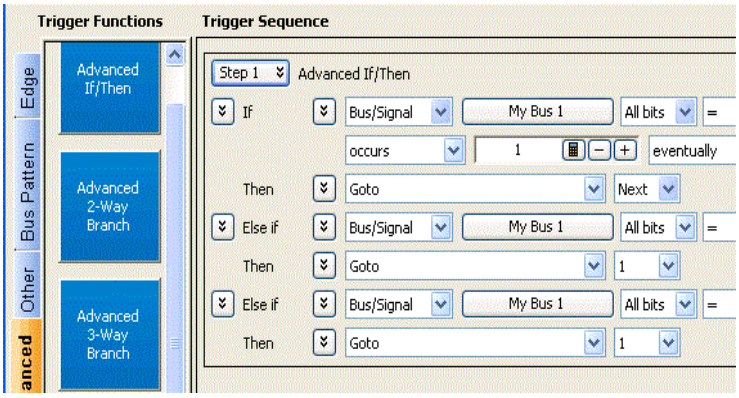
To hide empty "If" clause comments

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 160)

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 164 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 169 and "To trigger analyzer from another instrument - Trigger In" on page 170.

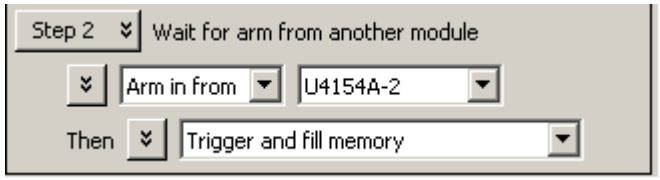
To arm one module with another module's trigger

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

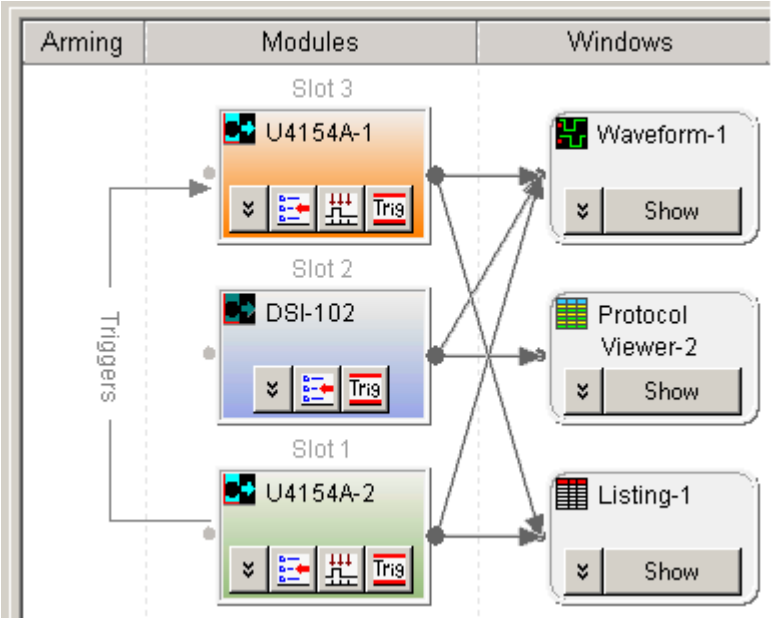
2 In the Advanced Trigger dialog (see page 392), select the **Other** trigger functions tab; then, drag-and-drop the **Wait for arm from another module** trigger function into the trigger sequence area.



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:

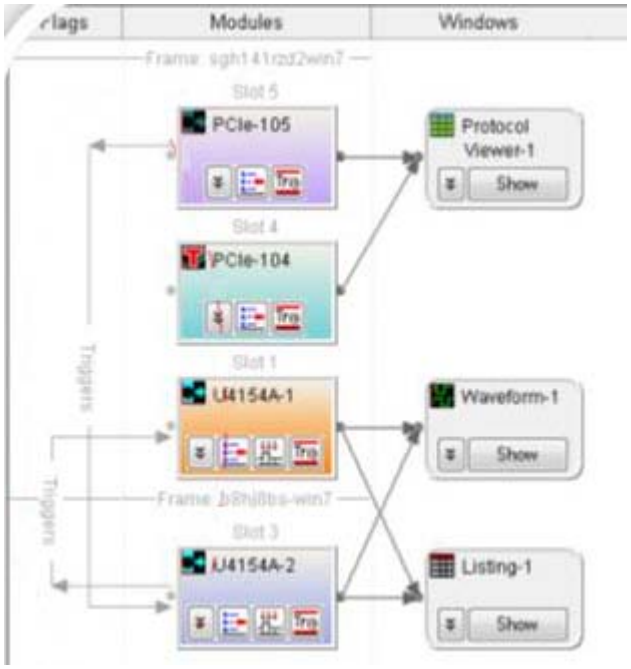


5 Run the measurement (see Running/Stopping Measurements (see page 174)).

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

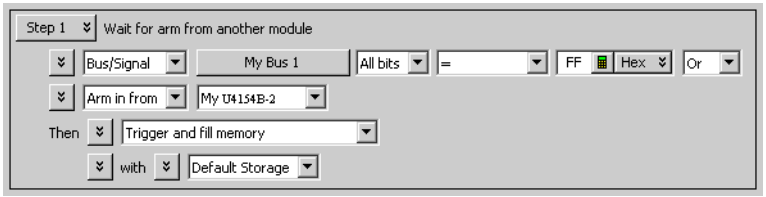
You can create an ARM In trigger in the same manner (as described above) between modules installed in different chassis of a multiframe setup. In a multiframe setup, flags are consumed for each ARM IN trigger configuration that you create. There are 4 flags available. If all these flags are unused, then 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.



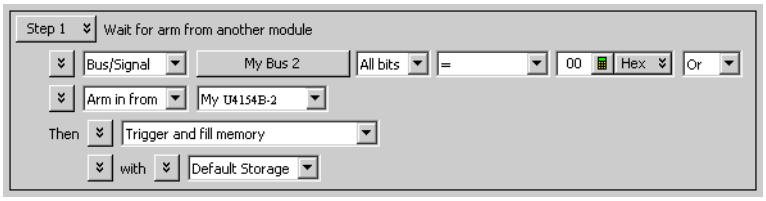
And vice-versa

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:

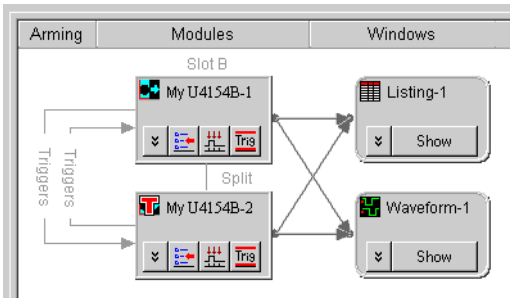
1 Set up the first module's trigger:



2 Set up the second module's trigger:



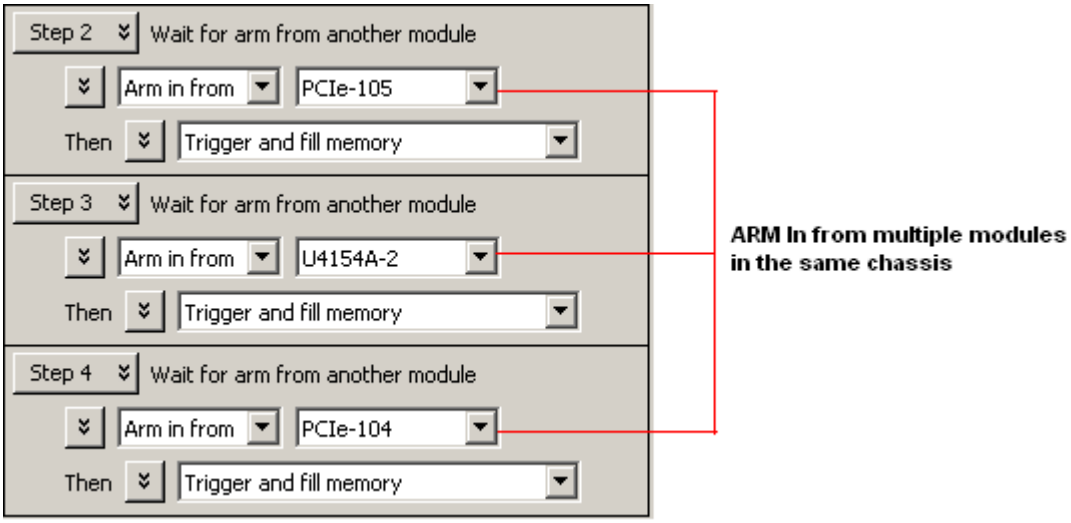
The arming setup in the Overview window looks like:



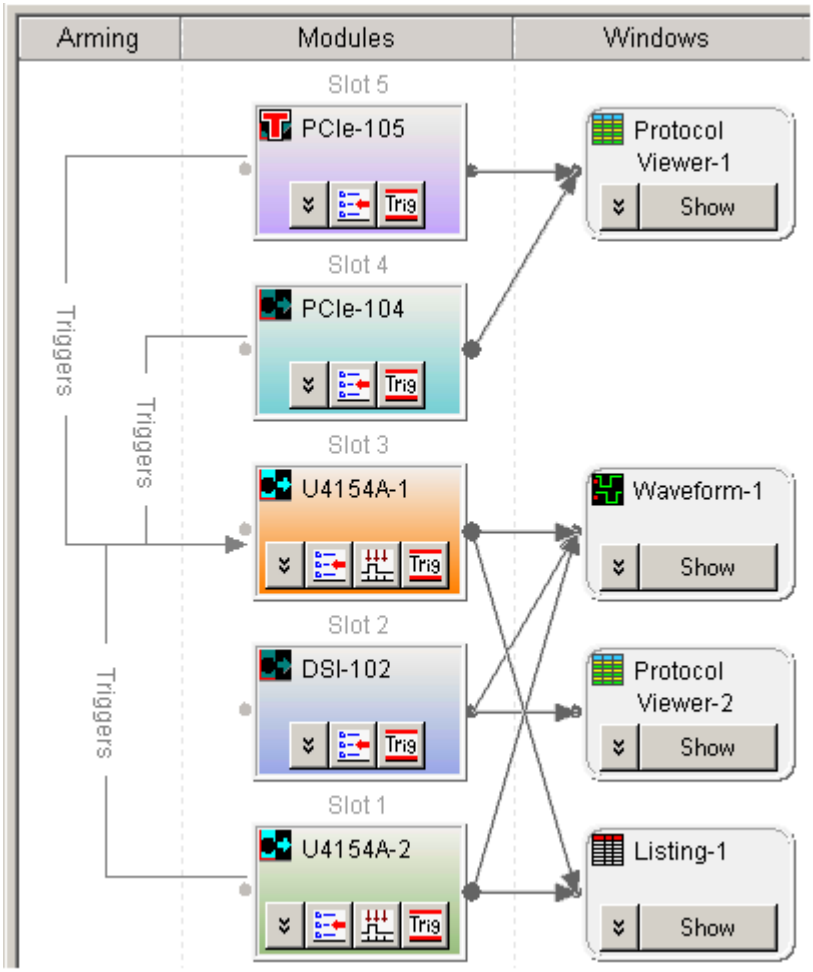
3 Run the measurement.

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.



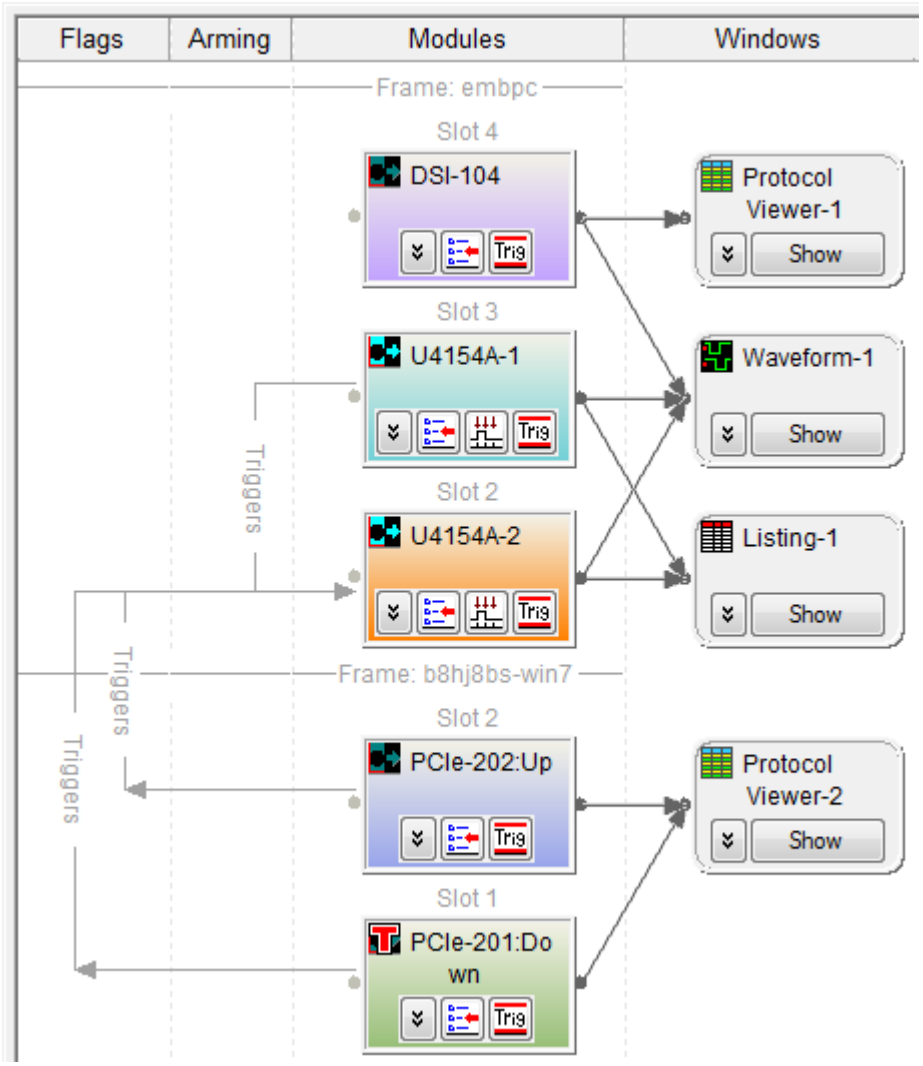
The arming setup in the Overview window looks like:



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 461)
- Wait for arm from another module (timing) (see page 449)

To trigger other instruments - Trigger Out

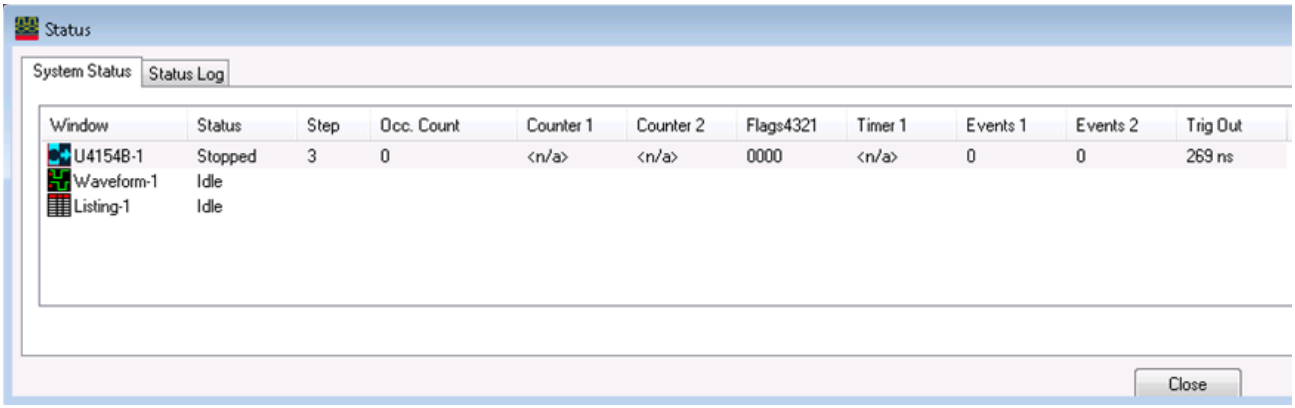
1 Connect a BNC cable from the **Trigger Out** BNC to the external instrument you want to trigger.
2 Choose **Setup>External Trigger....**
3 In the External Trigger dialog (see page 404):
   - Enable the output.
   - Select the polarity (active high or active low).
   - Select the output mode (use feedthrough 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.

5 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 404)

To trigger analyzer from another instrument - Trigger In

1 Connect a BNC cable from the TRIGGER IN BNC to the external instrument that will send the trigger signal.

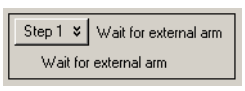
2 Choose **Setup>External Trigger....**

3 In the External Trigger dialog (see page 404), specify whether a rising or falling edge on the **Trigger In** BNC will indicate a trigger; then, click OK.

4 Click

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

5 In the Advanced Trigger dialog (see page 392), 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 174)).

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 460)
- Wait for external arm (timing) (see page 448)
- External Trigger Dialog (see page 404)

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 172) to the favorites list.
- By storing them (see page 172) to XML format trigger specification files.

You can recall triggers (see page 172) 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 173).

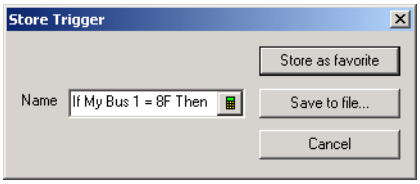
You can control the length of the recently-used and favorites list by setting the trigger history depth (see page 173).

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 172)
- To set the trigger history depth (see page 173)

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 417):

- 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 417), 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 172)

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 172)
- To recall a trigger (see page 172)

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 374).

See also To save captured data after each run (see page 174) 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 374).

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 174) so that the location being displayed doesn't change after each measurement.

See also To save captured data after each run (see page 174) and To stop repetitive runs after a certain number of acquisitions (see page 174) 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 430).

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 374).

To save captured data after each run

1. From the menu bar, choose **Run/Stop>Run Properties...**.
2. In the Run Properties dialog (see page 418), check **Save after every acquisition** and set the additional options for saving data after each run; then, click **OK**.
3. Run the analyzer in either single run mode or repetitive run mode.

To stop repetitive runs after a certain number of acquisitions

1. From the menu bar, choose **Run/Stop>Run Properties...**.
2. In the Run Properties dialog (see page 418), check **Stop running after** and enter the number of acquisitions; then, click **OK**.
3. Run the analyzer in repetitive run mode.

See Also

- To change the "Go to Trigger on Run" option (see page 174)
- Run Properties Dialog (see page 418)

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 412), 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).

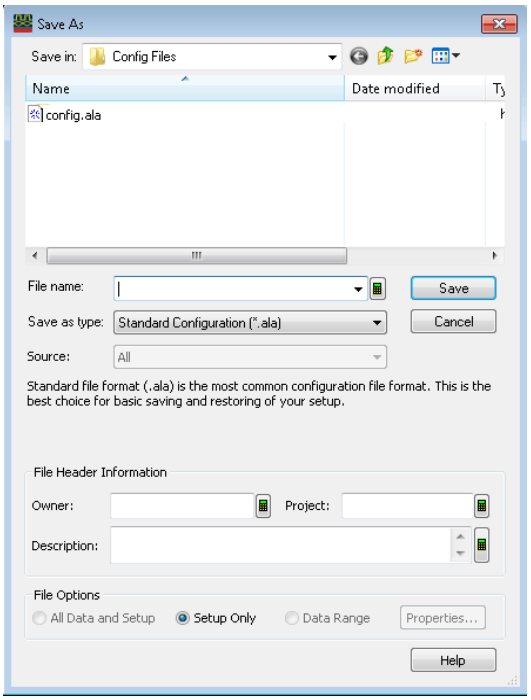
You can also save captured data to comma-separated value (CSV) files.

- To save a configuration file (see page 176)
- To export data to standard CSV format files (see page 177)
- To export data to module CSV format files (see page 179)
- To export data to module binary (ALB) format files (see page 181)

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 371).



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 352).

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.**

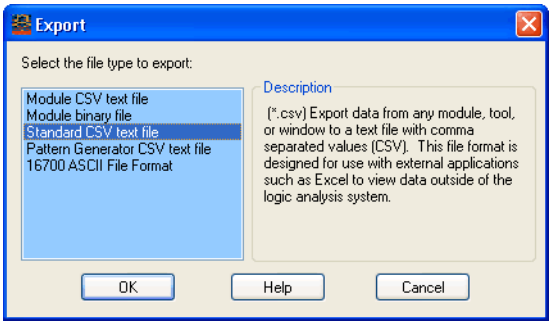
See Also
- To open a configuration file (see page 188)
- Offline Analysis (see page 194)
- ALA vs. XML, When to Use Each Format (see page 352)
- ALA Format (see page 466)
- "XML Format" (in the online help)

To export data to standard CSV format files

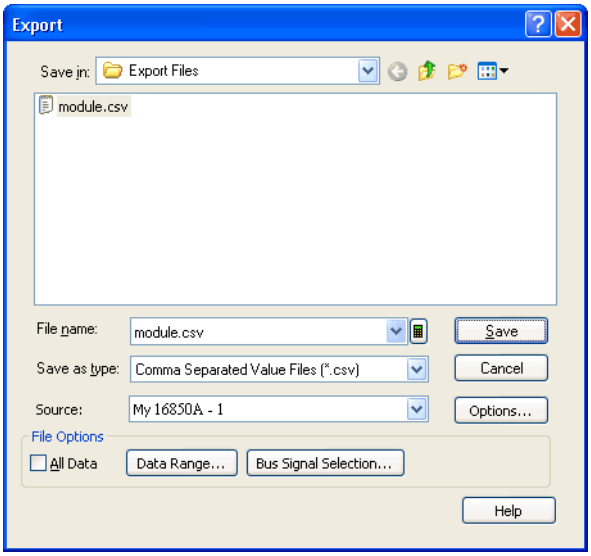
You can export captured data to standard comma-separated value (CSV) files.

1 From the menu bar, select **File>Export...**.

2 In the Export dialog, select **Standard CSV text file**; then, click **OK**.



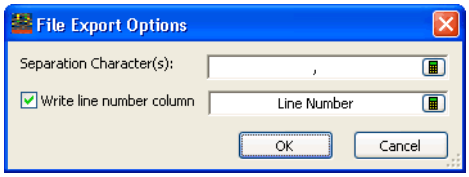
3 In the following Export dialog:



a Enter the CSV file name.

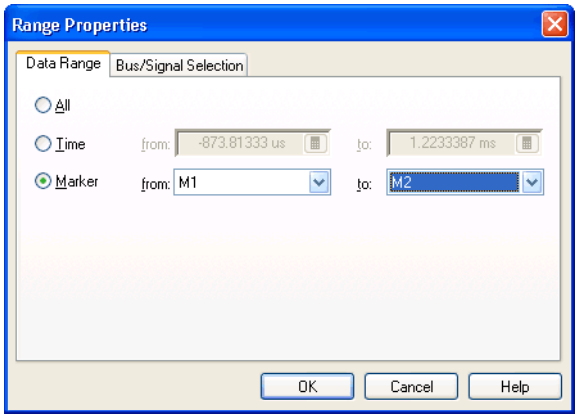
b Select the **Source** module, tool, or display window from which to export data.

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**.

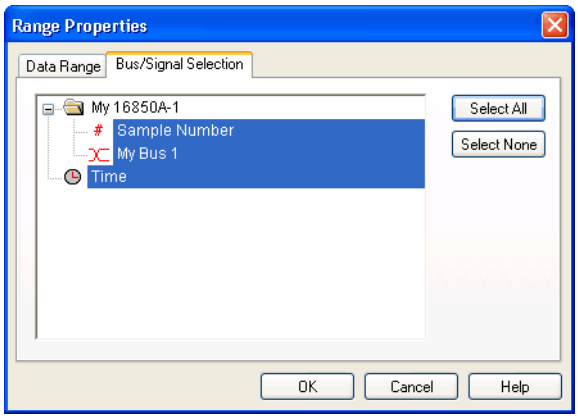


d If you want to export a range of data and/or selected bus/signal data, uncheck **All Data**.

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**.

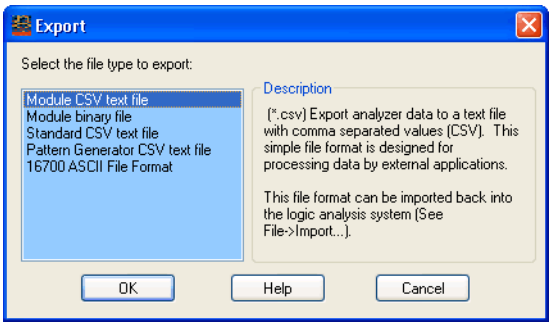
See Also
- Standard CSV Format (see page 466)

To export data to module CSV format files

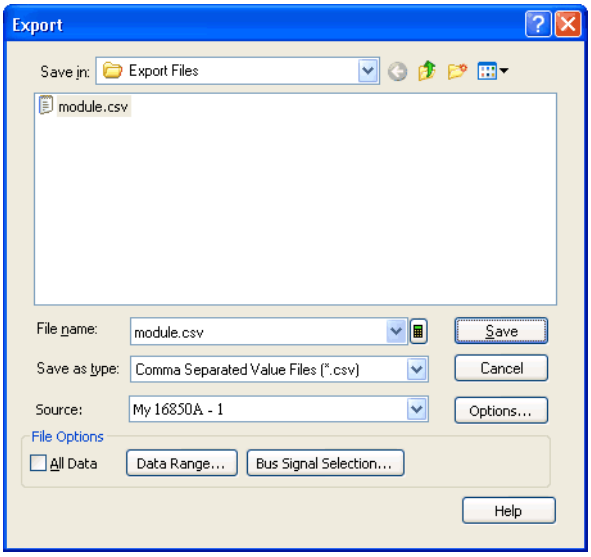
You can export captured data to module comma-separated value (CSV) files. Module CSV files can be post-processed.

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**.



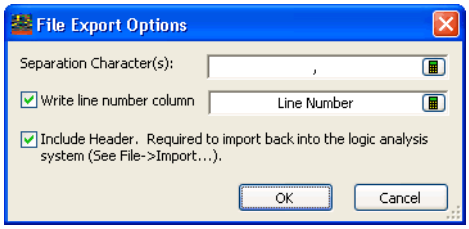
3 In the following Export dialog:



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.

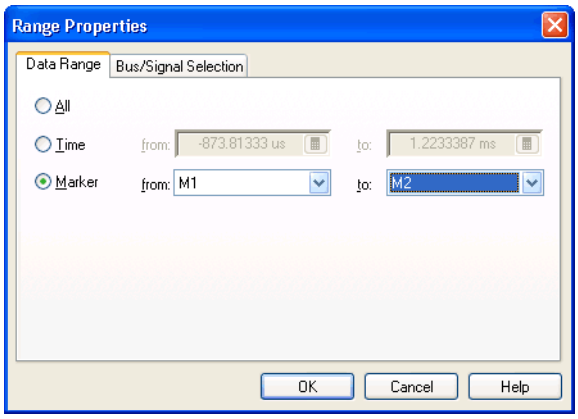
c If you want to use a delimiter other than a comma, if you want to specify that line numbers be written, or if you want to exclude the header information, click **Options...**. In the File Export Options dialog, make your selections; then, click **OK**.



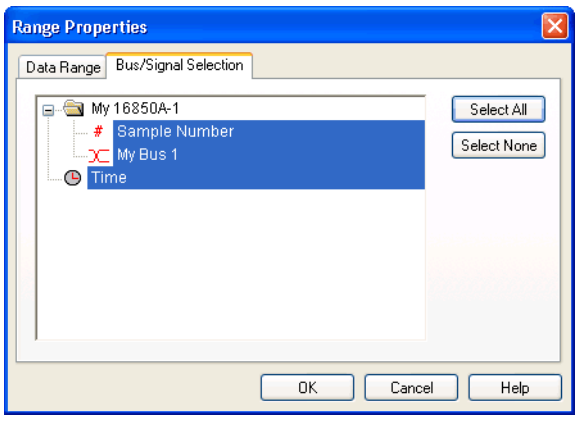
Header-less files may be easier for external tools to use.

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**.

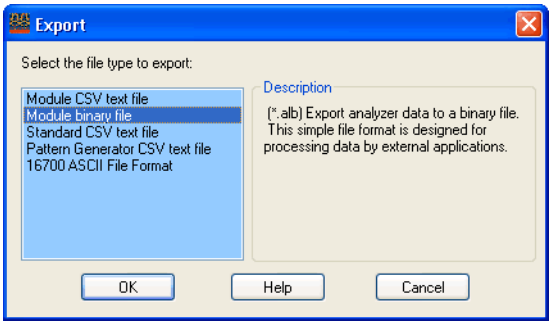
See Also
- Module CSV Format (see page 466)

To export data to module binary (ALB) format files

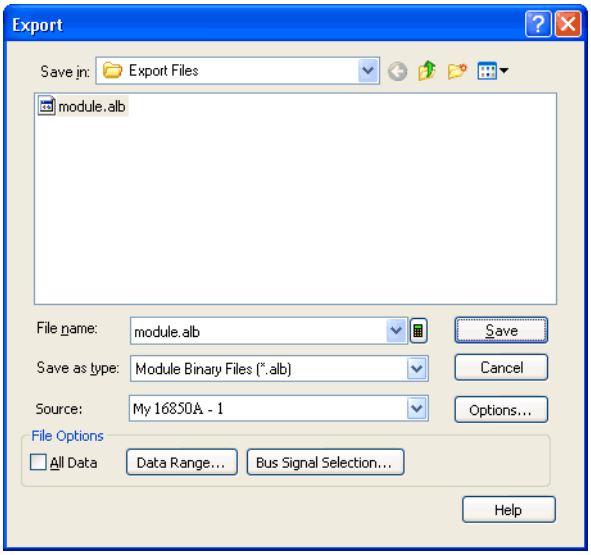
You can export captured data to module binary files.

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:

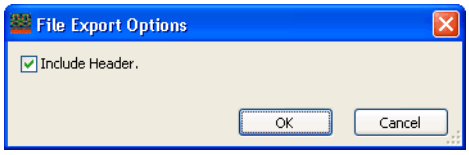


a Enter the ALB file name.

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.

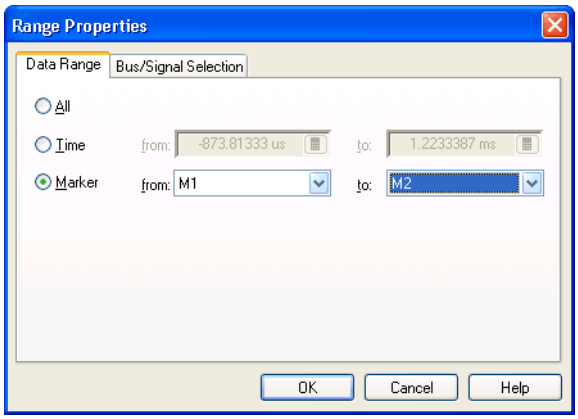
c If you want to exclude the header information, click **Options...**. In the File Export Options dialog, make your selections; then, click **OK**.



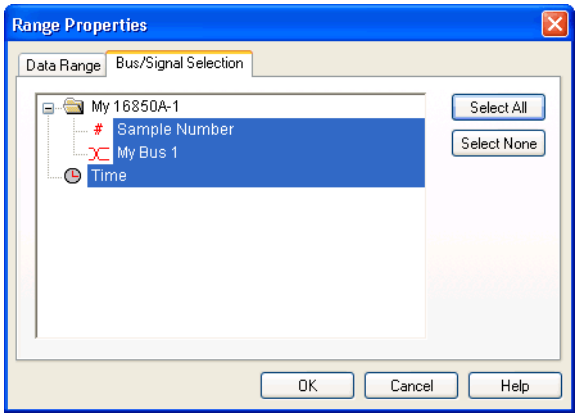
Header-less files may be easier for external tools to use.

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**.

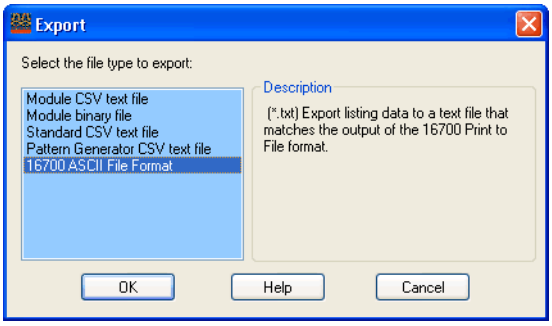
See Also
- Module Binary (ALB) Format (see page 473)

To export data to 16700 ASCII format files

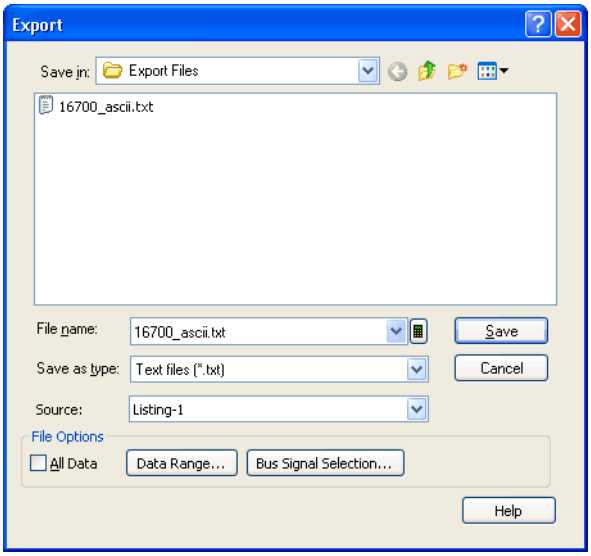
You can export captured data to a text file that matches the output of the 16700 logic analysis system's "Print to File" format.

1 From the menu bar, select **File>Export...**.

2 In the Export dialog, select **16700 ASCII File Format**; then, click **OK**.



3 In the following Export dialog:

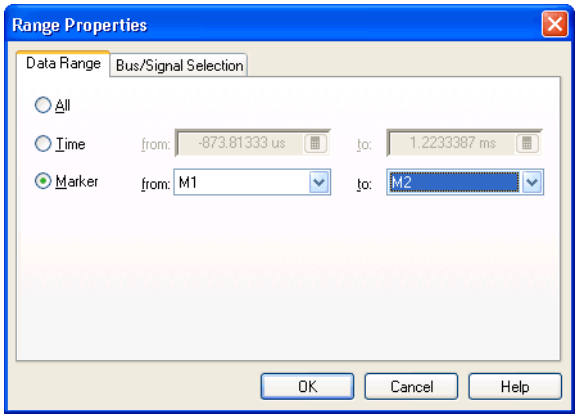


a Enter the 16700 ASCII text file name.

b Select the **Source** Listing display window from which to export data.

c If you want to export a range of data, uncheck **All Data**.

d 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**.


Range Properties
Data Range
Bus/Signal Selection
All
Time
from:
-873.81333 us
to:
1.2233387 ms
Marker
from:
M1
to:
M2
OK
Cancel
Help

4 Click **Save**.

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 / 188
Offline Analysis / 194
Analyzing Waveform Data / 197
Analyzing Listing Data / 212
Displaying Names (Symbols) for Bus/Signal Values / 220
Marking, and Measuring Between, Data Points / 222
Searching the Captured Data / 240
Comparing Captured Data to Reference Data / 250
Viewing Source Code Associated with Captured Data / 253
Analyzing Packet Data / 259
Analyzing the Same Data in Different Ways (Using the Overview Window) / 283
Setting the System Trigger and Skew Between Modules / 288
Using Display Windows / 289
Printing Captured Data / 290

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 188)
- To recall a recently used configuration file (see page 190)
- To transfer module setups to/from multi-module systems (see page 190)
- To set up multiple-modules with XML-format configurations (see page 191)

See Also
- Offline Analysis (see page 194)

To open a configuration file

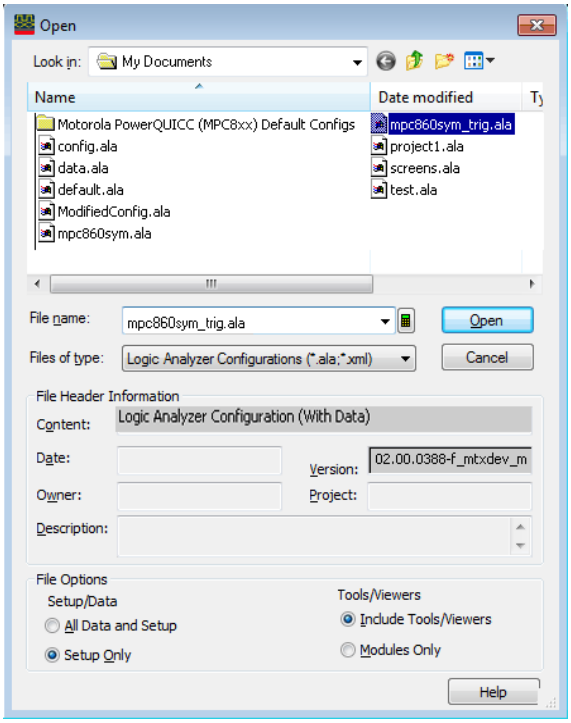
You can open configuration files to return to a previous logic analysis session, to load previously saved data for *offline analysis*, or to load saved logic analyzer setups.

NOTE **To avoid *pod truncation* (see page 540) 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 358).



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 352).

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.

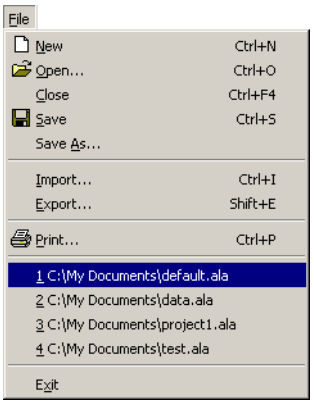
6 Select **Open**.

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 190)
- To save a configuration file (see page 176)
- To transfer module setups to/from multi-module systems (see page 190)
- Offline Analysis (see page 194)
- ALA vs. XML, When to Use Each Format (see page 352)
- ALA Format (see page 466)
- "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 188)).
3 Answer the question about clearing all modules before loading.
4 In the Module Mapping dialog (see page 409), select **Manually specify module mapping**; then, click **Specify Mapping...**.
5 In the Specify Mapping dialog (see page 429), 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 176)

- To open a configuration file (see page 188)
- ALA vs. XML, When to Use Each Format (see page 352)

To set up multiple-modules with XML-format configurations

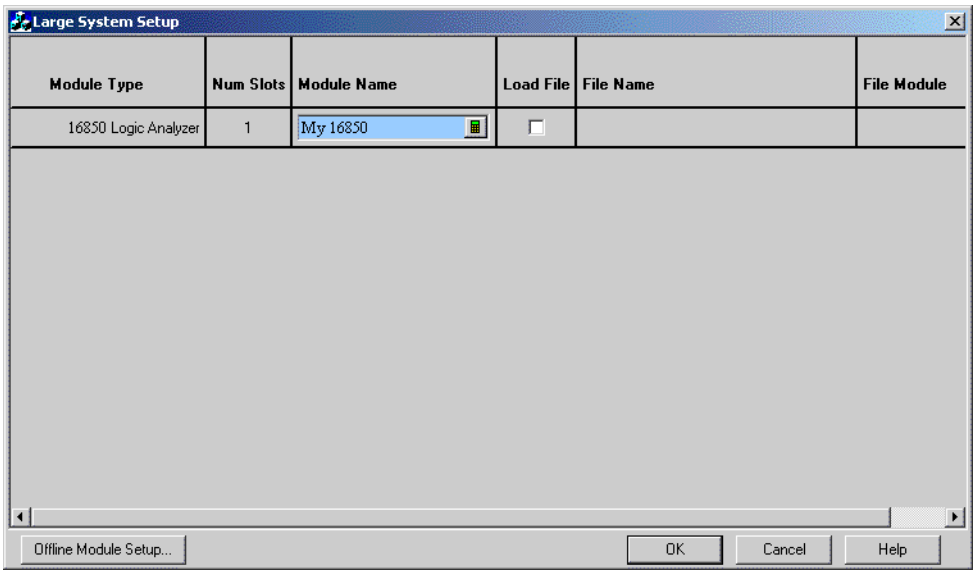
Included with the *Keysight Logic Analyzer* application is the Large System Setup utility program which makes it easy to set up large, multiple-module logic analysis systems using XML-format configuration files. For example, if your logic analysis system has several logic analyzer modules, each probing the same kind of bus, you can use the Large System Setup utility to set up each module with the same XML-format configuration file. (Doing the same thing in the main application requires many time-consuming steps.)

The Large System Setup utility program can also be used in *offline* mode. In this case, you can add as many logic analyzer modules as desired and specify XML-format configurations for each.

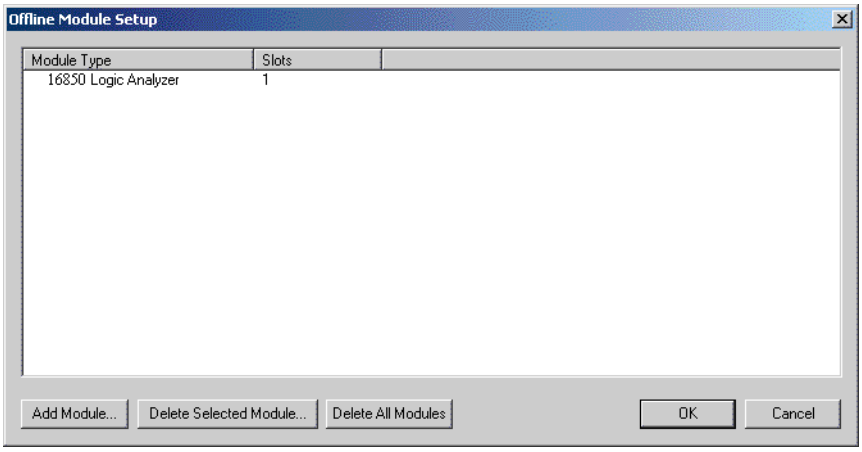
To use the Large System Setup utility program:

1 If the *Keysight Logic Analyzer* application hasn't already been started, start it, and connect to the logic analysis system you want to set up, or go offline if you want to set up in *offline* mode.

2 From the Windows Start menu, choose **Start>All Programs>Keysight Logic Analyzer>Utilities>Large System Setup**.



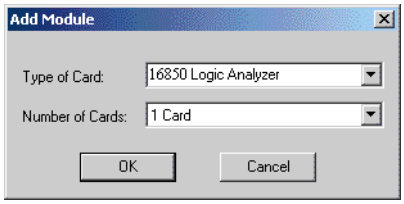
3 If you have connected to the logic analysis system you want to set up, go to step 5; otherwise, in offline mode, click **Offline Module Setup...** in the Large System Setup dialog.



4 In the Offline Module Setup dialog, for each module that you want to add:

a Click **Add Module...**.

b In the Add Module dialog, choose the type of card the module is made up of and the number of cards in the module.



c Click **OK**.

You can delete all or selected modules by clicking **Delete All Modules** or **Delete Selected Module**.

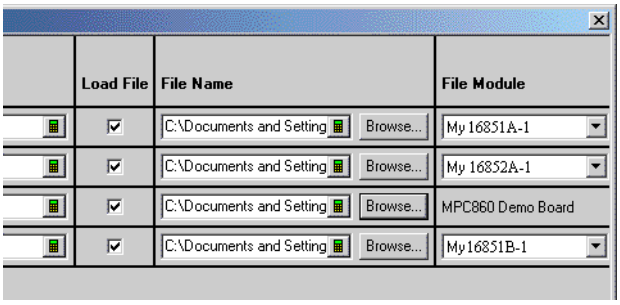
When you are finished adding modules, click **OK** to close the Offline Module Setup dialog.

5 In the Large System Setup dialog, enter the **Module Name** for each module.

(The module name from the XML-format configuration file is not used because the same XML file can be used to set up multiple modules.)

6 For each module that you want to configure with an XML-format configuration file, make sure the **Load File** box is checked, and click **Browse...** to select the configuration file name.

7 If the XML-format configuration files contain setups for multiple logic analyzer modules, select the file's module setup you want to use.



8 Click **OK** to close the Large System Setup dialog and set up the logic analysis system as specified.

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

- To analyze data from 16850-series logic analyzers, the data must be saved in the fast binary data format. This is done using the File Out Tool.
- 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 195); 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 194)
- Offline Analysis on Personal Computers (see page 195)

To return to online analysis, see:

- Connecting to a Logic Analysis System (see page 75)

See Also

- Offline File Formats (see page 195)

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 291) to enable file sharing.

See Also
- To open a configuration file (see page 188)
- To save a configuration file (see page 176)

Offline Analysis on Personal Computers

A personal computer (PC) with the *Keysight Logic Analyzer* application installed can perform offline analysis on fast binary data files saved from a logic analyzer.

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 291) to enable file sharing.

See Also
- To open a configuration file (see page 188)
- To save a configuration file (see page 176)
- Minimum PC Requirements (see page 195)

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 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 466) 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.

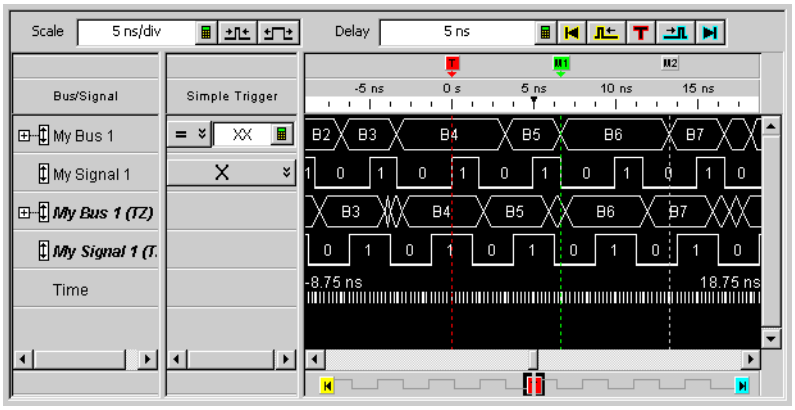
The generic configuration (.xml) file format is an eXtensible Markup Language format that can be edited (using an ASCII text editor) and post-processed by scripts (or other tools) and re-opened by the *Keysight Logic Analyzer* application. These files contain buses/signals, channels assigned to buses/signals, and user-defined symbols.

CSV Format

You can export captured data to CSV (Comma-Separated Values) Format (see page 466) 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 289) turned on, you can also select a tab at the bottom of the window.

- To change the display scale (time/division) (see page 198)
- To go to different locations in the captured data (see page 199)
- To re-arrange waveforms (see page 200)
- To overlay waveforms (see page 200)
- To find a bus/signal row (see page 201)
- To view bus data as a chart (see page 201)
- To show/hide parts of the waveform display (see page 203)
- To insert or delete buses/signals (see page 204)
- To group signals into a bus (see page 204)
- To expand/collapse buses (see page 204)
- To insert separator rows (see page 204)
- Changing Waveform Window Properties (see page 204)
  - To change the waveform background color (see page 205)
  - To change the overlaid waveform color (see page 205)
  - To change the filtered data color (see page 206)
  - To change the timing zoom background color (see page 206)
  - To change the waveform font size (see page 206)
  - To change the Fast Zoom In option (see page 207)
  - To lock scrolling with other display windows (see page 206)
  - To change the waveform tool tip display (see page 207)
- Changing Bus/Signal Row Properties (see page 207)
  - To change a waveform's color (see page 208)
  - To change a waveform's height (see page 209)
  - To change a bus/signal's number base (see page 209)

- To show/hide a bus/signal's numeric data values (see page 209)

- Changing Analog Signal Row Properties (see page 209)
  - To change the analog properties (see page 210)

See Also

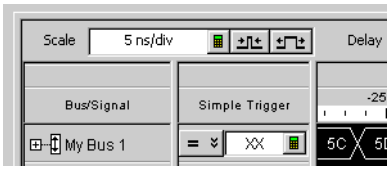
- Defining Buses and Signals (see page 93)
- Setting Up Quick (Draw Box) Triggers (see page 129)
- Specifying Simple Triggers (see page 132)
- Marking, and Measuring Between, Data Points (see page 222)
- Setting Up Symbols (see page 118)
- Displaying Names (Symbols) for Bus/Signal Values (see page 220)
- Searching the Captured Data (see page 240)

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.



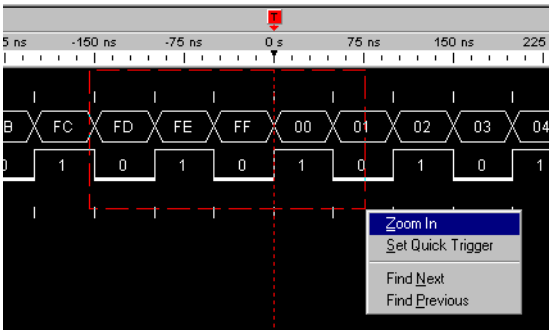
The scale ranges from 1 ps/div to 1 ks/div.

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 207) option is not selected, choose **Zoom In** from the popup menu.

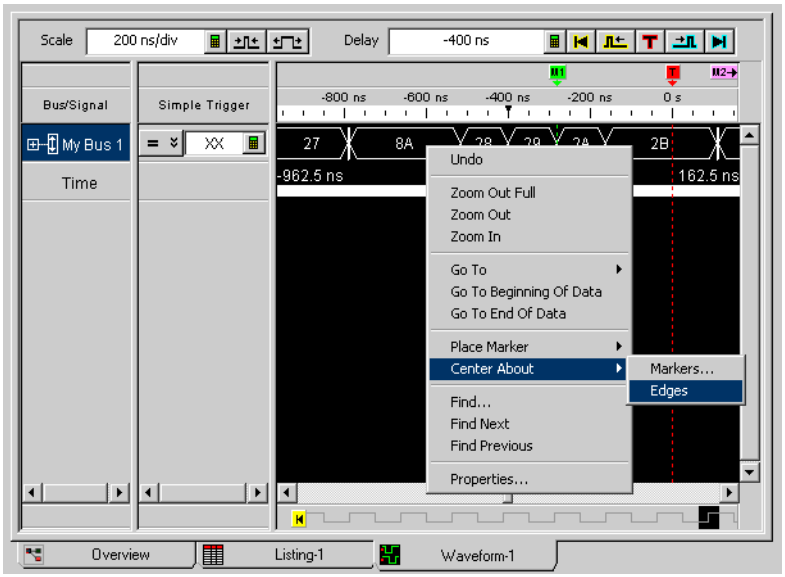


The new display scale is adjusted to the width of the box drawn.

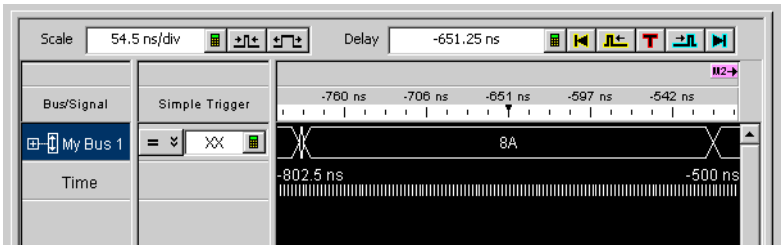
To center about edges

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

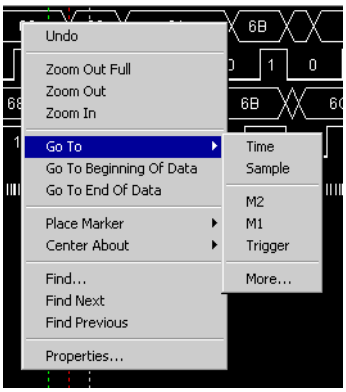
1 Click one of the buttons in the Delay field or enter a delay value.

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.

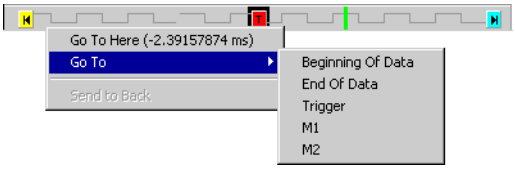
Menu	Description
	Use the keypad to enter a numeric value. If the value you enter is greater than or less than the time of the data range, the window will be moved to the beginning or end limit.
	Moves the window over the beginning of data.
	Scrolls the window towards the beginning of data.
	Moves the 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**.

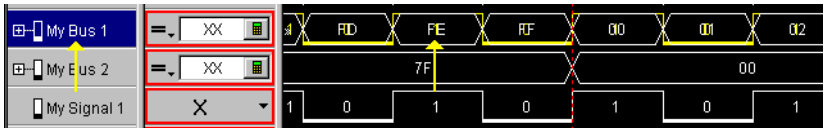
To re-arrange waveforms

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 200)
- To expand/collapse buses (see page 204)

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 205).

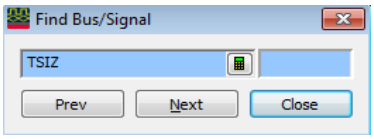
When analog signals are overlaid onto other signals, the overlay signal color comes from the external oscilloscope module's setup options (which you can access by right-clicking the analog signal name, choosing **Assign Channels...**, and selecting the " Options tab" (in the online help)) instead of 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:

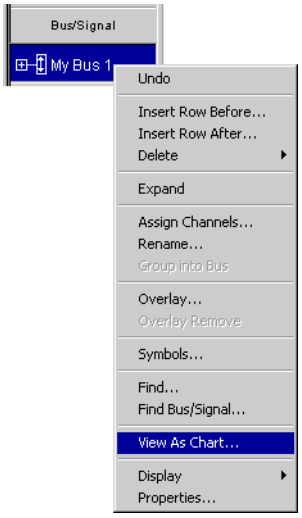
- **Prev** – to search for the string backward in the bus/signal rows.
- **Next** – to search for the string forward in the bus/signal rows.
- **Close** – to close the Find Bus/Signal dialog.

To view bus data as a chart

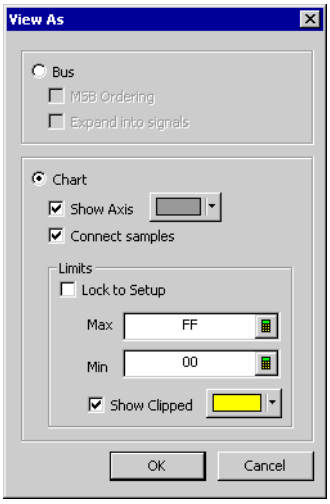
You can view bus data values as a chart instead of the conventional bus shape.

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...**.

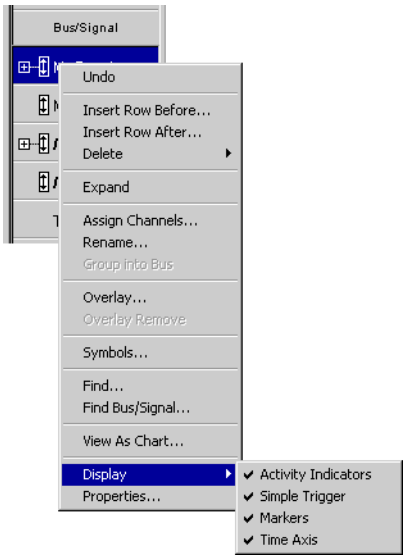
2 In the View As dialog, check or uncheck the following options as desired:

- **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 207)

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 380).
- **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 204)

To insert or delete buses/signals

To insert buses/signals

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

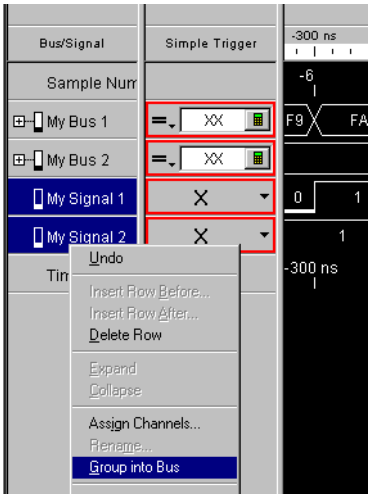
1 In the Bus/Signal column, highlight the buses/signals you want to delete (by clicking, Shift-clicking, or Ctrl-clicking the bus/signal names).
2 Right-click in the Bus/Signal column; then, choose **Delete>Row**.

See Also

- Defining Buses and Signals (see page 93)

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 93)

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.

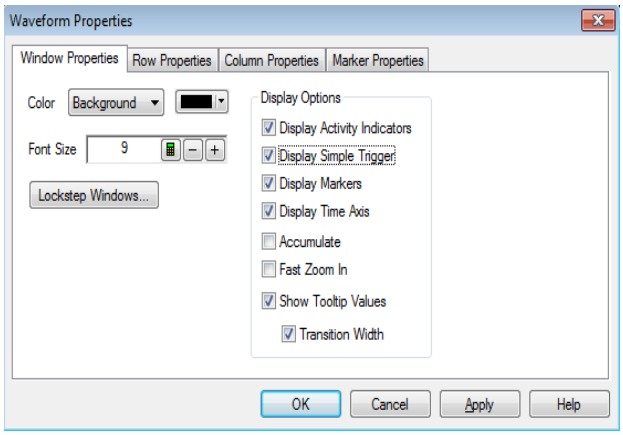
Changing Waveform Window Properties

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 205)
- Change the overlaid waveform color (see page 205)
- Change the filtered data color (see page 206)
- Change the timing zoom background color (see page 206)
- Change the waveform font size (see page 206)
- Lock scrolling with other display windows (see page 206)
- Show/hide parts of the waveform display (see page 203)
- Change the accumulate waveforms option (see page 206)
- Change the Fast Zoom In option (see page 207)
- Change the waveform tool tip display (see page 207)

3 Click **OK** to apply the changes and close the Waveform Properties dialog.

See Also
- Changing Bus/Signal Row Properties (see page 207)

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 200)), 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.

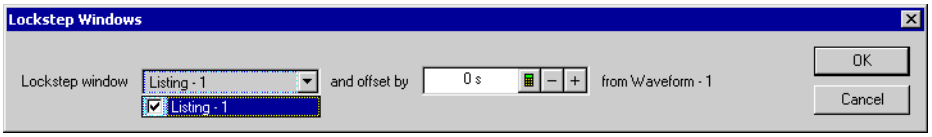
As the font size is changed, the row height may be automatically increased to fit the new text size.

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.

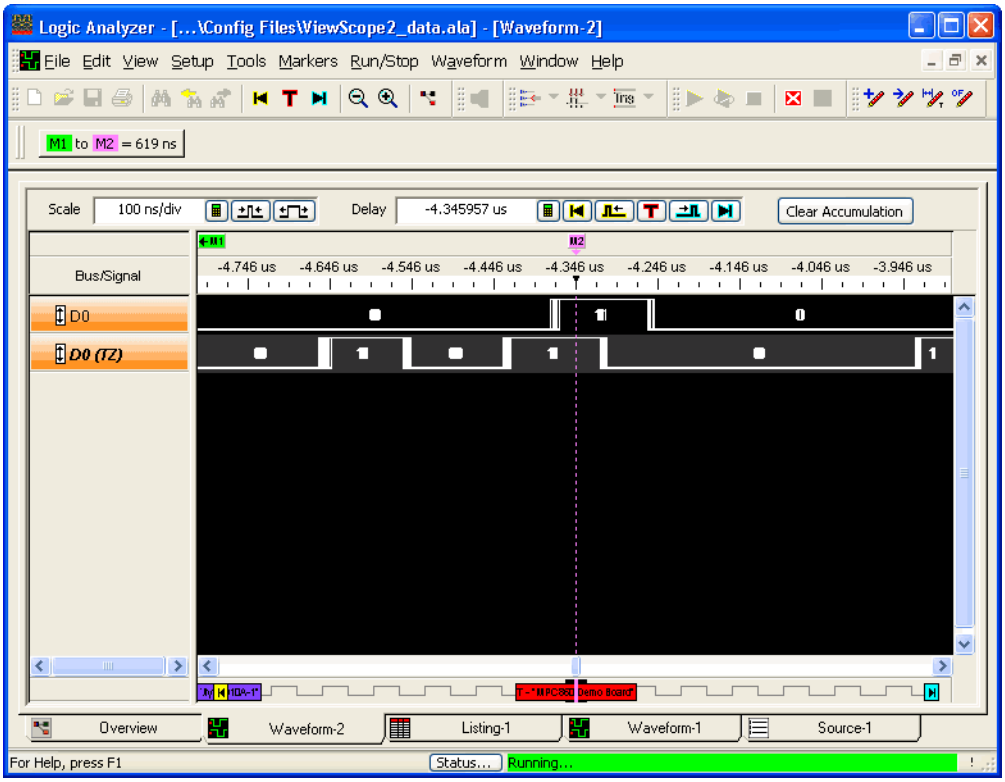
4 Click **OK** to apply the changes and close the Waveform Properties dialog.

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 198)
- To set a Quick Trigger in the Waveform window (see page 129)
- To quickly find bus signal patterns (see page 240)

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.

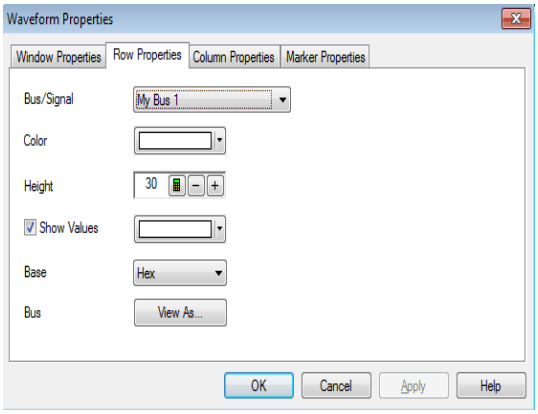
NOTE **Property changes to a bus affect all signals within the bus. For example, if you change the color of a bus and then expand the bus, you will see that the color is changed for all signals in the bus.**

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 93)).
  - **<all>** buses/signals.
  - **<selected>** buses/signals if more than one is highlighted in the Bus/Signal column.
- Change a waveform's color (see page 208)
- Change a waveform's height (see page 209)
- Change a bus/signal's number base (see page 209)
- Show/hide a bus/signal's numeric data values (see page 209)
- View bus data as a chart or a bus (see page 201)

3 Click **OK** to apply the changes and close the Waveform Properties dialog.

See Also
- Changing Waveform Window Properties (see page 204)

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 209)

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 206)). 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 209)

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 209)), 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 220))

2 Click **OK** to apply the changes and close the Waveform Properties dialog.

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 209)

Changing Analog Signal Row Properties

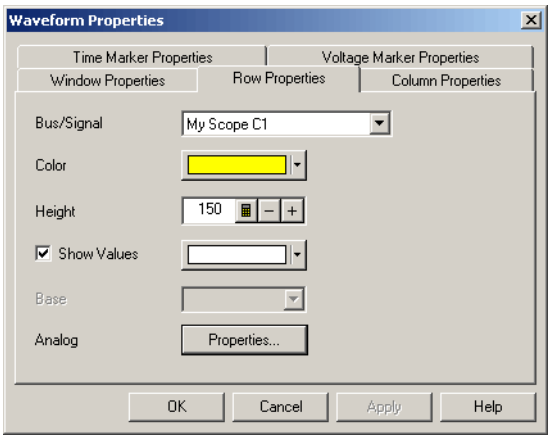
In the Waveform display window, you can change the color or size of an analog signal waveform, and you can choose whether voltage and volts/division values are displayed with the waveform. Analog signals come from an external oscilloscope module (see " External Oscilloscope Time Correlation and Data 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 93)).
  - **<all>** buses/signals.
  - **<selected>** buses/signals if more than one is highlighted in the Bus/Signal column.
- Change a waveform's color (see page 208)
- Change a waveform's height (see page 209)
- Show/hide a signal's numeric data values (see page 209)
- Change the analog properties (see page 210)

3 Click **OK** to apply the changes and close the Waveform Properties dialog.

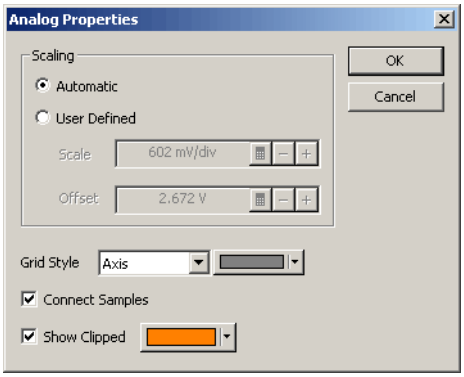
See Also
- Changing Waveform Window Properties (see page 204)

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...**.

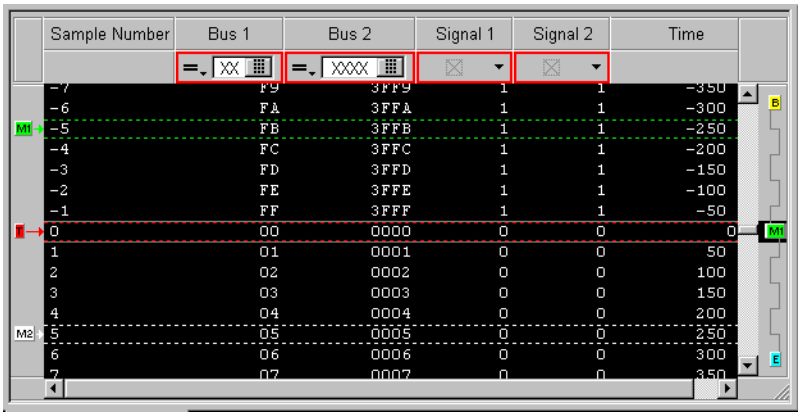
2 In the Analog Properties dialog, set the following options:



- **Scaling** – you can select **Automatic** (to have the voltage scale and offset automatically set) or **User Defined** (to be able to set your own voltage scale and offset values). Be careful not to enter scale and offset values that will move the waveform out of the display area.
- **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.



The Listing window is accessed through the menu bar's **Window>Listing**. If you have Tabbed Windows (see page 289) 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 213)
- To re-arrange bus/signal columns (see page 214)
- To find a bus/signal column (see page 214)
- To show/hide parts of the listing display (see page 214)
- To insert or delete buses/signals (see page 215)
- Changing Listing Window Properties (see page 215)
  - To change the listing background color (see page 216)
  - To change the timing zoom background color (see page 216)
  - To change the listing font size (see page 216)
  - To lock scrolling with other display windows (see page 216)
  - To show/hide the center rectangle (see page 216)
- Changing Bus/Signal Column Properties (see page 217)
  - To change a bus/signal's data color (see page 217)
  - To change the width of a bus/signal column (see page 218)
  - To change the alignment of a bus/signal column (see page 218)
  - To change a bus/signal's number base (see page 218)
  - To select the marker for marker-relative times (see page 219)
  - To select fixed time units (see page 219)

See Also

- Defining Buses and Signals (see page 93)
- Setting Up Quick (Draw Box) Triggers (see page 129)
- Specifying Simple Triggers (see page 132)
- Marking, and Measuring Between, Data Points (see page 222)
- Setting Up Symbols (see page 118)
- Displaying Names (Symbols) for Bus/Signal Values (see page 220)

- Searching the Captured Data (see page 240)

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.

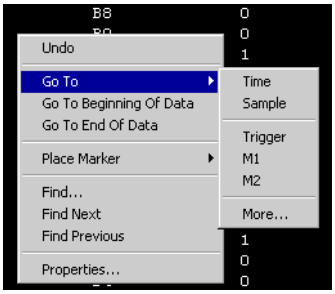
To go to different locations using toolbar buttons

1 Click one of the Go To buttons in the standard toolbar.

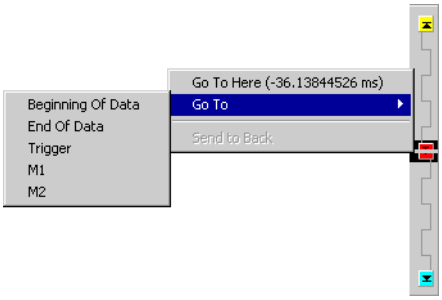
Menu	Description
	Go to Beginning – centers the beginning of the acquisition data.
	Go to Trigger – centers the trigger point of the acquisition.
	Go to End – centers the end of the acquisition data.

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**.

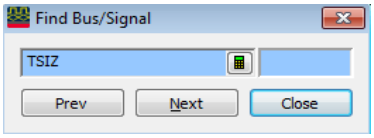
To re-arrange bus/signal columns

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.

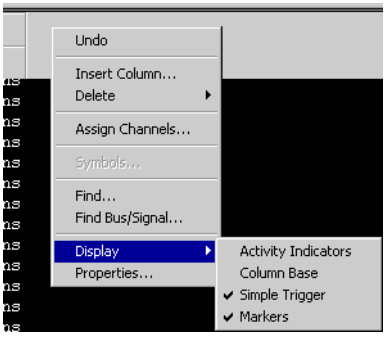
1 In the Listing display window, right-click in any Bus/Signal column heading, and choose **Find Bus/Signal...**.
2 In the Find Bus/Signal dialog, enter the name (or part of the name) of the bus/signal you wish to find.



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 132)).
- **Markers** – the markers display bar (see page 380).

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 215)

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**.

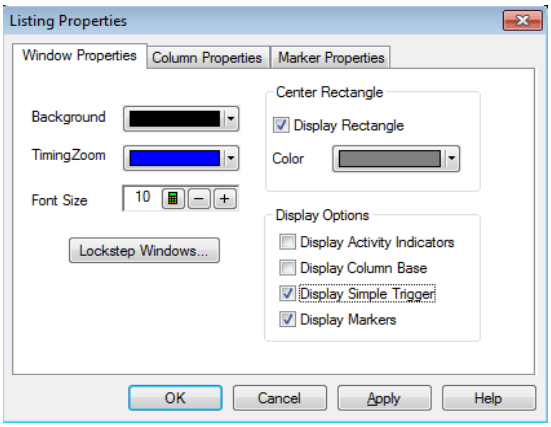
To delete all buses/signals
1. Right-click anywhere in the column headings row; then, choose **Delete>All Columns**.

See Also
- Defining Buses and Signals (see page 93)

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 216)
- Change the timing zoom background color (see page 216)
- Change the listing font size (see page 216)
- Lock scrolling with other display windows (see page 216)
- Show/hide the center rectangle (see page 216)
- Show/hide parts of the listing display (see page 214)

3. Click **OK** to apply the changes and close the Listing Properties dialog.

See Also
- Changing Bus/Signal Column Properties (see page 217)

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.

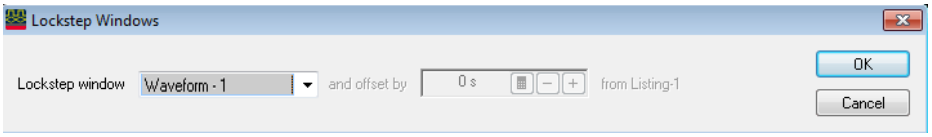
As the font size is changed, the column width may be automatically increased to fit the new text size.

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

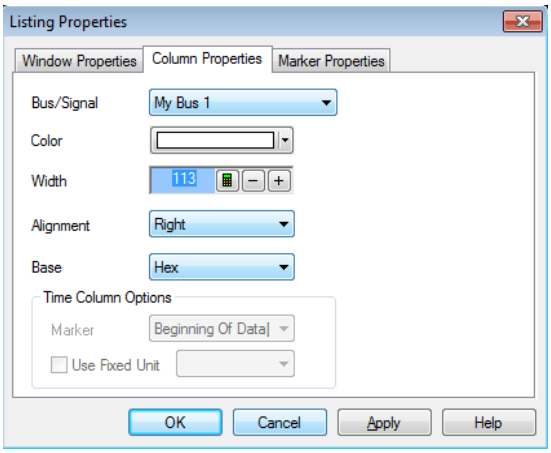
In the Listing display window, you can change the color, width, alignment, or number base of bus/signal data columns.

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 93)).
  - **<all>** buses/signals.
  - **<selected>** buses/signals if more than one column is highlighted.
- Change a bus/signal's data color (see page 217)
- Change the width of a bus/signal column (see page 218)
- Change the alignment of a bus/signal column (see page 218)
- Change a bus/signal's number base (see page 218)
- Select the marker for marker-relative times (see page 219)
- Select fixed time units (see page 219)

3 Click **OK** to apply the changes and close the Listing Properties dialog.

See Also
- Changing Listing Window Properties (see page 215)

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 220))

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" (in the online help)) column has been selected instead of a data column, **Voltage** is the only choice for number base.

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

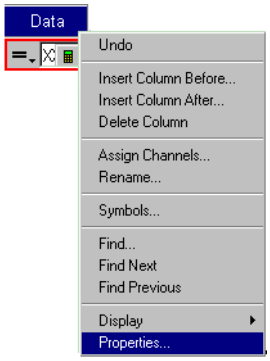
You can display a bus or signal using meaningful names rather than numeric values.

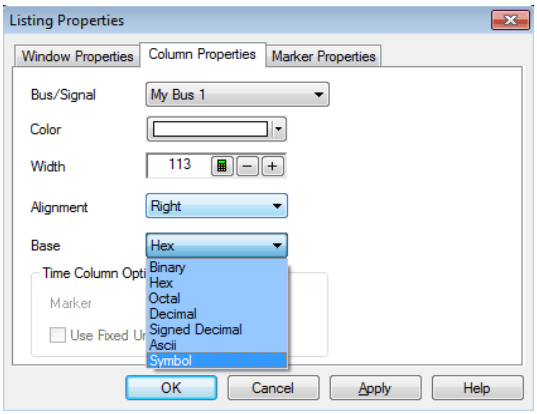
Symbols can be displayed in Waveform, Listing, Compare, and Source windows.

To display symbols:

1 Set up the symbols (see page 118).

2 Change the number base (see page 218) of the bus or signal to Symbols.





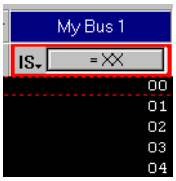
If the symbol is defined as a range, values in the range will be displayed with an offset from the lowest end of the range.

If the definitions of several symbols overlap, the first one listed in the Symbols dialog has precedence over the others.

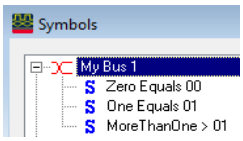
In the Waveform display, "..." will be shown when the full name of the symbol will not fit into the space available.

Once you have set up symbols, it's usually a good idea to save (see page 176) the logic analyzer configuration. The symbol definitions will be stored as part of the configuration.

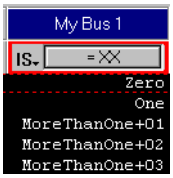
Example Here is what "My Bus 1" looks like before defining any symbols:



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

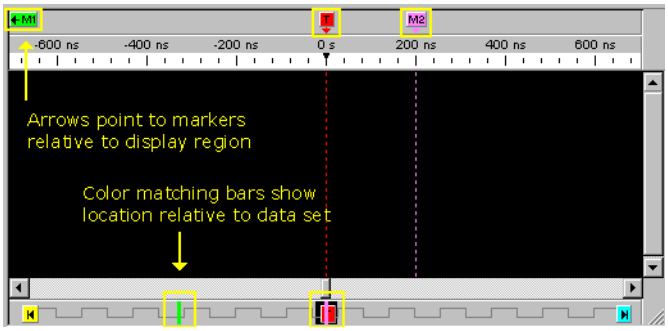
Once a marker is created, you can use it as a reference point in the data when measuring intervals or viewing the data value at the marker.

- To read the markers display and overview bars (see page 222)
- To create new markers (see page 223)
- To place markers in data (see page 224)
- To go to a marker (see page 226)
- To center the display about a marker pair (see page 226)
- To change a marker's snap to edge setting (see page 227)
- To delete a marker (see page 227)
- To create a new time interval measurement (see page 228)
- To create a new sample interval measurement (see page 229)
- To create a new value at measurement (see page 230)
- To rename a marker (see page 231)
- To send a marker to the back (see page 231)
- Changing Marker Properties (see page 232)
  - To change a marker's background color (see page 233)
  - To change a marker's foreground color (see page 233)
  - To hide/show a marker (see page 233)
  - To change a marker's lock in viewer setting (see page 233)
  - To lock a marker relative to another marker (see page 234)
  - To add comments to a marker (see page 234)
- Using Voltage Markers for Analog Signals (in the Waveform Display) (see page 234)
  - To create new voltage markers (see page 235)
  - To place voltage markers (see page 236)
  - To delete voltage markers (see page 237)
  - To create a new voltage interval measurement (see page 237)
  - To rename a voltage marker (see page 238)
  - To send a voltage marker to the back (see page 238)
  - To change voltage marker properties (see page 239)

See Also

- Markers Display Bar (see page 380)
- Marker Measurement Display Bar (see page 376)
- Markers Menu (see page 364)
- Markers Toolbar (see page 373)

To read the markers display and overview bars



In the upper markers display bar (see page 380), 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 381), 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.**

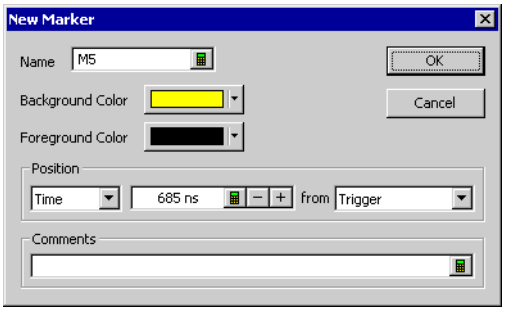
To create new markers

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 380) while the long name appears in the marker *tool tip* (see page 544).



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 240).

5 Enter comments for the marker.

  Comments appear in the marker's *tool tip* (see page 544).

6 Click **OK**.

See Also

- To place markers in data (see page 224)
- To go to a marker (see page 226)
- To read the markers display and overview bars (see page 222)
- To center the display about a marker pair (see page 226)
- To delete a marker (see page 227)
- To rename a marker (see page 231)
- To send a marker to the back (see page 231)
- To change a marker's snap to edge setting (see page 227)
- Changing Marker Properties (see page 232)

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 224)
- To place marker at the mouse cursor (see page 225)
- To place marker at center screen (see page 225)
- To change a marker's position property (see page 225)
- You can also place markers where data is found when searching (see To specify "found" marker placement (see page 248)).

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.

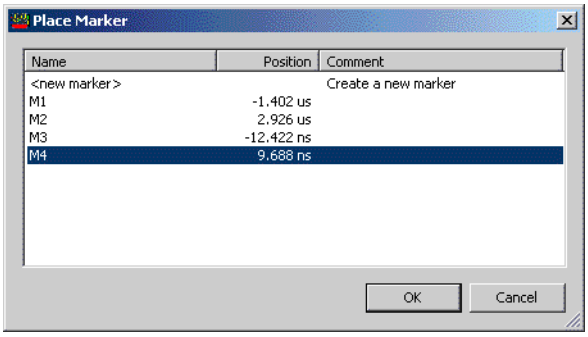
2 Right-click, and select **Place Marker>Time>(desired marker)**.

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 240), except you can click **Properties...** to open the Value Properties dialog. In 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 399), 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 248) (for placing markers where data is found)
- To create new markers (see page 223)
- To go to a marker (see page 226)

- To read the markers display and overview bars (see page 222)
- To center the display about a marker pair (see page 226)
- To delete a marker (see page 227)
- To rename a marker (see page 231)
- To send a marker to the back (see page 231)
- To change a marker's snap to edge setting (see page 227)
- Changing Marker Properties (see page 232)

To go to a marker

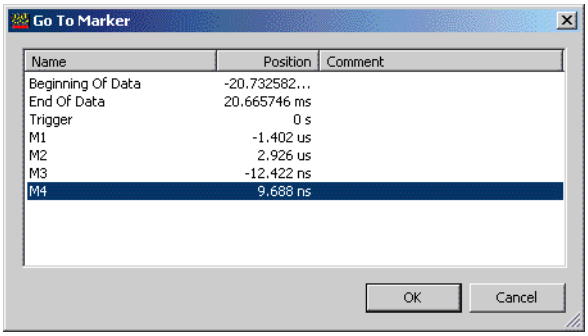
To quickly find a previously set marker in the data, or to go to the beginning of data, end of data, or the trigger point:

- Click in the markers display bar (see page 380) or the markers overview bar (see page 381), 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 373).

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 223)
- To place markers in data (see page 224)
- To center the display about a marker pair (see page 226)
- To delete a marker (see page 227)
- To rename a marker (see page 231)
- To send a marker to the back (see page 231)
- To change a marker's snap to edge setting (see page 227)
- Changing Marker Properties (see page 232)

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 223)
- To place markers in data (see page 224)
- To go to a marker (see page 226)
- To delete a marker (see page 227)
- To rename a marker (see page 231)
- To send a marker to the back (see page 231)
- To change a marker's snap to edge setting (see page 227)
- Changing Marker Properties (see page 232)

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 224)
- Changing Marker Properties (see page 232)

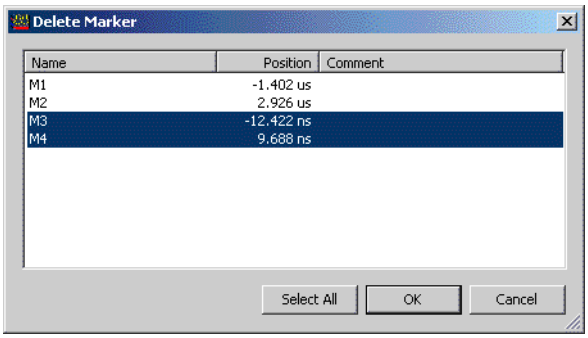
To delete a marker

- In the markers display bar (see page 380), 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 223)
- To place markers in data (see page 224)

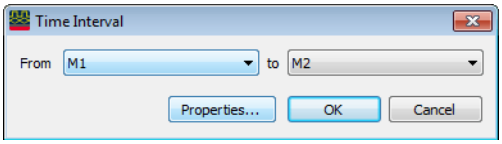
- To go to a marker (see page 226)
- To center the display about a marker pair (see page 226)
- To rename a marker (see page 231)
- To send a marker to the back (see page 231)
- To change a marker's snap to edge setting (see page 227)
- Changing Marker Properties (see page 232)

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 376).

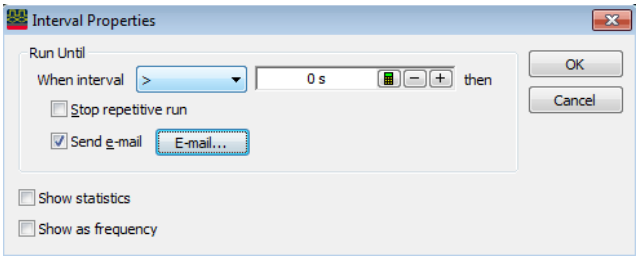
1 From the menu bar select **Markers>New Time Interval Measurement**, or click the icon in the markers toolbar (see page 373).

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 410). 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 399), 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 410). 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:

If statistics are shown, the low, high, and average interval measurements are included.

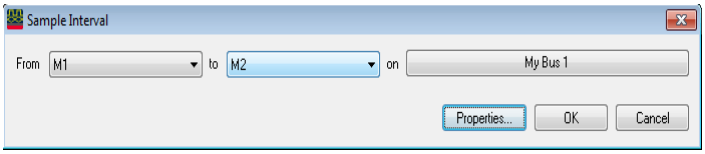
See Also
- To create a new sample interval measurement (see page 229)
- To create a new value at measurement (see page 230)

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 376).

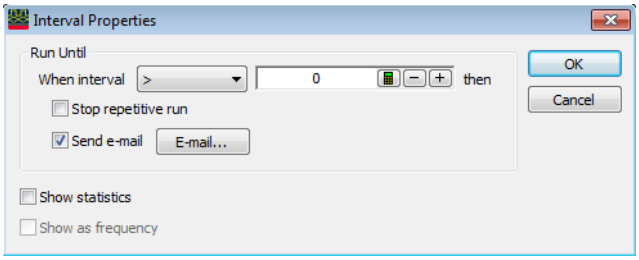
1 From the menu bar select **Markers>New Sample Interval Measurement**, or click the icon in the markers toolbar (see page 373).

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 410). 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 399), 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 410). 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:

If statistics are shown, the low and high interval measurements are included.

See Also

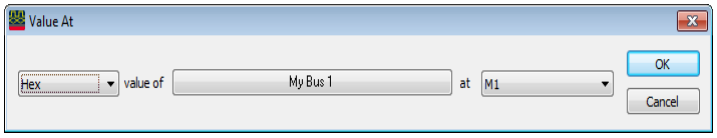
- To create a new time interval measurement (see page 228)
- To create a new value at measurement (see page 230)

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 376).

1 From the menu bar select **Markers>New Value At Measurement**, or click the icon in the markers toolbar (see page 373).

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 410). 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

is displayed in the marker measurement display bar.

See Also
- To create a new time interval measurement (see page 228)
- To create a new sample interval measurement (see page 229)

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 380) while the long name appears in the marker *tool tip* (see page 544).

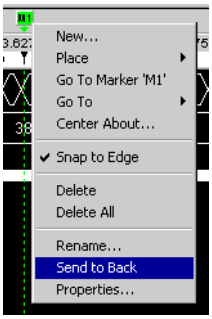
3 Click **OK**.

See Also
- To create new markers (see page 223)
- To place markers in data (see page 224)
- To go to a marker (see page 226)
- To center the display about a marker pair (see page 226)
- To delete a marker (see page 227)
- To send a marker to the back (see page 231)
- To change a marker's snap to edge setting (see page 227)
- Changing Marker Properties (see page 232)

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 223)
- To place markers in data (see page 224)
- To go to a marker (see page 226)
- To center the display about a marker pair (see page 226)
- To delete a marker (see page 227)
- To rename a marker (see page 231)
- To change a marker's snap to edge setting (see page 227)
- Changing Marker Properties (see page 232)

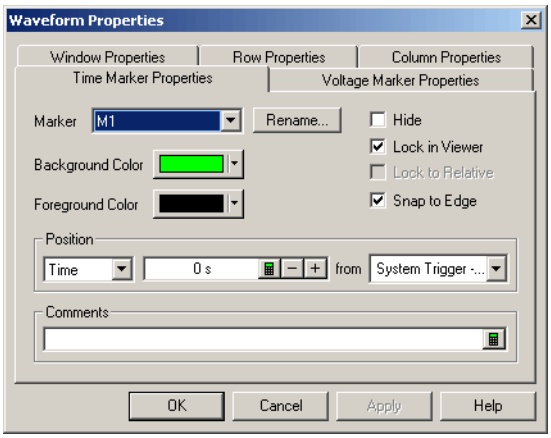
Changing Marker Properties

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 231)
- Change a marker's background color (see page 233)
- Change a marker's foreground color (see page 233)
- Change a marker's position property (see page 225)
- Hide/show a marker (see page 233)
- Change a marker's lock in viewer setting (see page 233)
- Lock a marker relative to another marker (see page 234)
- Change a marker's snap to edge setting (see page 227)
- Add comments to a marker (see page 234)

3 Click **OK** to apply the changes and close the properties dialog.

See Also
- To create new markers (see page 223)
- To place markers in data (see page 224)
- To go to a marker (see page 226)
- To center the display about a marker pair (see page 226)
- To delete a marker (see page 227)
- To rename a marker (see page 231)
- To send a marker to the back (see page 231)
- To read the markers display and overview bars (see page 222)

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 voltage marker properties (see page 239)

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 voltage marker properties (see page 239)

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 voltage marker properties (see page 239)

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

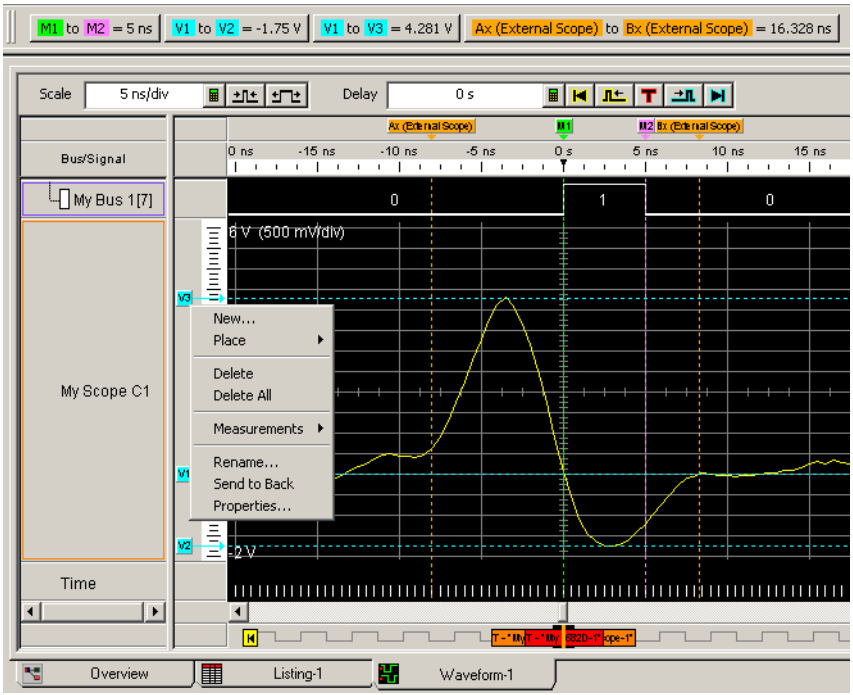
You can add comments to a marker that appear in the marker's *tool tip* (see page 544).

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 voltage marker properties (see page 239)

Using Voltage 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 voltage markers and voltage interval measurements.



- To create new voltage markers (see page 235)
- To place voltage markers (see page 236)
- To delete voltage markers (see page 237)
- To create a new voltage interval measurement (see page 237)
- To rename a voltage marker (see page 238)
- To send a voltage marker to the back (see page 238)
- To change voltage marker properties (see page 239)

See Also
- " External Oscilloscope Time Correlation and Data Display" (in the online help)

To create new voltage markers

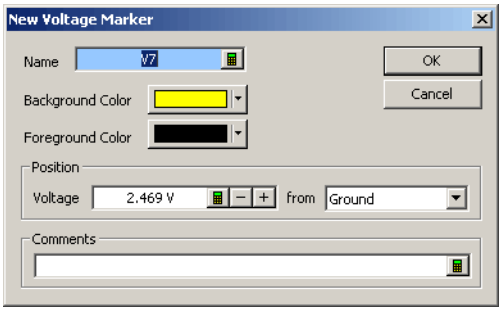
When creating a new voltage marker, you can give it a name, specify its color, position it, and add comments.

1 In an analog signal row's voltage 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>Voltage>New Marker...**.

2 In the New Voltage 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 voltage marker display bar while the long name appears in the marker *tool tip* (see page 544).



3 Select the marker's background and foreground colors.

4 Specify the position of the new voltage marker by its voltage from ground or another voltage marker.

5 Enter comments for the voltage marker.

Comments appear in the marker's *tool tip* (see page 544).

6 Click **OK**.

See Also

- To place voltage markers (see page 236)
- To delete voltage markers (see page 237)
- To create a new voltage interval measurement (see page 237)
- To rename a voltage marker (see page 238)
- To send a voltage marker to the back (see page 238)
- To change voltage marker properties (see page 239)

To place voltage markers

Use Place Markers to quickly position a voltage 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 voltage markers intersect time markers, you can drag both markers at the same time.

- To drag and drop voltage markers (see page 236)
- To place a voltage marker at the mouse cursor (see page 236)
- To change a voltage marker's position property (see page 236)

To drag and drop voltage markers

Using the drag and drop feature you can move voltage 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 voltage marker at the mouse cursor

1 Point the mouse to the desired data point in the display.

2 Right-click, and select **Place Marker>Voltage>(desired marker)**.

To change a voltage marker's position property

1 Right-click on a voltage marker, and choose **Properties...**.

Or, when viewing a display window that has voltage markers, choose **Markers>Properties...** from the main menu.

2 In the Waveform Properties dialog's Voltage Marker Properties tab, select the **Marker** to which the property changes apply.

3 In the **Position** box, position the marker by its voltage from ground or another voltage marker.

4 Click **OK** to apply the changes and close the properties dialog.

See Also
- To create new voltage markers (see page 235)
- To delete voltage markers (see page 237)
- To create a new voltage interval measurement (see page 237)
- To rename a voltage marker (see page 238)
- To send a voltage marker to the back (see page 238)
- To change voltage marker properties (see page 239)

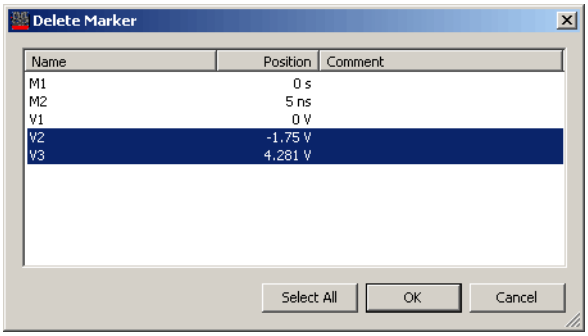
To delete voltage markers

- In an analog signal's voltage markers/vertical scale display bar (to the left of the waveform), click the voltage marker you want to delete, and choose **Delete** from the popup menu (or choose **Delete All** to delete all voltage 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 voltage markers (see page 235)
- To place voltage markers (see page 236)
- To create a new voltage interval measurement (see page 237)
- To rename a voltage marker (see page 238)
- To send a voltage marker to the back (see page 238)
- To change voltage marker properties (see page 239)

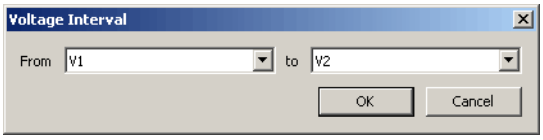
To create a new voltage interval measurement

Use the voltage interval measurement feature to measure a voltage between two voltage markers. Measurement results are displayed in the marker measurement display bar (see page 376).

1 In an analog signal row's voltage marker/vertical scale display bar (to the left of the waveform), click and choose **Measurements>New Voltage Interval Measurement**.

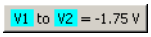
Or, right-click in the marker measurement display bar and choose **New Voltage Interval Measurement>(analog signal name)**.

2 In the Voltage Interval dialog, select the markers you want to measure voltage between.



3 Click **OK** to close the Voltage Interval dialog.

The result of the interval measurement is displayed in the marker measurements display bar:



To rename a voltage marker

You can give voltage 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 Voltage 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 voltage marker display bar while the long name appears in the marker *tool tip* (see page 544).

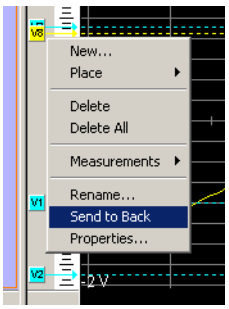
3 Click **OK**.

See Also
- To create new voltage markers (see page 235)
- To place voltage markers (see page 236)
- To delete voltage markers (see page 237)
- To create a new voltage interval measurement (see page 237)
- To send a voltage marker to the back (see page 238)
- To change voltage marker properties (see page 239)

To send a voltage marker to the back

When voltage markers overlap on the display, you can send the visible marker to the back in order to see the marker underneath.

1 Click the voltage 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 235)

- To place voltage markers (see page 236)
- To delete voltage markers (see page 237)
- To create a new voltage interval measurement (see page 237)
- To rename a voltage marker (see page 238)
- To change voltage marker properties (see page 239)

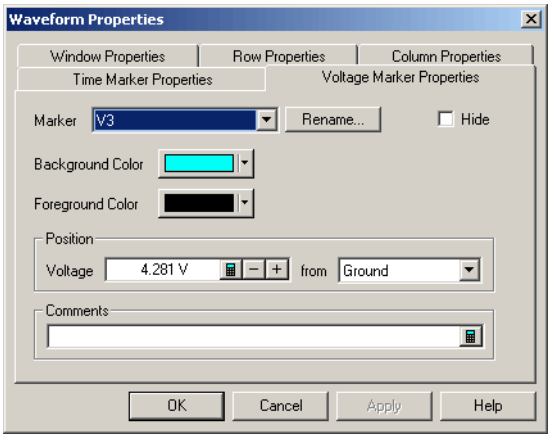
To change voltage marker properties

Once a voltage marker is created, you can modify any of its attributes from the Voltage Marker Properties tab..

1 Right-click on a voltage marker, and choose **Properties...**.

Or, when viewing a Waveform display window that has voltage markers, choose **Markers>Properties...** from the main menu.

2 In the Waveform Properties dialog's Voltage Marker Properties tab:



You can:

- Select the **Marker** to which the property changes apply.
- Rename a voltage marker (see page 238)
- Change a marker's background color (see page 233)
- Change a marker's foreground color (see page 233)
- Change a voltage marker's position property (see page 236)
- Hide/show a marker (see page 233)
- Add comments to a marker (see page 234)

3 Click **OK** to apply the changes and close the properties dialog.

See Also

- To create new voltage markers (see page 235)
- To place voltage markers (see page 236)
- To delete voltage markers (see page 237)
- To create a new voltage interval measurement (see page 237)
- To rename a voltage marker (see page 238)
- To send a voltage marker to the back (see page 238)

Searching the Captured Data

You can search for bus/signal patterns in the captured data.

- To quickly find bus/signal patterns (see page 240)
- To find bus/signal patterns in the captured data (see page 241)
- To find packet patterns in the captured data (see page 243)
- To find complex patterns in the captured data (see page 246)
- To store, recall, or delete favorite find patterns (see page 247)
- To specify "found" marker placement (see page 248)

To quickly find bus/signal patterns

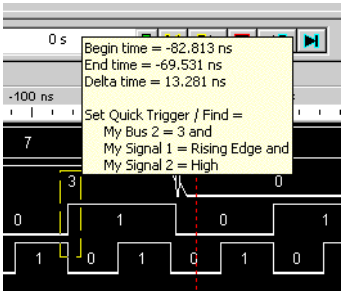
In the Waveform or Listing windows, you can quickly draw a rectangle and find the next or previous occurrence of that bus/signal pattern.

1 In the Waveform window, make sure the Fast Zoom In (see page 207) 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:

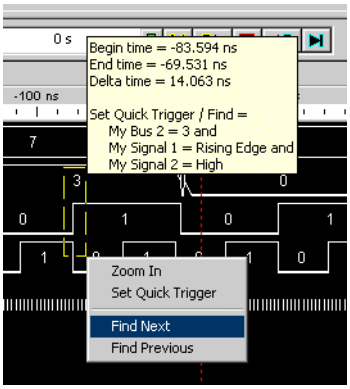
a If drawing starts on a bus, none of its expanded signals can be included.

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

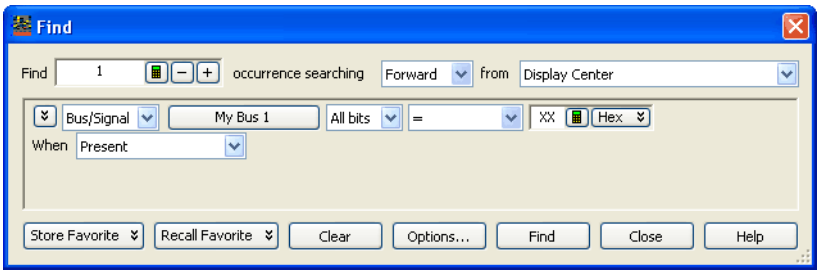
- To find bus/signal patterns in the captured data (see page 241)
- To find packet patterns in the captured data (see page 243)
- To find complex patterns in the captured data (see page 246)
- To store, recall, or delete favorite find patterns (see page 247)

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 371).

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 240)
- To find packet patterns in the captured data (see page 243)
- To find complex patterns in the captured data (see page 246)

- To store, recall, or delete favorite find patterns (see page 247)
- To specify "found" marker placement (see page 248)

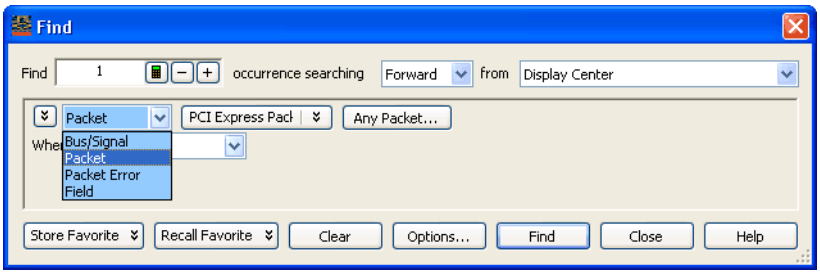
To find packet patterns in the captured data

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 371).

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.

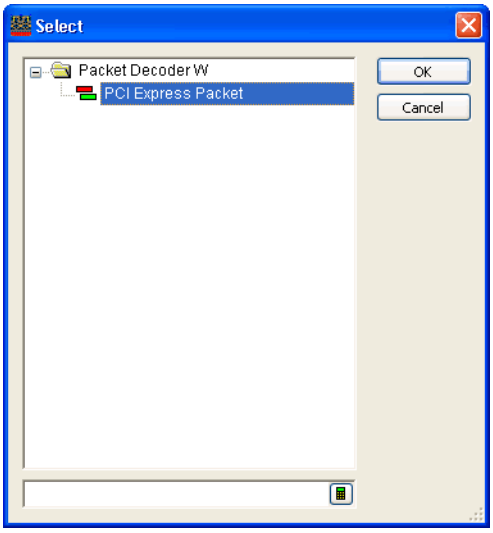
3 In the event specification area, select the **Packet**, **Packet Error**, or **Field** pattern type.



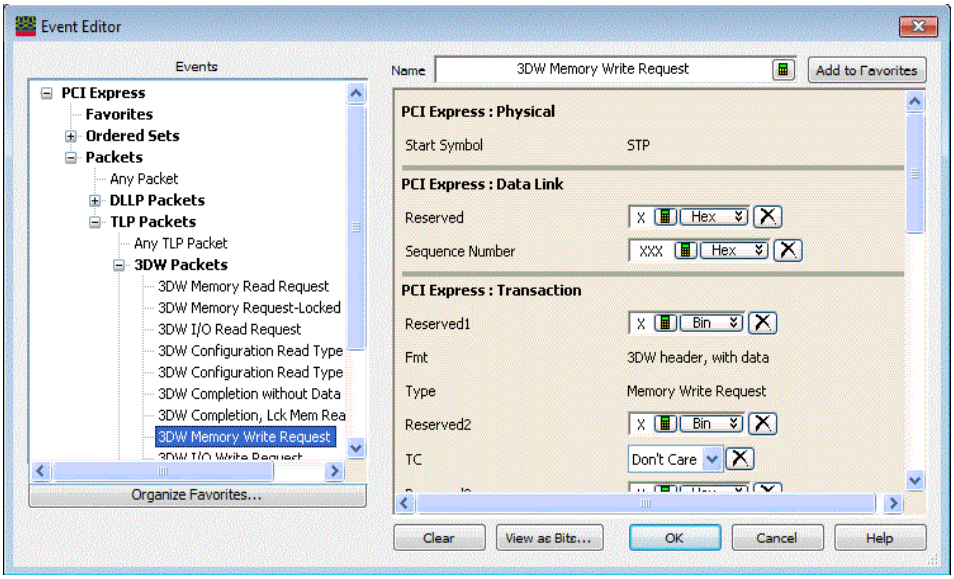
- To find packet events (see page 243)
- To find packet errors (see page 244)
- To find field values (see page 245)

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 148)

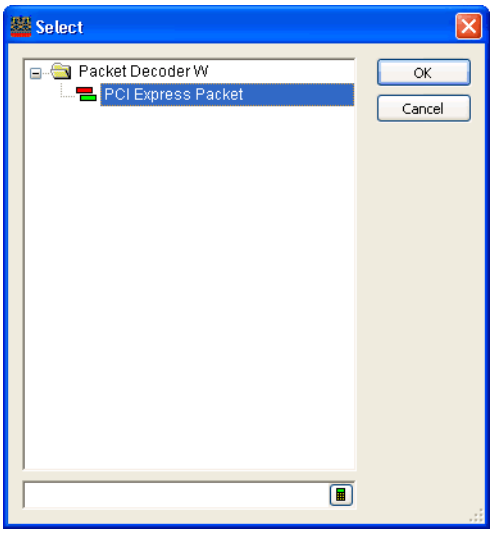
3 Select the desired **When** find qualifier (which further qualifies the find criteria with a time duration or other operator). For more information, see To find bus/signal patterns in the captured data (see page 241).

4 Click **Find**.

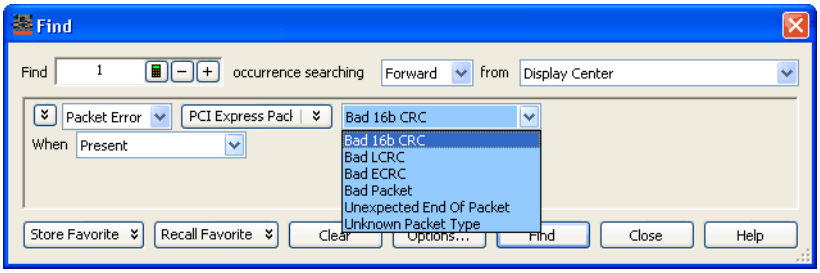
5 Click the Previous or Next icons to see more occurrences.

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 241).

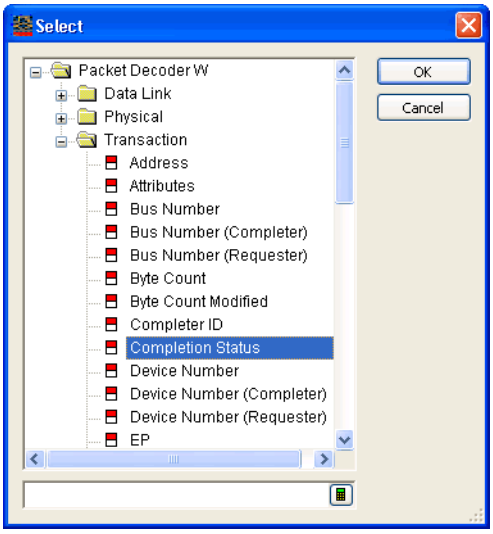
4 Click **Find**.

5 Click the Previous or Next icons to see more occurrences.

To find field values

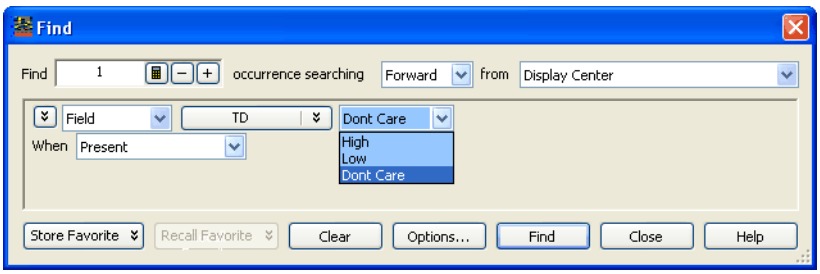
1 Select the field name.

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.



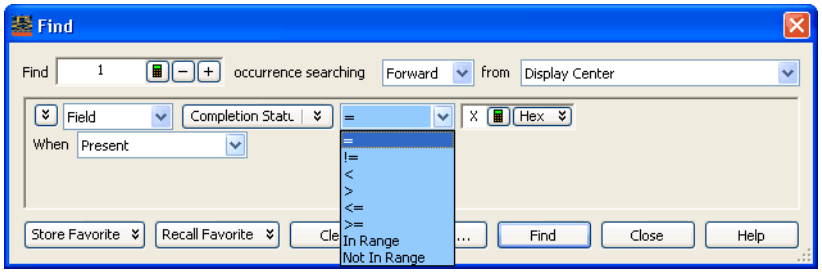
2 Specify the field value:

If a single-bit field has been selected, select the signal pattern value (**High**, **Low**, or **Dont Care**).

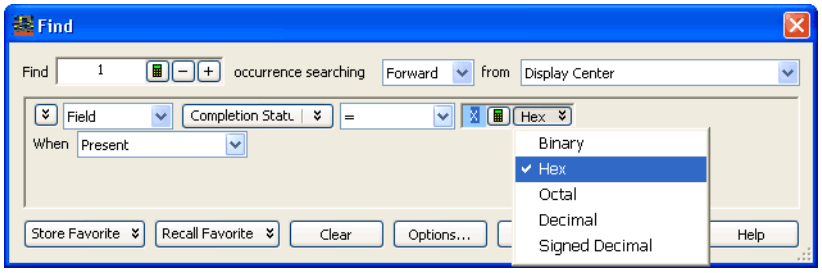


If a multiple-bit field has been selected:

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 241).

4 Click **Find**.

5 Click the Previous or Next icons to see more occurrences.

See Also
- To quickly find bus/signal patterns (see page 240)
- To find bus/signal patterns in the captured data (see page 241)
- To find complex patterns in the captured data (see page 246)
- To store, recall, or delete favorite find patterns (see page 247)
- To specify "found" marker placement (see page 248)

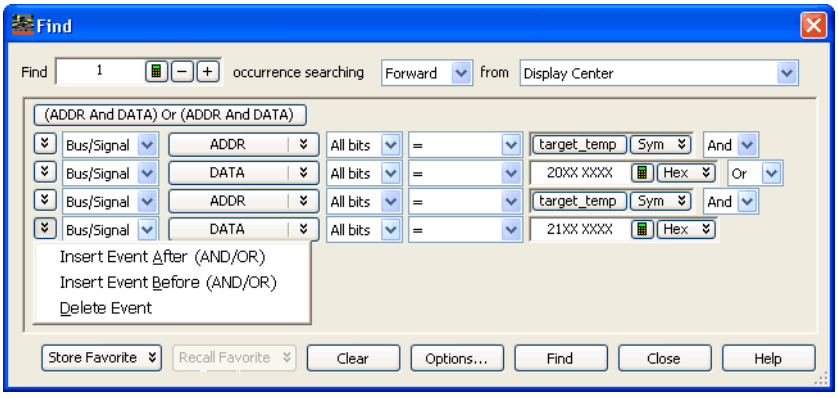
To find complex patterns in the captured data

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 371).

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 240)
- To find bus/signal patterns in the captured data (see page 241)
- To find packet patterns in the captured data (see page 243)
- To store, recall, or delete favorite find patterns (see page 247)
- To specify "found" marker placement (see page 248)

To store, recall, or delete favorite find patterns

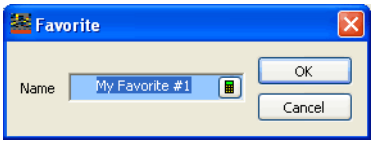
- To store a favorite pattern (see page 247)
- To recall a favorite pattern (see page 247)
- To delete a favorite pattern (see page 248)

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 241) or To find complex bus/signal patterns in the captured data (see page 246)).

3 Click **Store Favorite**.



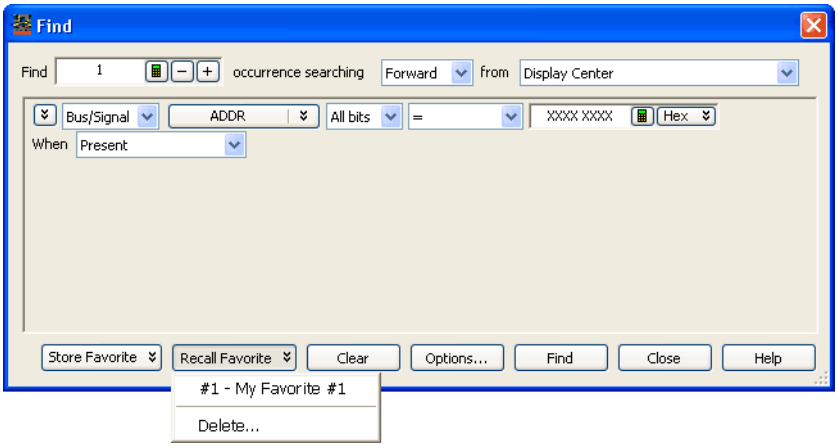
4 Enter the name of the find pattern.

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**.

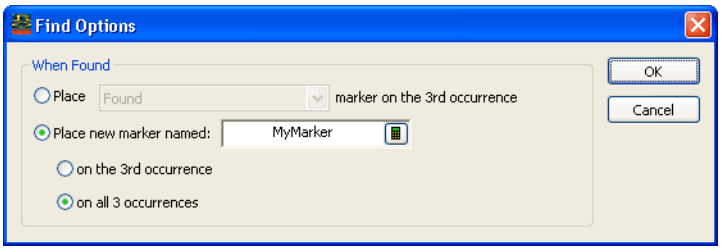
See Also

- Searching the Captured Data (see page 240)
- To specify "found" marker placement (see page 248)

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 371).
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 240)
- To store, recall, or delete favorite find patterns (see page 247)
- To place markers in data (see page 224)

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 250)
- To find differences in the compared data (see page 250)
- To compare only a range of samples (see page 251)
- To offset the reference data (see page 251)
- To run until a number of compare differences (see page 251)
- To set Compare window properties (see page 252)

See Also
- Compare Display Window (see page 384)
- Capturing Data from the Device Under Test (see page 127)
- Loading Saved Data and Setups (see page 188)

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).

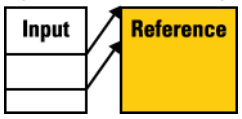
- Click a blue tick mark in the **Compare Overview** bar (between the vertical scroll bar and the Marker Overview bar on the right side of the window) to go to that difference.

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 251) 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 251)

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 399), 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 174)

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 215)
- To run until a number of compare differences (see page 251)

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 119)).

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**.

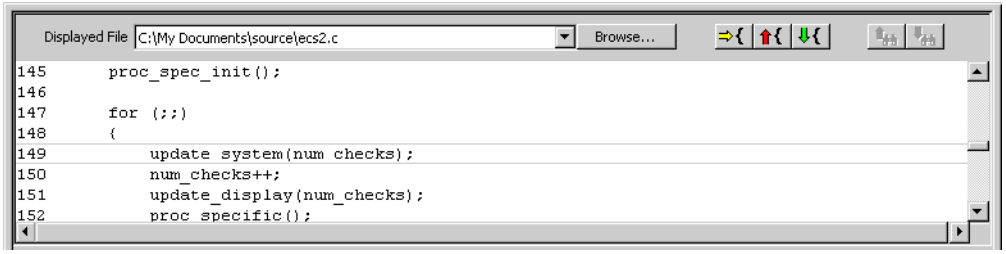
8 Click **OK** to close the Source Viewer Properties dialog.

For more information on viewing the source code associated with captured data, see:

- To step through captured data by source lines (see page 253)
- To go to captured data associated with a source line (see page 254)
- To browse source files (see page 254)
- To search for text in source files (see page 255)
- To set a Quick Trigger in the Source window (see page 131)
- To go to the source line associated with the listing center rectangle (see page 255)
- To edit the source code directory list (see page 255)
- To select the correlation bus (see page 256)
- Changing Source Window Properties (see page 257)
  - To change the source background color (see page 258)
  - To change the source text color (see page 258)
  - To change the source font size (see page 258)
  - To change the source tab width (see page 258)
  - To show/hide source line numbers (see page 258)
  - To change the "Set Quick Trigger" alignment (see page 258)

See Also
- Analyzing Listing Data (see page 212)
- Source Display Window (see page 385)

To step through captured data by source lines

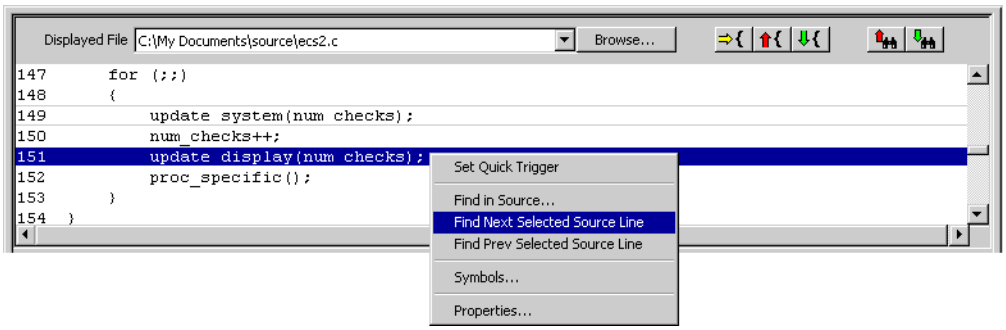


1 In the Source window's source pane, click the step to next source line or step to previous source line buttons.

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 253)

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 253)

To browse source files

1 In the Source window's source pane, click **Browse...**.

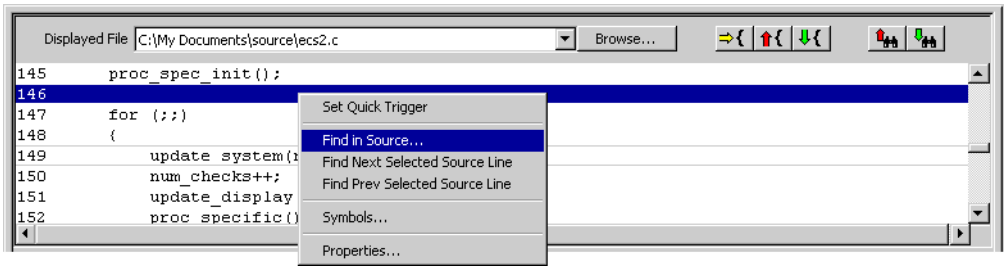
2 In the Select Source File to Open dialog, select the source file to browse, and click **Open**.

The selected source file appears in the source pane.

See Also
- To search for text in source files (see page 255)
- To set a Quick Trigger in the Source window (see page 131)
- To go to the source line associated with the listing center rectangle (see page 255)
- Viewing Source Code Associated with Captured Data (see page 253)

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**.

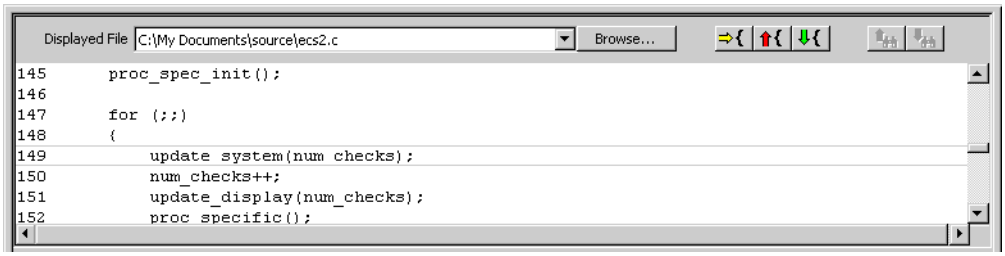
If the text is found, the source line is highlighted.

If the text is not found, an information dialog is displayed.

See Also
- To set a Quick Trigger in the Source window (see page 131)
- To go to the source line associated with the listing center rectangle (see page 255)
- Viewing Source Code Associated with Captured Data (see page 253)

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 253)

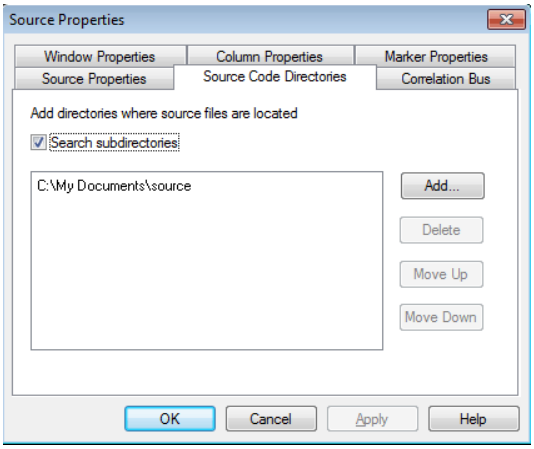
To edit the source code directory list

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.

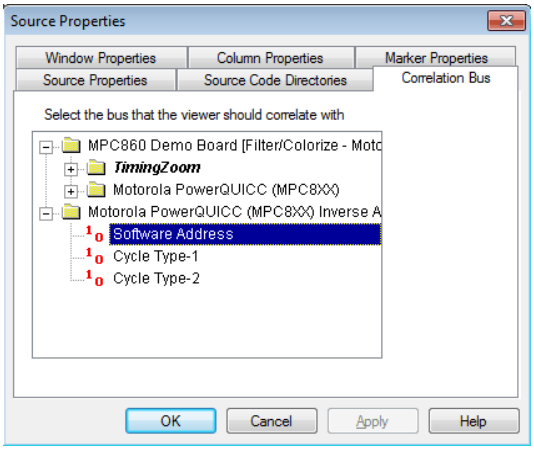
See Also
- Viewing Source Code Associated with Captured Data (see page 253)

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 253)

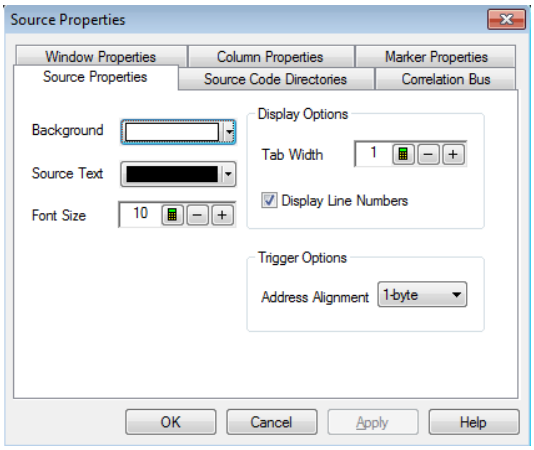
Changing Source Window Properties

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 258)
- Change the source text color (see page 258)
- Change the source font size (see page 258)
- Change the source tab width (see page 258)
- Show/hide source line numbers (see page 258)
- Change the "Set Quick Trigger" alignment (see page 258)

4 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

See Also
- Changing Listing Window Properties (see page 215)
- Changing Bus/Signal Column Properties (see page 217)

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.

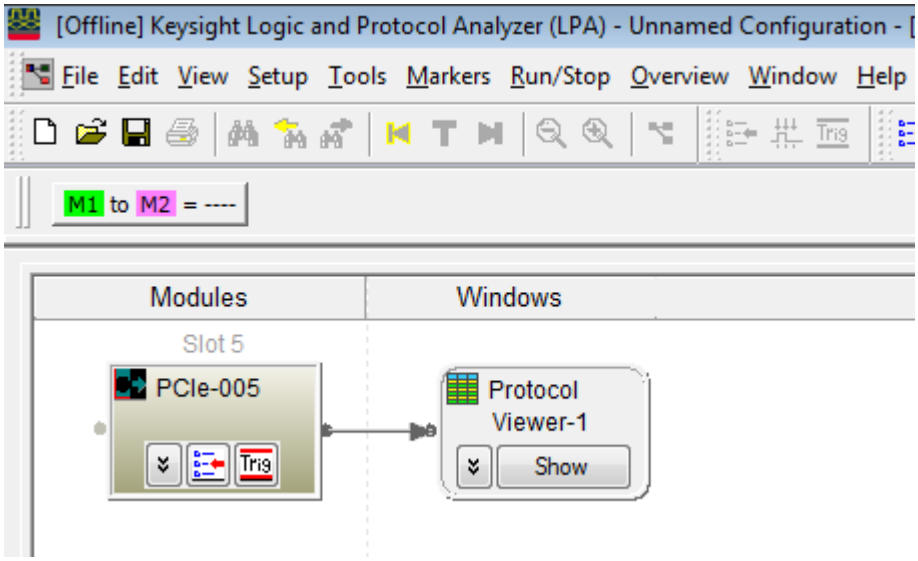
1 In the Source Viewer Properties dialog's Source Properties tab, select the desired **Address Alignment**.

2 Click **OK** to apply the changes and close the Source Viewer Properties dialog.

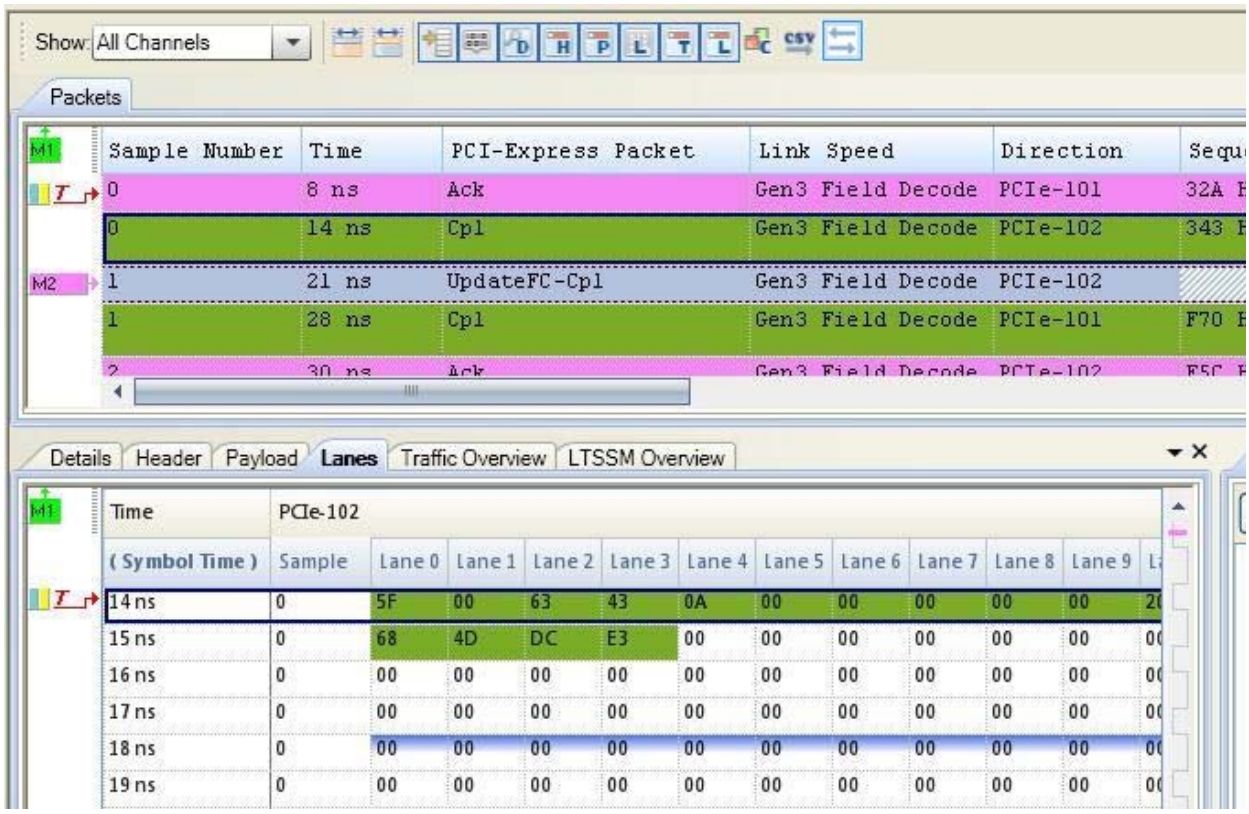
See Also
- To set a Quick Trigger in the Source window (see page 131)

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 289) 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 261)
  - To go to different locations in the displayed data (see page 262)
  - To re-arrange packet columns (see page 263)
  - To insert or delete packet columns (see page 264)
  - To show/hide parts of the packet summary display (see page 268)
  - To lockstep Protocol Viewer with other display windows (see page 269)
- Viewing a Selected Packet (see page 269)
  - To view and compare packet details (see page 270)
  - To view a packet header (see page 273)
  - To view a packet payload (see page 274)
  - To view a packet's lanes (see page 275)
  - To show/hide Protocol Viewer panes (see page 277)

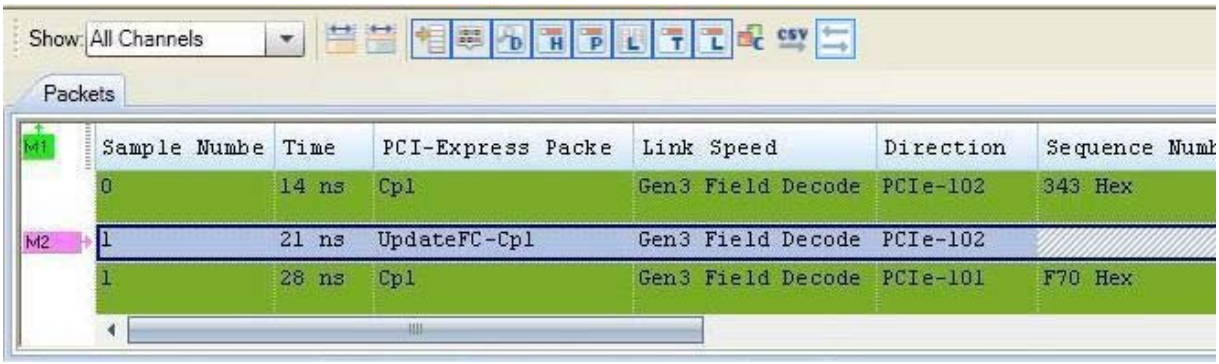
- Changing Packet Summary Event Colors (see page 277)
- Changing Protocol Viewer Window Properties (see page 278)
  - To change the selected row box color (see page 279)
  - To change the Protocol Viewer font size (see page 279)
  - To lock scrolling with other display windows (see page 279)
- Changing Packet Summary Column Properties (see page 280)
  - To change the directions of a packet column (see page 281)
  - To change the width of a packet column (see page 281)
  - To change the alignment of a packet column (see page 281)
  - To change a packet column's number base (see page 282)
  - To select the marker for marker-relative times (see page 282)
  - To select fixed time units (see page 282)
- 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 148)
- To find packet patterns in the captured data (see page 243)
- "To specify packet patterns to filter" (in the online help)
- Marking, and Measuring Between, Data Points (see page 222)
- Displaying Names (Symbols) for Packet Summary Column Values (see page 220)
- Searching the Captured Data (see page 240)

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.



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 262

- "To re-arrange packet columns" on page 263
- "To insert or delete packet columns" on page 264
- "To show/hide parts of the packet summary display" on page 268
- To lockstep Protocol Viewer with other display windows (page 269)

See Also

- "Changing Packet Summary Event Colors" on page 277
- "Changing Protocol Viewer Window Properties" on page 278
- "Changing Packet Summary Column Properties" on page 280
- "Viewing a Selected Packet" on page 269
- "Marking, and Measuring Between, Data Points" on page 222
- "Searching the Captured Data" on page 240
- 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.

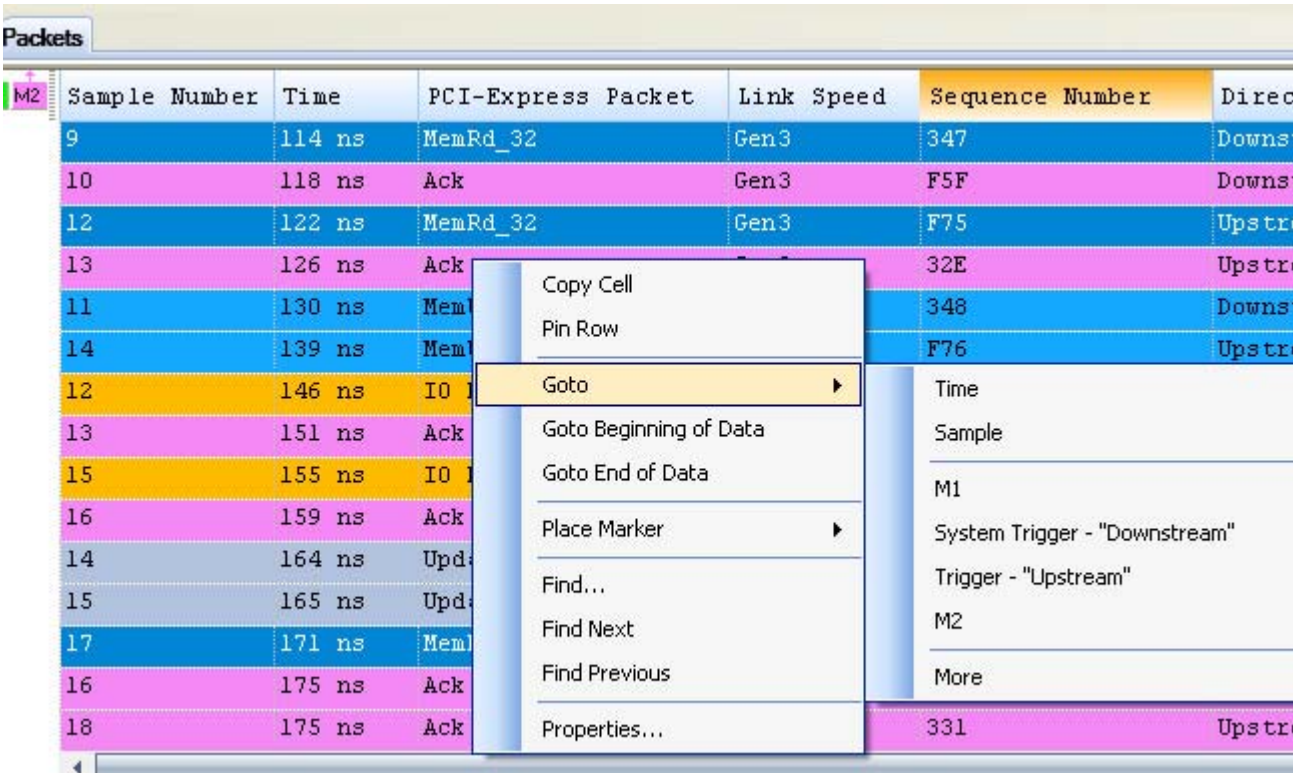
To go to different locations using toolbar buttons

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.

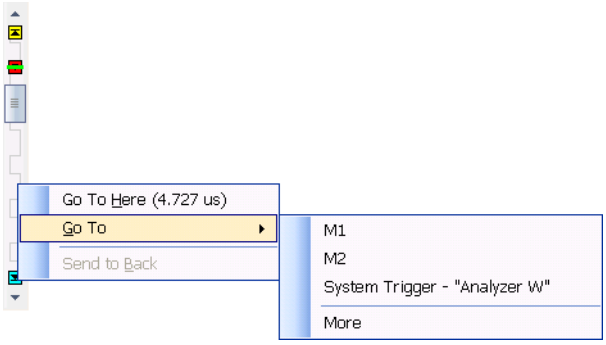
To go to different locations using popup/context menus

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

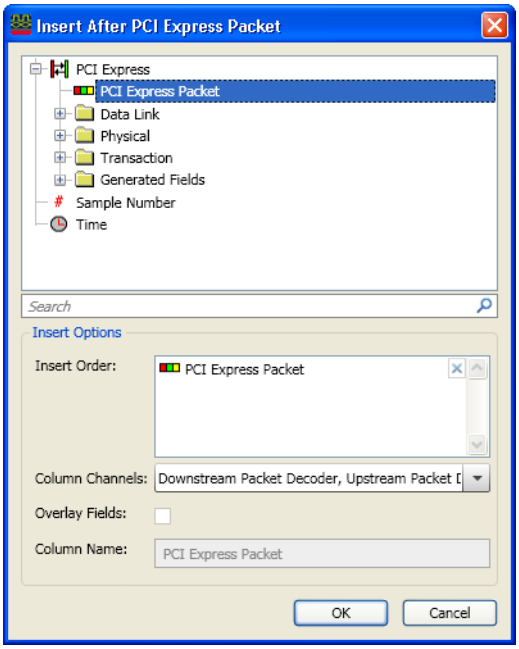
1 Position the mouse pointer over the packet field column you want to move.
2 Click and hold the mouse button.
3 Drag-and-drop the packet decode column to its new position.
The column is placed to the left of the red position indicator that appears.

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 264.
- 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:

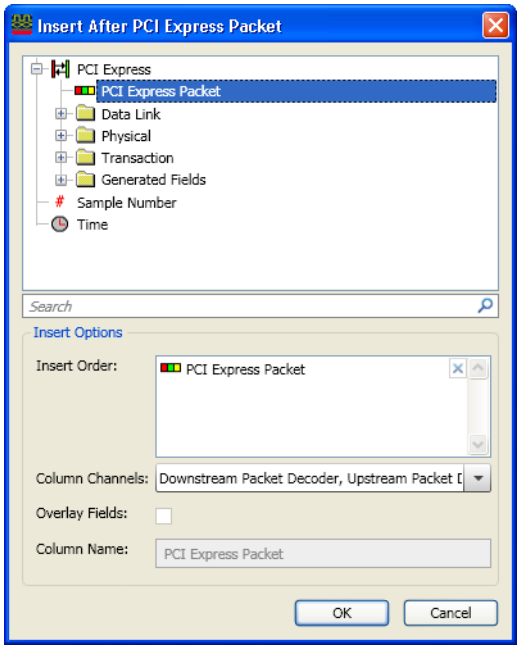


Figure 6 The Insert column dialog from the Protocol Viewer context menu

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:

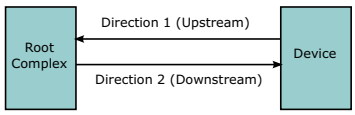


Figure 7 Two directions of PCI Express traffic

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:

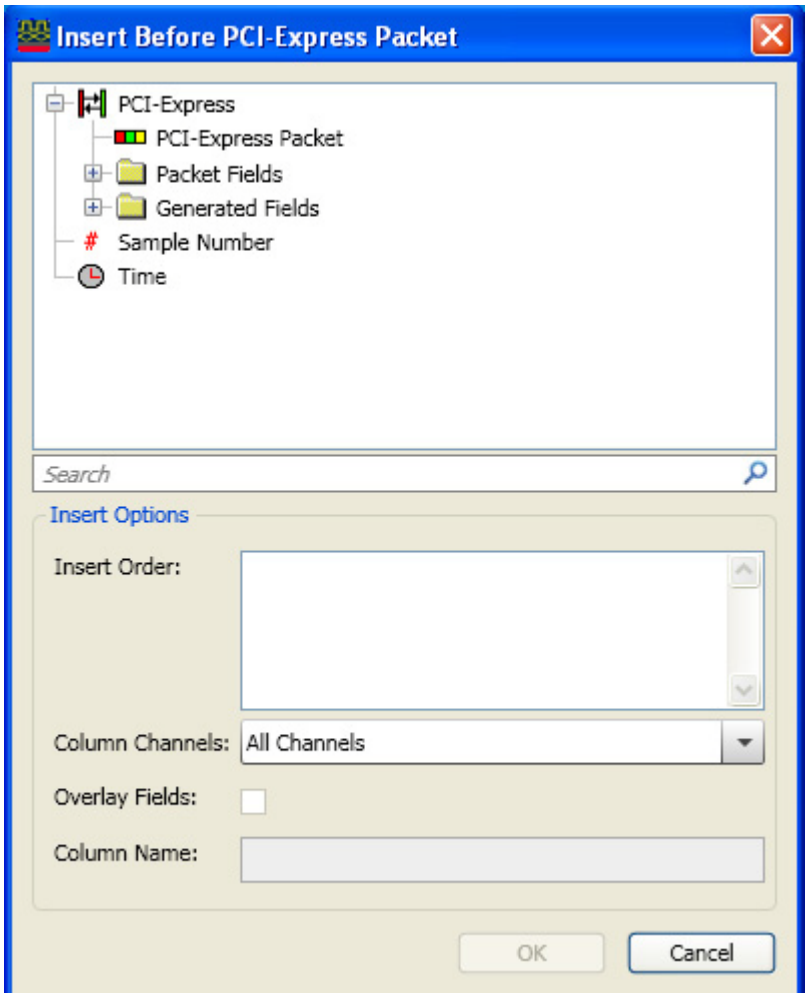


Figure 8 Insert a PCI Express Packet column for both directions

For example, if you specify that a new PCI Express Packet column should show data for All Channels (directions), it looks like:

Packets

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 (Gen…	Downstream
14	139 ns	Memory Write 32b (Gen…	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:

Packets

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	IO Rd		Downstream
13	151 ns	Ack		Downstream
15	155 ns		IO Rd	Upstream
16	159 ns		Ack	Upstream

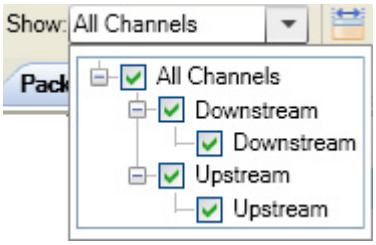
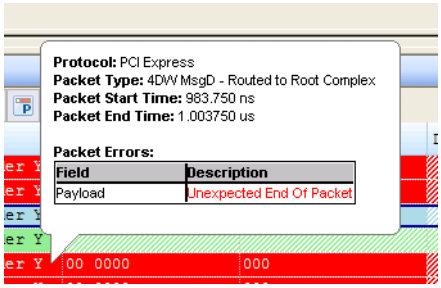
Figure 10 Two PCI Express Packet columns that are each dedicated to a unique direction

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 264.
	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 277).

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 278

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.

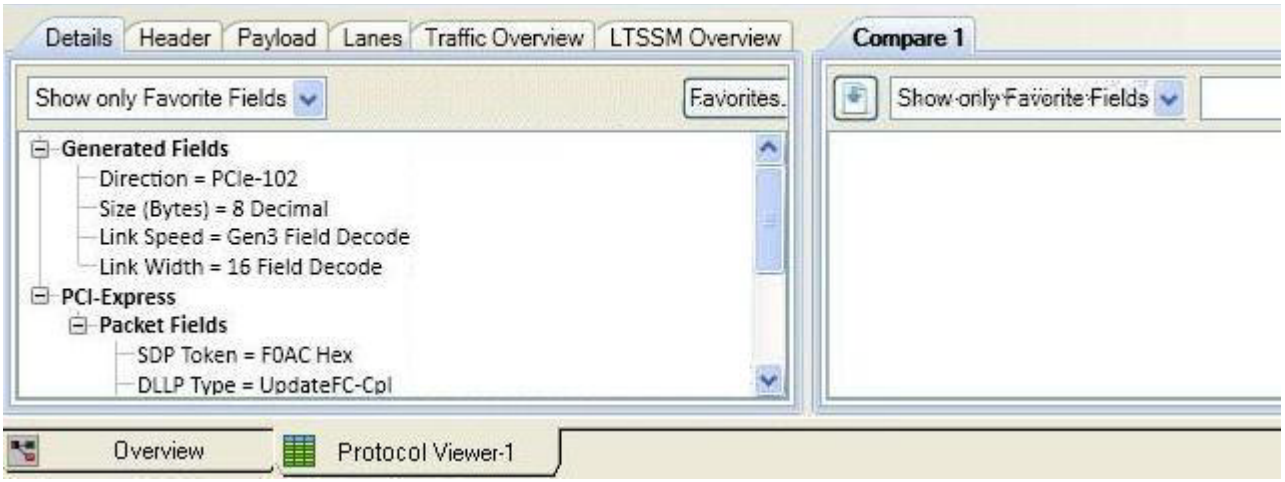
2 In the **Lockstep Windows** dialog, select the display window(s) whose scrolling should be locked with this window and specify any offset from this window.

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 270
- "To view a packet header" on page 273
- "To view a packet payload" on page 274
- "To view a packet's lanes" on page 275
- "To view LTSSM States and Transitions" on page 276
- "To show/hide Protocol Viewer panes" on page 277

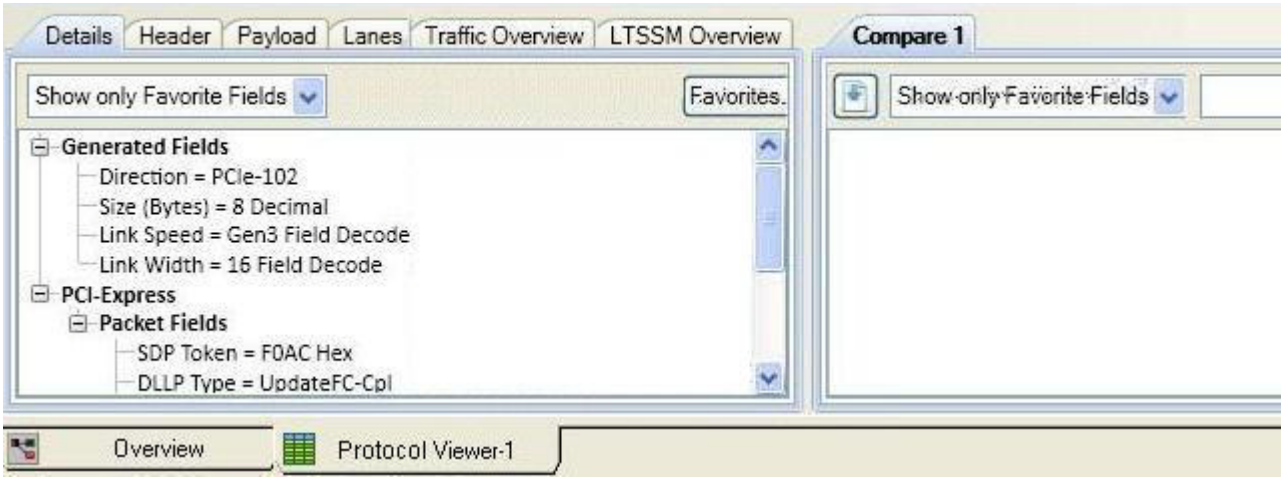
See Also

- "Viewing the Packet Summary" on page 261

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.

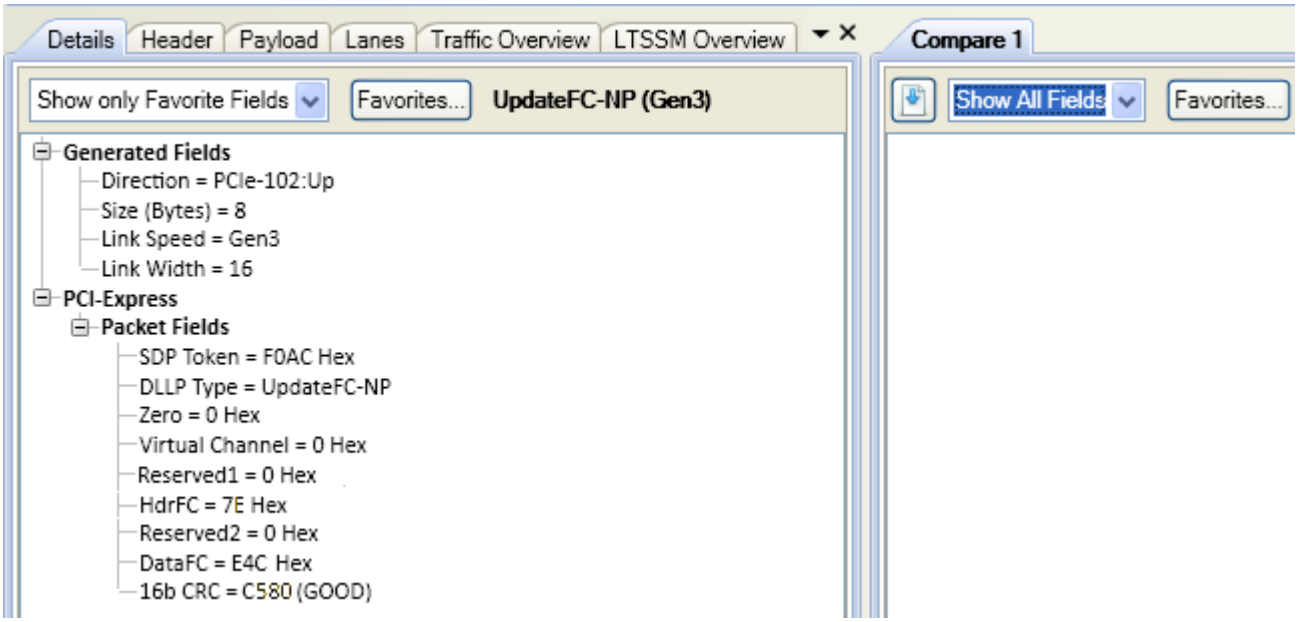
If you hold the mouse pointer motionless for a second over one of the packets, a *tool tip* (that is, a small box with text) appears with more information.

3 You can expand or collapse the displayed information by clicking "+" or "-" in the packet hierarchy tree.

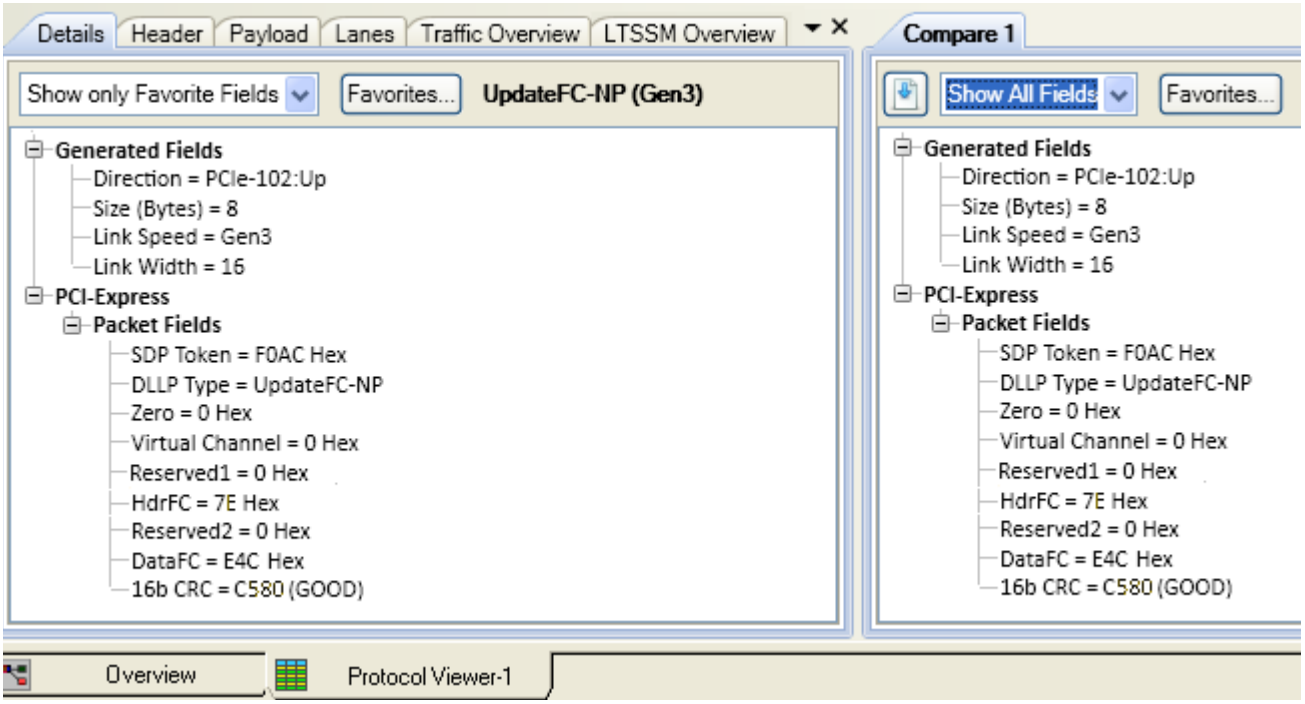
To compare packet details

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.

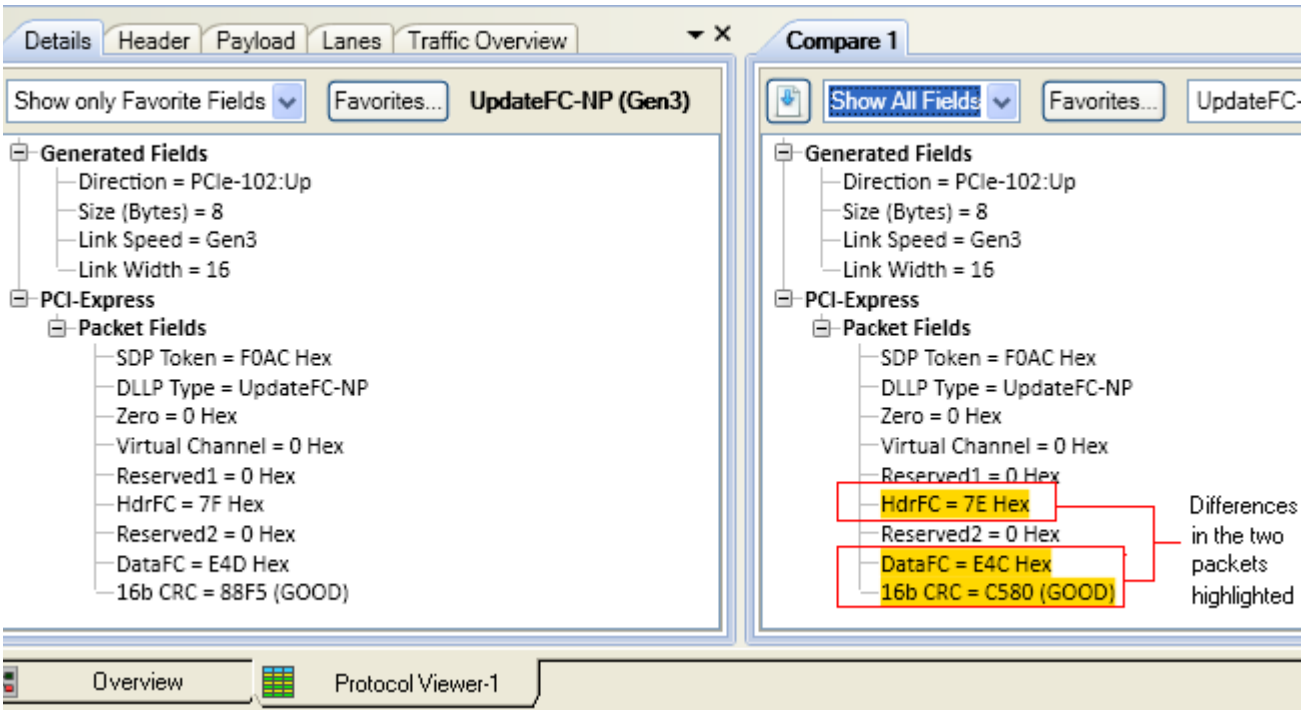


2 Click the button displayed in the Compare buffer in the lower-right pane. Clicking this button copies the details of the packet currently displayed in the lower-left pane to the compare buffer at the right.



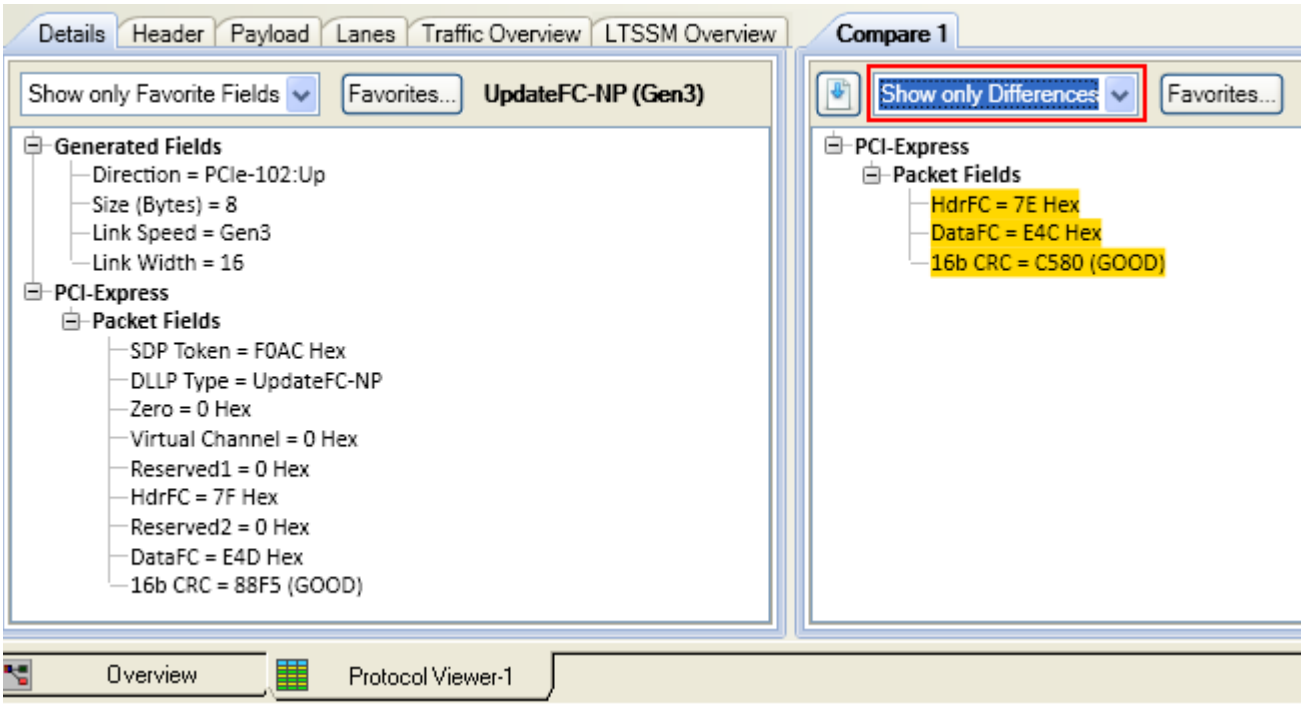
3 From the list of the packets displayed in the upper pane, select the other packet with which you want to compare the packet already displayed in the compare buffer.

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.



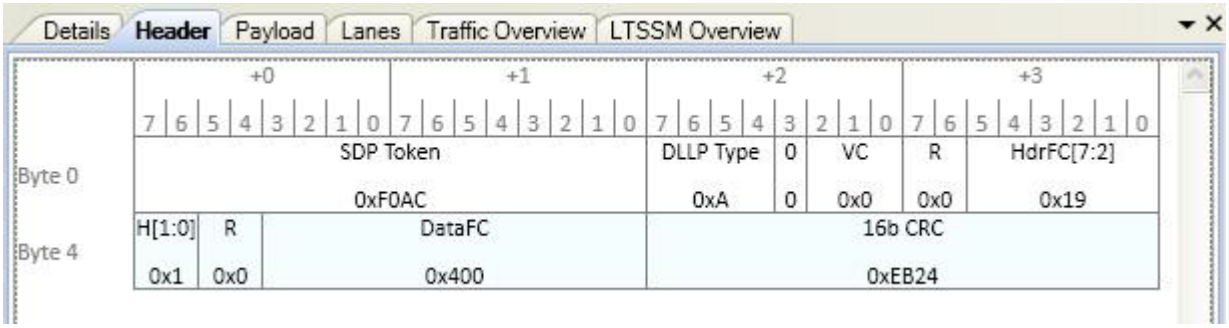
- If you want to display only a few selected fields in the comparison, then click the **Favorites** button in the compare buffer, uncheck the fields that you do not want to display, and click **OK**. Finally, select the **Show only Favorite Fields** option from the listbox in the compare buffer. Doing 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 273
- "To view a packet payload" on page 274
- "To view a packet's lanes" on page 275
- "To view LTSSM States and Transitions" on page 276

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.



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.

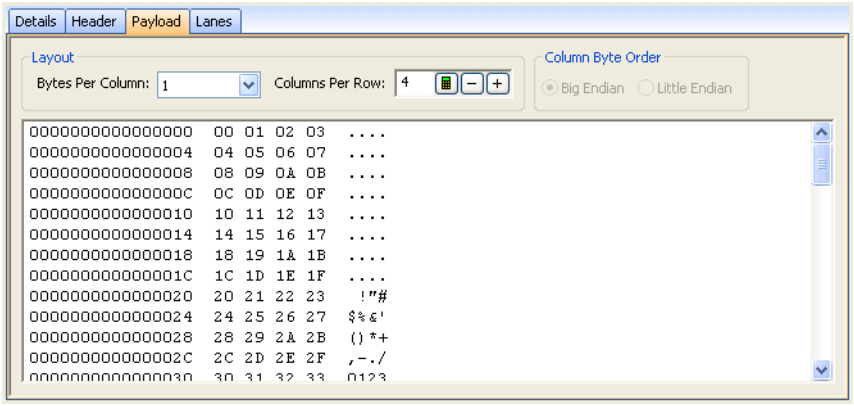
3 If desired, you can select a different number **Base**.

See Also
- "To view and compare packet details" on page 270
- "To view a packet payload" on page 274
- "To view a packet's lanes" on page 275
- "To view LTSSM States and Transitions" on page 276

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 270)
- To view a packet header (see page 273)
- To view a packet's lanes (see page 275)
- "To view LTSSM States and Transitions" on page 276

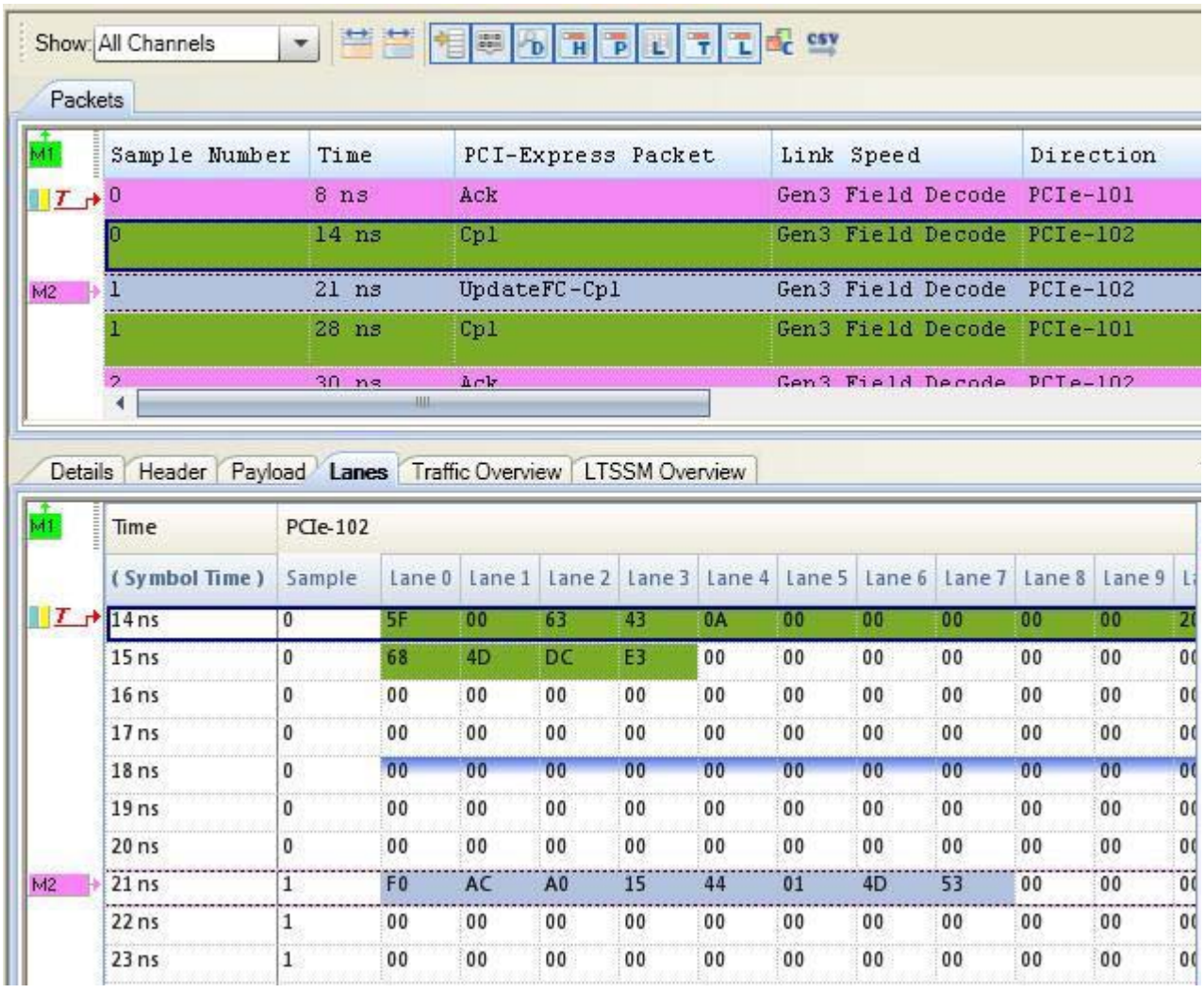
To view a packet's lanes

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.

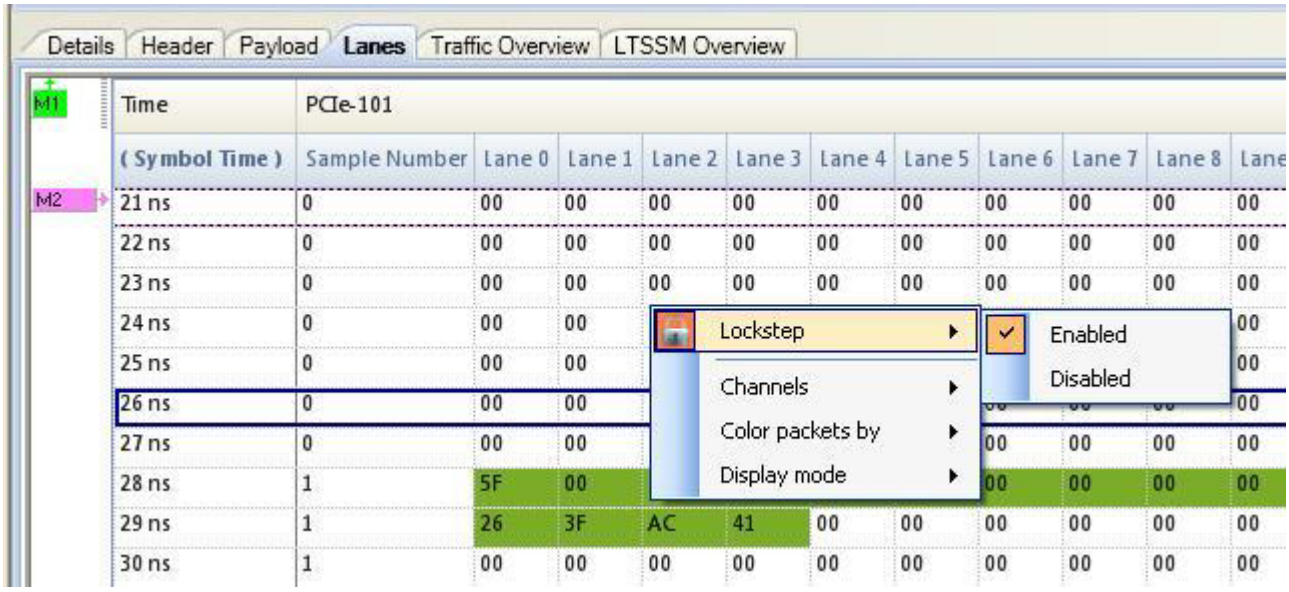
Lanes view in Protocol Viewer

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.

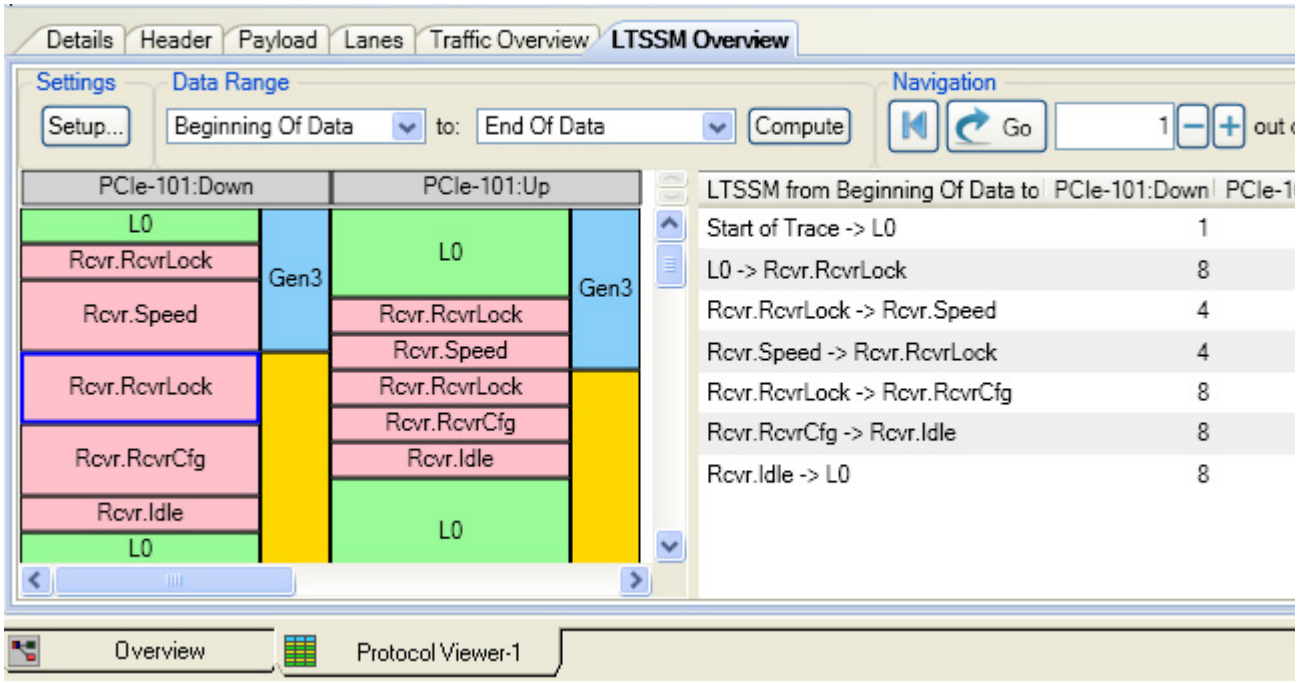


You can use this menu to customize the display of data in the Lanes view. For instance, if you disable the Lockstep option, then the upper summary pane and the lower Lanes view of Protocol viewer are not synchronized. Clicking on a packet in the summary view does not show its details in the Lane view 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 270)
- To view a packet header (see page 273)
- To view a packet payload (see page 274)

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.

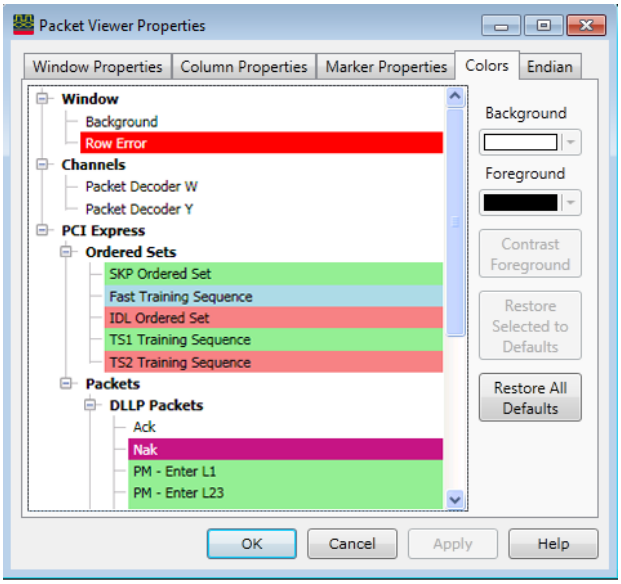
See Also
- Changing Protocol Viewer Window Properties (see page 278)

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.

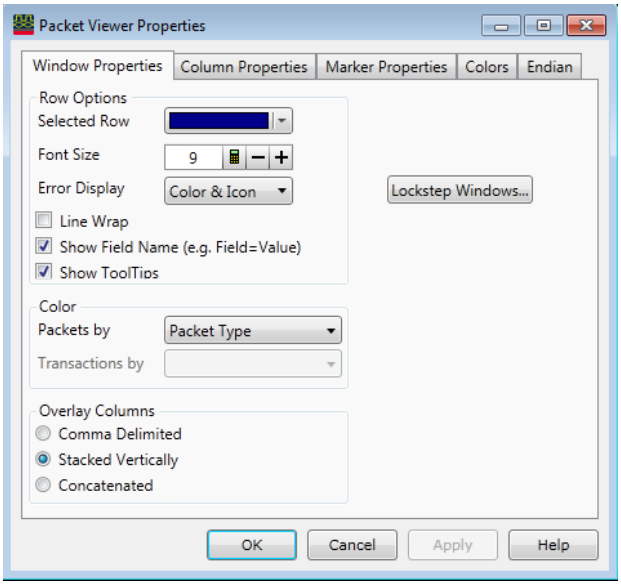
See Also

- "Changing Protocol Viewer Window Properties" on page 278
- "Changing Packet Summary Column Properties" on page 280

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 279)
   - Change the Protocol Viewer font size (see page 279)
   - Lock scrolling with other display windows (see page 279)
   - Show/hide parts of the packet summary display (see page 268)
   - Show/hide Protocol Viewer panes (see page 277)

4 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

See Also
- Changing Packet Summary Event Colors (see page 277)
- Changing Packet Summary Column Properties (see page 280)

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.

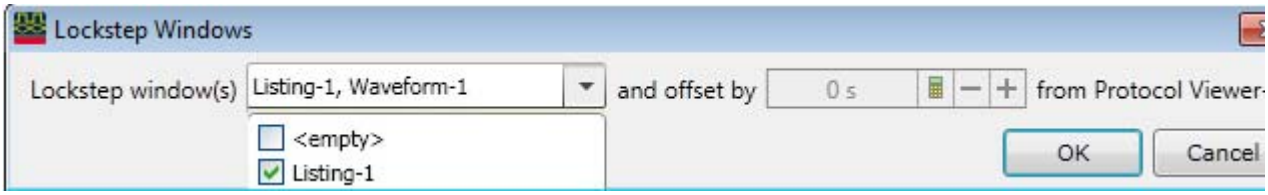
As the font size is changed, the column width may be automatically increased to fit the new text size.

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

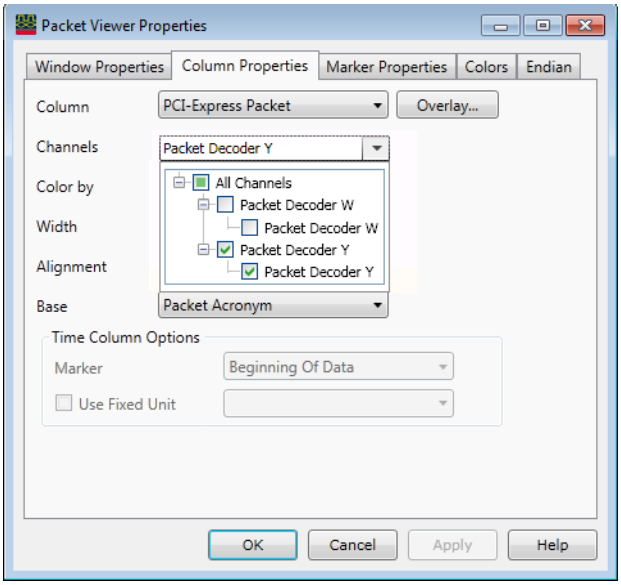
In the Protocol Viewer display window, you can change the color, width, alignment, or number base of bus/signal data columns.

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 281
- "To change the width of a packet column" on page 281
- "To change the alignment of a packet column" on page 281
- "To change a packet column's number base" on page 282
- "To select the marker for marker-relative times" on page 282
- "To select fixed time units" on page 282

3 Click **OK** to apply the changes and close the Protocol Viewer Properties dialog.

See Also
- "Changing Packet Summary Event Colors" on page 277
- "Changing Protocol Viewer Window Properties" on page 278

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 264.

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 220))
- **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 283)
- To add, duplicate, or delete windows and tools (see page 283)
- To edit window or tool properties (see page 286)
- To rename windows, tools, and modules (see page 287)
- To redraw the Overview window (see page 287)
- To delete the Overview window (see page 287)

See Also

- Overview Window (see page 389)
- Waveform Display Window (see page 377)
- Listing Display Window (see page 382)
- Compare Display Window (see page 384)
- Source Display Window (see page 385)
- "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 289) 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 283)

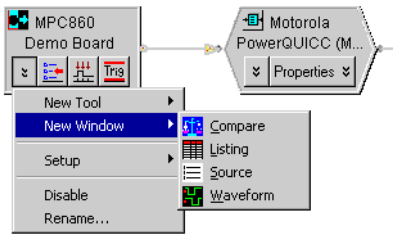
To add, duplicate, or delete windows and tools

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

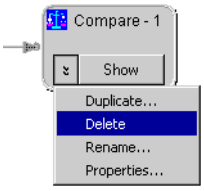
- From the menu bar, select **Window>New** *type*....
- If the windows are tabbed, you can also right-click on the tab and select **Window>New** *type*....
- In the Overview window, right-click in the background, and select **New Window** from the popup menu.
- In the Overview window, select **New Window** from a module or tool menu.



The new window is placed after the module or tool.

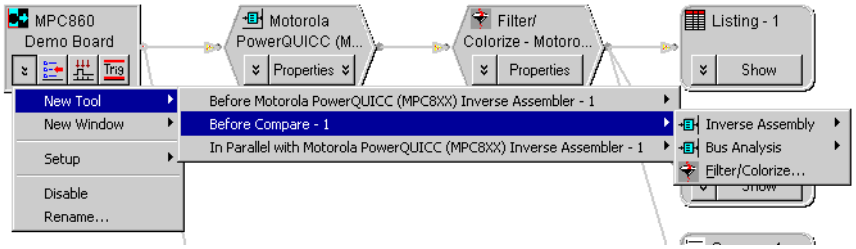
To delete windows

- 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.



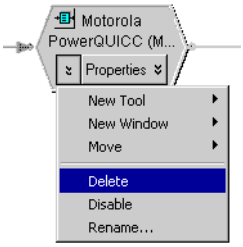
The new tool is placed after the module or tool.

If the tool you want is not listed, make sure that you have "installed" (in the online help) and licensed (see page 295) 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.

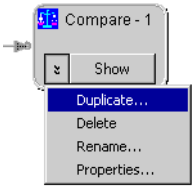
To delete tools

- In the Overview window, select **Delete** from the tool's menu.



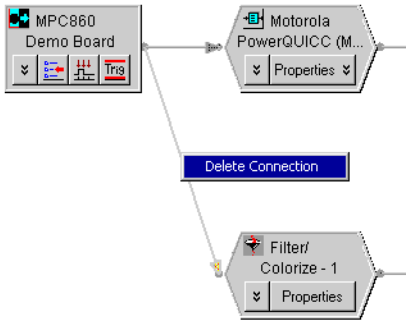
To duplicate windows

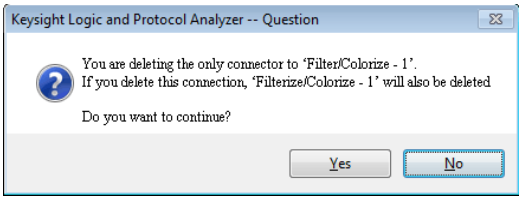
In the Overview window, you can duplicate windows from window menus. Duplicating a window is the same as adding a new window except that the new window has the same properties of the duplicated window.



To delete connections

1. In the Overview window, select the connection you wish to delete.
2. 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 389).

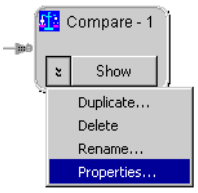
See Also

- Analyzing the Same Data in Different Ways (Using the Overview Window) (see page 283)
- To turn window tabs on/off (see page 289)

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.

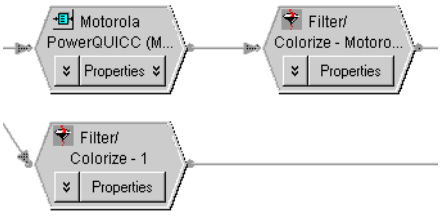
Or, right-click in the display window and select the **Properties...** command from the popup menu.

- 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.

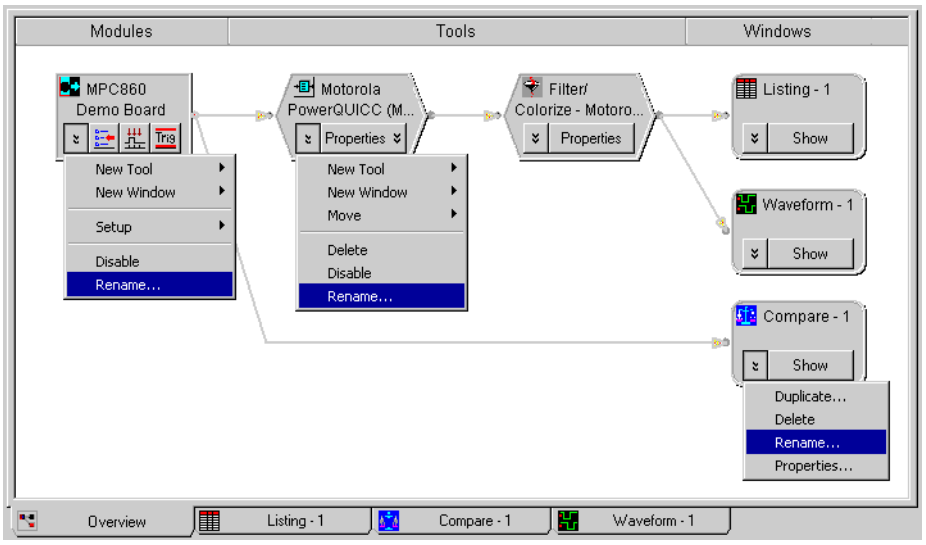
See Also

- Analyzing the Same Data in Different Ways (Using the Overview Window) (see page 283)
- To set waveform window properties (see page 204)

- To set listing window properties (see page 215)
- To set Compare window properties (see page 252)

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 283)

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 283)

To delete the Overview window

1 Display the Overview window.
2 Select the **Overview>Delete** command, or:
If Tabbed Windows (see page 289) 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 283)

Setting the System Trigger and Skew Between Modules

When there are multiple *module* (see page 540)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 410), 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 289)
- To turn window tabs on/off (see page 289)

See Also
- To change the "Go to Trigger on Run" option (see page 174)

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

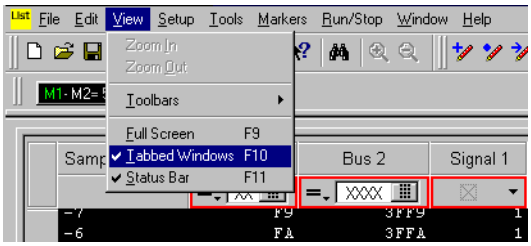
1 From the menu bar, click **Window>Close**. If windows are tabbed, you can also right-click on the tab, then select Close.

See Also
- To turn window tabs on/off (see page 289)

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 289)

Printing Captured Data

There are three ways to create printed documentation of your measurement:

- To print captured data (see page 290)
- To copy text to the clip board (see page 290)
- To copy a screen to the clip board (see page 291)

See Also
- To install a printer (see page 291)
- To connect a LAN (see page 291)

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 291). 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 177).

See Also
- To export data to CSV format files (see page 177)
- To install a printer (see page 291)

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 291)

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 320)

8 Managing Software Licenses

To view active software license information / 294
To activate software licenses / 295
To access floating license servers / 296
To borrow floating licenses and return them early / 298

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.

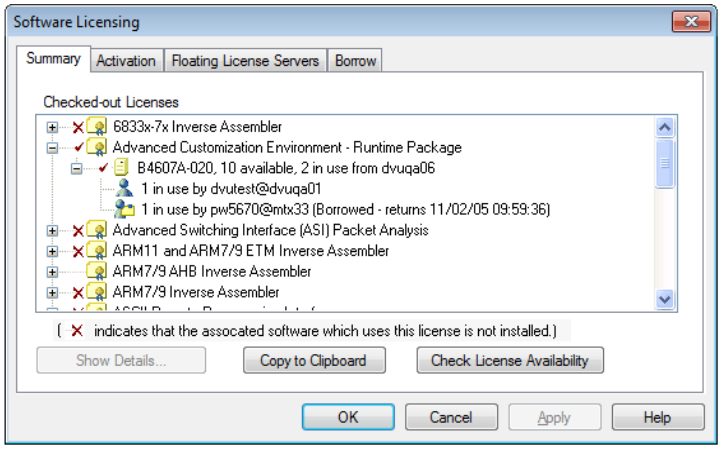
Before you can use floating licenses, you need to set up a license server.

See Also

- "*License Server Administration Guide*" for more information on setting up license server.
- Software Licensing Dialog (see page 423)

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 423):



- 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.

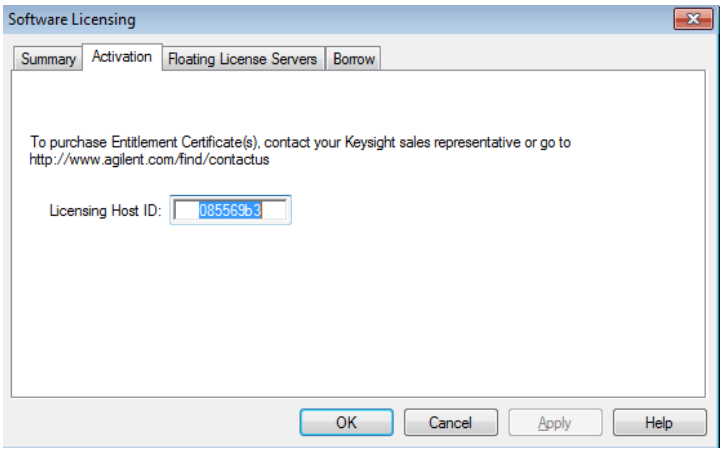
See Also
- To activate software licenses (see page 295)
- To access floating license servers (see page 296)
- To borrow floating licenses and return them early (see page 298)

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 424), 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 294)
- To access floating license servers (see page 296)
- To borrow floating licenses and return them early (see page 298)

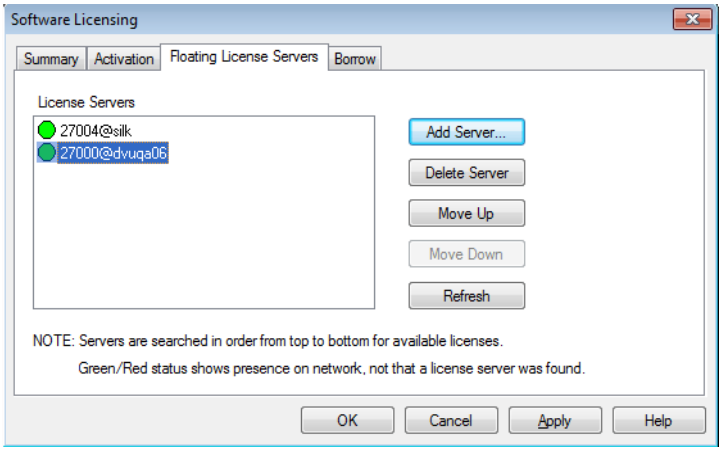
To access floating license servers

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...**.

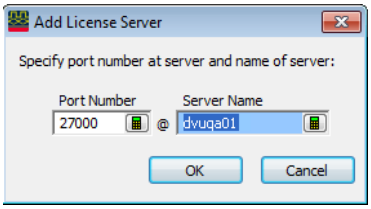
3 Select the Software Licensing dialog's Floating License Servers tab (see page 425).



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 294)
- To activate software licenses (see page 295)
- To borrow floating licenses and return them early (see page 298)

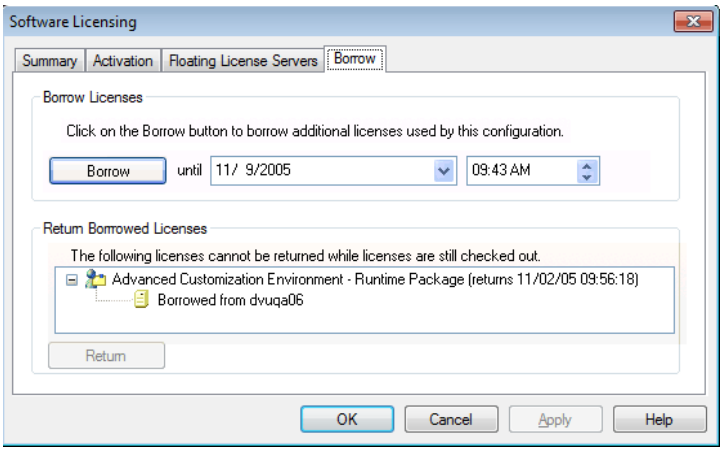
To borrow floating licenses and return them early

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 426).



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.

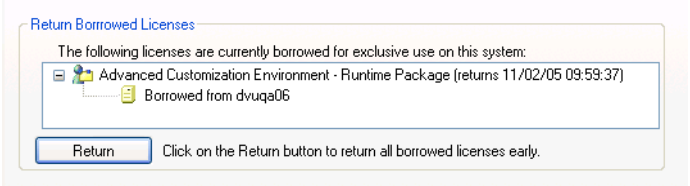
To return floating licenses early

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 294)
- To activate software licenses (see page 295)
- To access floating license servers (see page 296)

9 Updating Software

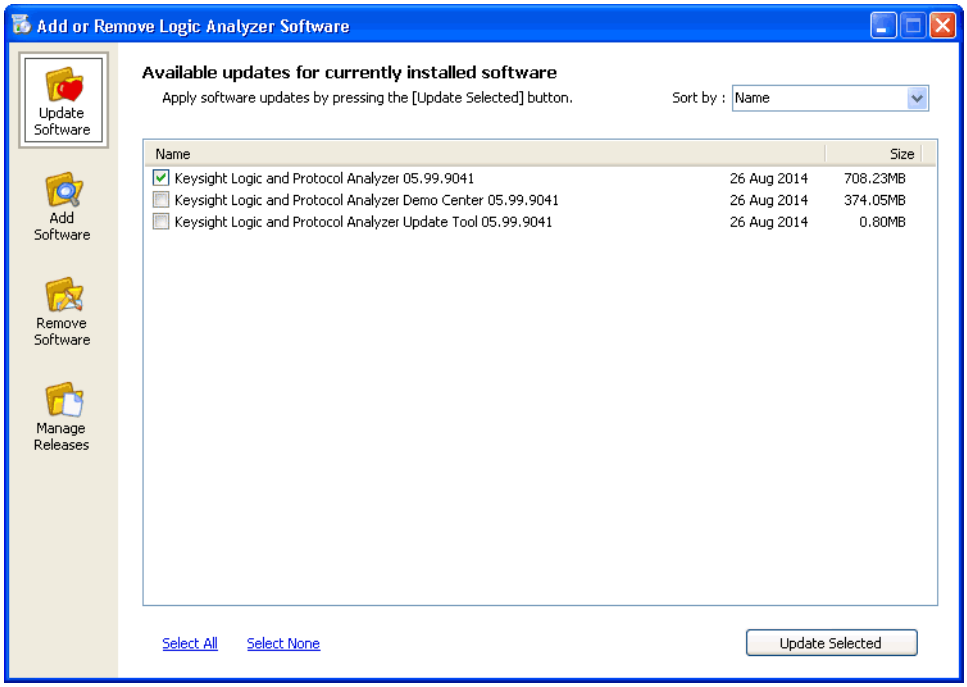
You may be able to install logic analyzer software from the logic analysis system's hard disk (depending on when it shipped from the factory or the application install CD that was last used).

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...**.

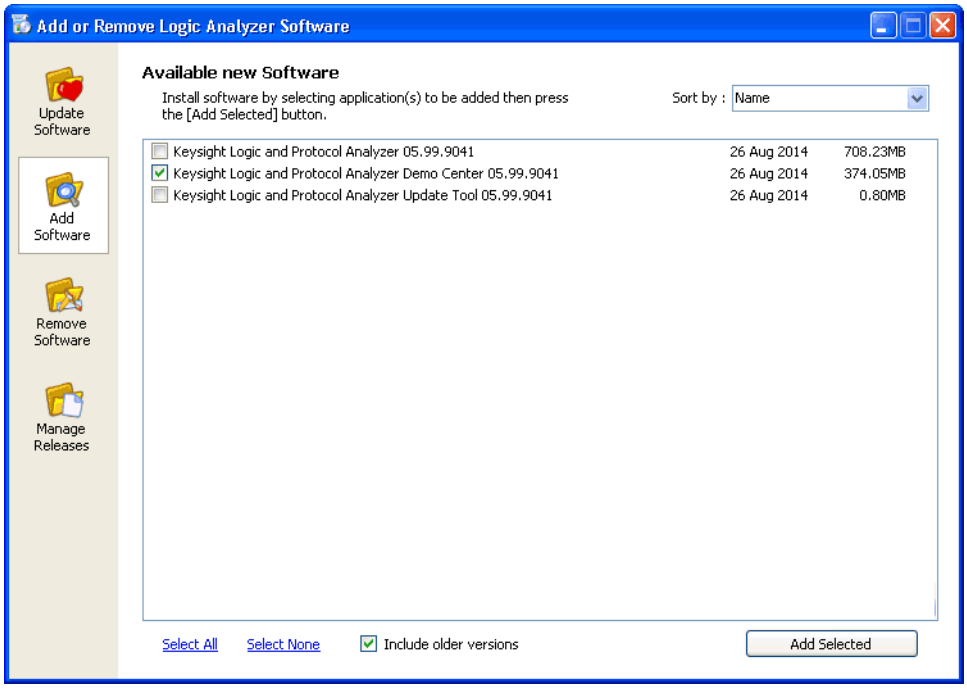
2 In the Add or Remove Keysight Logic Analyzer Software tool:

- To update software: click **Update Software** and select the software you want to update; then, click **Update Selected**.

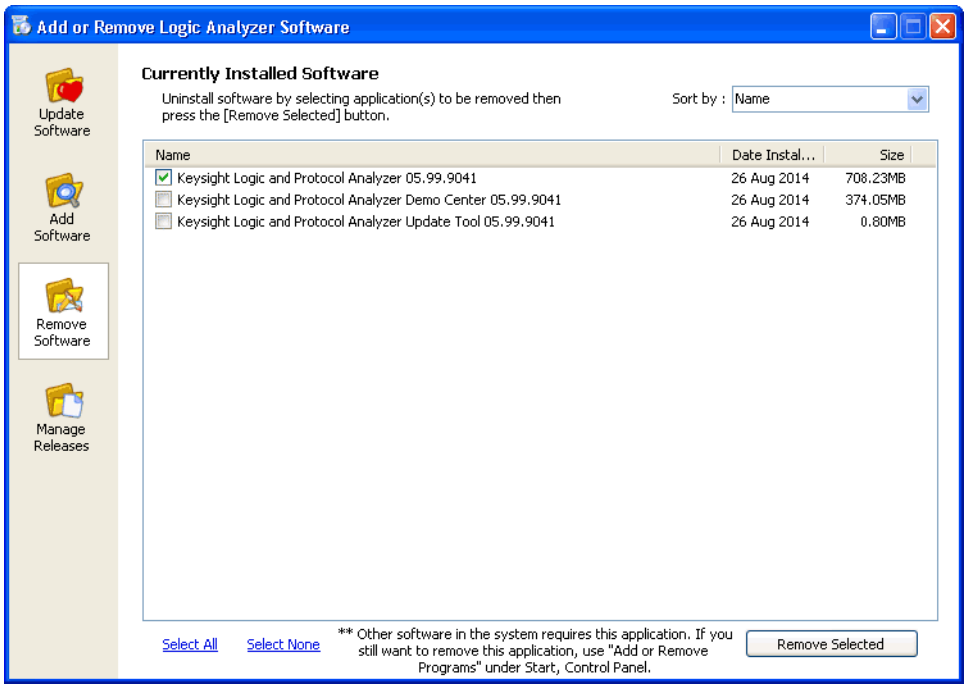


- To add software: click **Add New Software** and select the software you want to add; then, click **Add Selected**.

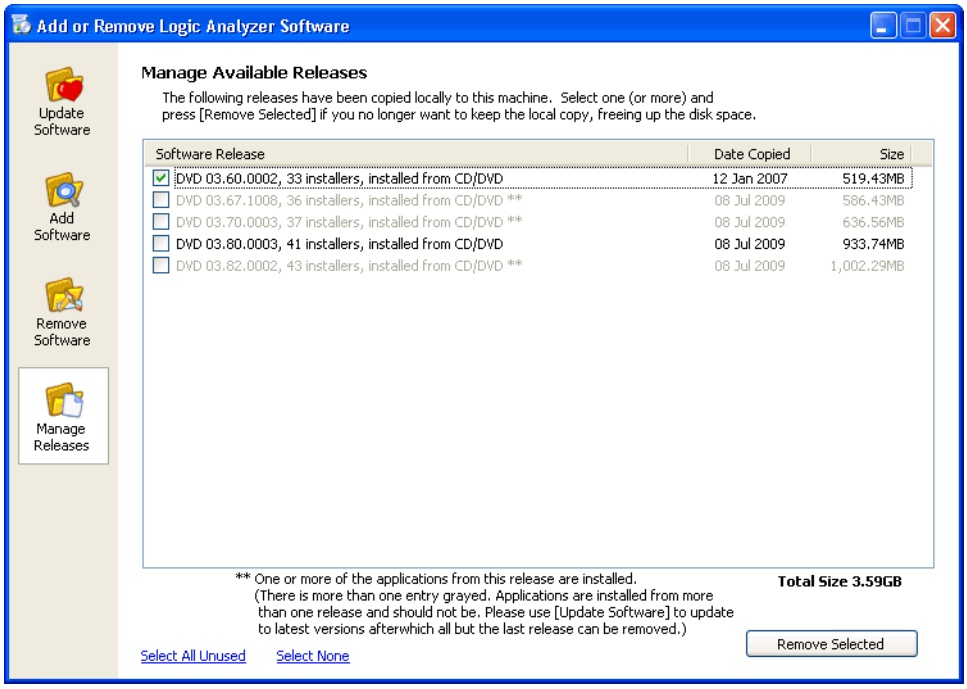
  When adding new software, you can show all versions or just the latest version.



- 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"

10 Solving Problems

When troubleshooting problems or looking for more information, see:

- If starting in offline mode is unexpected (see page 304)
- If an ALA format configuration file won't open (see page 305)
- Interpreting Error Messages (see page 306)
- License Problems (see page 314)
- Running Self Tests (see page 315)
- Accessing Japanese Online Help (Windows XP) (see page 317)
- Network Troubleshooting Guide (see page 320)
- Remote Desktop Set Up (see page 322)

See Also

- For More Information (see page 323)
- 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 176)).

3 Go back online (see Returning to Online Analysis (see page 75)).

4 Open the XML format configuration file (see To open a configuration file (see page 188) and possibly To transfer module setups to/from multi-module systems (see page 190)).

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 306)
- Warning Messages (see page 311)
- Informational Messages (see page 312)

See Also

- Solving Problems (see page 303)

Error Messages

- Acquisition Errors (see page 306)
- Bus/Signal Errors (see page 306)
- File Errors (see page 308)
- Hardware Errors (see page 309)
- Help File Errors (see page 309)
- Trigger Errors (see page 309)

See Also

- Interpreting Error Messages (see page 306)

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 528)). 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 113) 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 528)). 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 204) or column (see page 215) 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 [*time*] - 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: *duration, counted*.	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 "*filename*" 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 "*filename*" 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 462), 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 139).

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 462), 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 140).
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 462), 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 151) 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 462), 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 *Full Channel Timing Mode* (see page 492) or *Half Channel Timing Mode* (see page 492), or one edge statement per pod pair in *Transitional / Store Qualified Timing Mode* (see page 492). 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 156) 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	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. 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).
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 395) 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 395) 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 412). 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 142). For more on how to interpret the trigger sequence, see Reading Event and Action Statements (see page 139).
Timer *n* 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 139) for more information.
Counter *n* 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 140) 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 *n*...	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 *n*	The logic analyzer is waiting for a sample that matches the events defined in step *n* 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 138).
"No signal activity. Check connection, threshold, and stimulus."	This channel appears to be completely quiet. • 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 91)).

- 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 395).

"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 314)
- Floating License Server Communication Timeout (see page 314)

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 294) 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 296)).
- 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"). When you get your Entitlement Certificate, activate your licenses by using the License Activation Wizard (see To activate software licenses (see page 295)).

See Also

- Managing Software Licenses (see page 293)

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*".

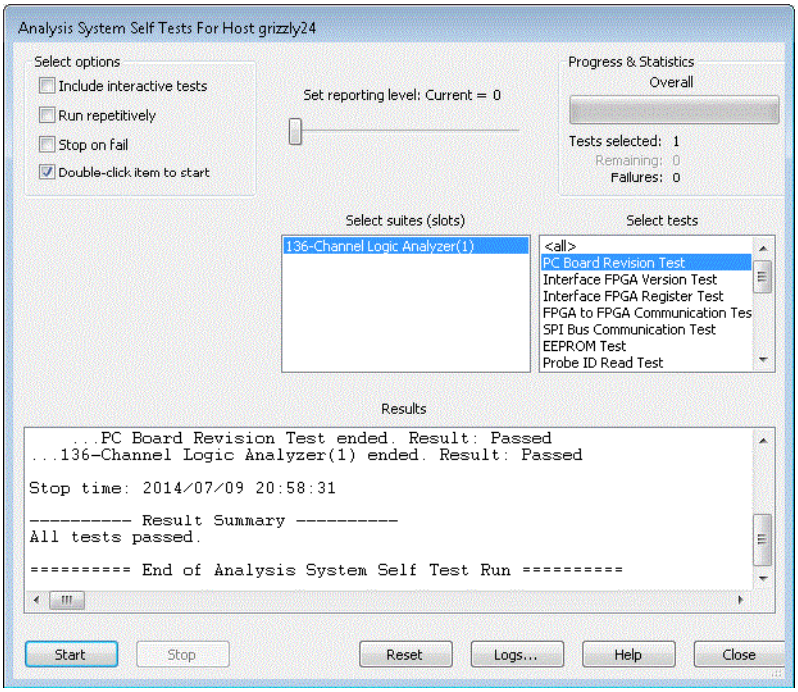
Running Self Tests

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".

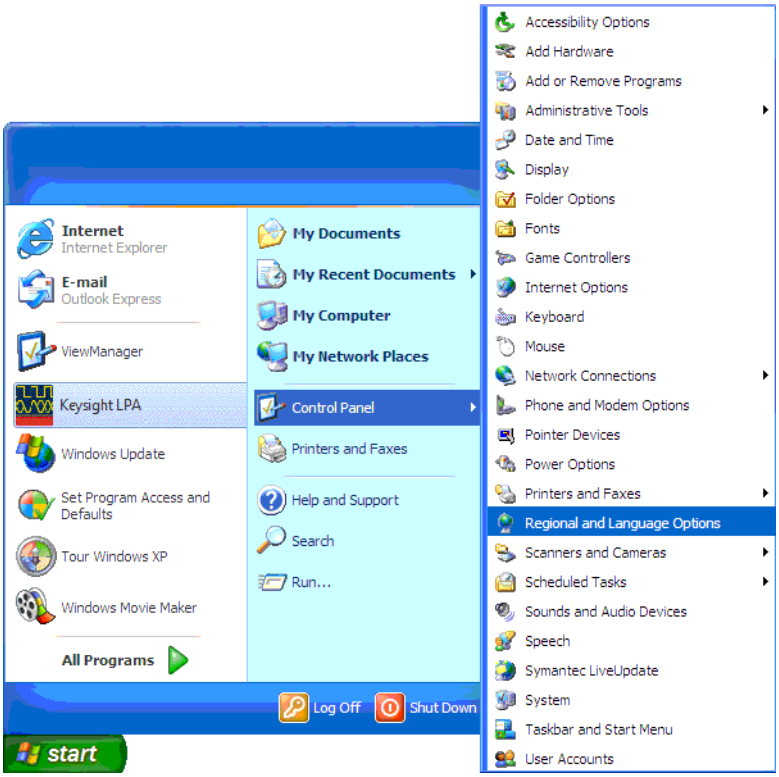
See Also

- For More Information (see page 323)

Accessing Japanese Online Help

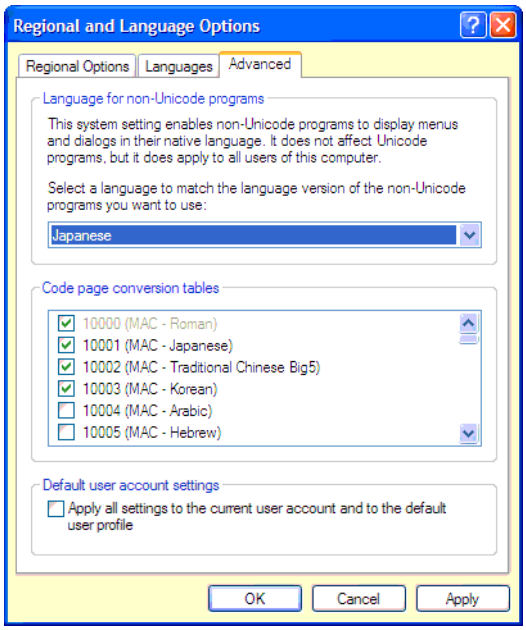
On Windows XP

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**.



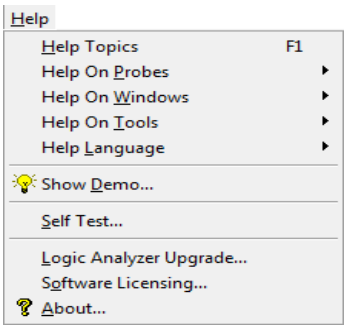
b Click **OK** to close the Regional and Language Options dialog.

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 320)
- Network Access Issues (see page 320)
- Login Issues (see page 320)
- Using with Multiframe (see page 320)
- Network Hardware and Configuration (see page 320)
- Network Topology (see page 321)
- Known OS Issues (see page 321)
- Logic Analyzer Specific Issues (see page 321)
- Keep System Protected, Up-To-Date (see page 321)

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 XP, 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 all systems are Windows XP, or 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 XP Professional or 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 XP / 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 Windows Firewall Settings, XP Service Pack 2" (in the online help)
- " Changing Firewall Settings, XP Service Pack 1" (in the online help)
- " 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 56) 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"
- Contact Us - "http://www.keysight.com/find/contactus"
- Email Updates - "http://www.keysight.com/find/emailupdates"

Product Information

- Logic Analysis - "http://www.keysight.com/find/logic"
- Logic Analysis Software Download - "http://www.keysight.com/find/la-sw-download"
- Software on CD - "http://software.cos.keysight.com/LogicAnalyzerSW"

See Also

- Tutorial - Getting to know your logic analyzer (see page 56)

11 Concepts

Logic Analysis Basics

- When should you use an oscilloscope? (see page 326)
- When should you use a logic analyzer? (see page 327)
- What is a logic analyzer? (see page 328)

  Timing analyzer:
  - Sampling clock (see page 328)
  - Sampling (see page 328)
  - Triggering (see page 329)

  State analyzer:
  - Sampling clock (see page 330)
  - Sampling (see page 331)
  - Triggering (see page 331)

Other Logic Analysis Concepts

- Pod and Channel Naming Conventions (see page 333)
- Why Are Pods Missing? (see page 334)
- Memory Depth and Channel Count Trade-offs (see page 335)
- Transitional Timing (see page 337)
- Understanding State Mode Sampling Positions (see page 339)
- Understanding Logic Analyzer Triggering (see page 341)
- ALA vs. XML, When to Use Each Format (see page 352)
- Multiframe Logic Analysis Systems (see page 353)

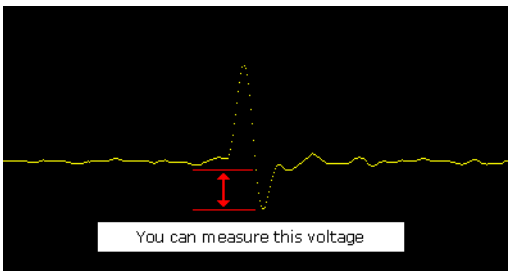
When should you use an oscilloscope?

[Tutorial Home (see page 56)] [Next Topic (see page 327)] [Previous Topic (see page 56)]

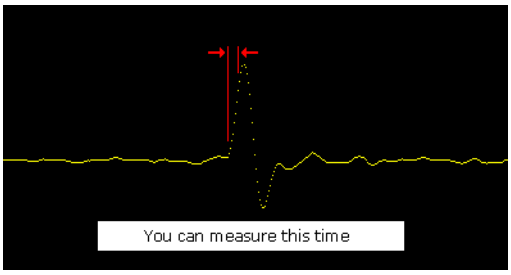
Generally, an oscilloscope is used when you need precise parametric information such as time intervals and voltage readings.

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 56)] [Next Topic (see page 328)] [Previous Topic (see page 326)]

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 56)] [Next Topic (see page 328)] [Previous Topic (see page 327)]

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 328)
- Sampling in the timing acquisition mode (see page 328)
- Triggering the timing analyzer (see page 329)

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 330)
- Sampling in the state acquisition mode (see page 331)
- Triggering the state analyzer (see page 331)

Sampling clock in the timing analyzer

[Tutorial Home (see page 56)] [Next Topic (see page 328)] [Previous Topic (see page 328)]

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:

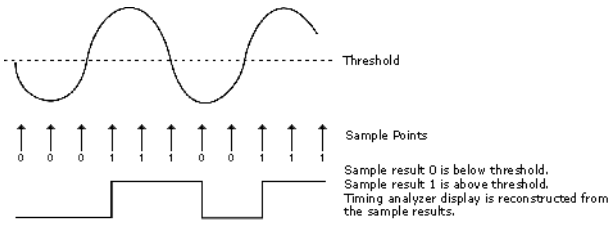
- A timing analyzer is good at showing you "When" signal activity occurs "Relative to other signals".
- A timing analyzer is more interested in viewing the timing relationships between individual signals, than the timing relationships to the signals that are controlling execution in the device under 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 56)] [Next Topic (see page 329)] [Previous Topic (see page 328)]

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.

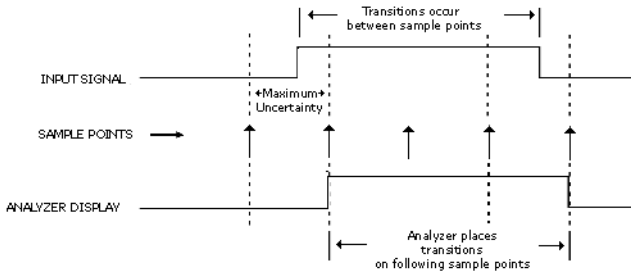


The sample points are then stored in memory and used to reconstruct a more squared-off digital waveform.

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 56)] [Next Topic (see page 330)] [Previous Topic (see page 328)]

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 know as the trigger point.

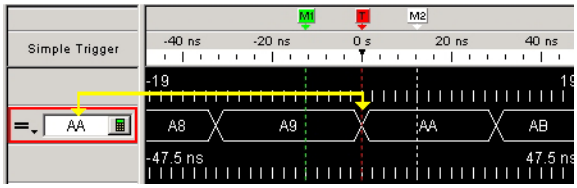
One way to get the analyzer to trigger is to configure the analyzer to look for either a pattern of highs and lows from a group of signals (bus), or a rising or falling edge from a single signal. When the analyzer sees the specified patterns or edges in data, it triggers.

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.

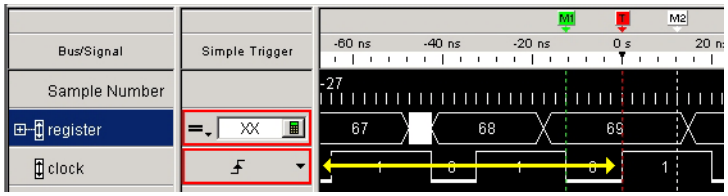
Edge Trigger

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 56)] [Next Topic (see page 331)] [Previous Topic (see page 329)]

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.

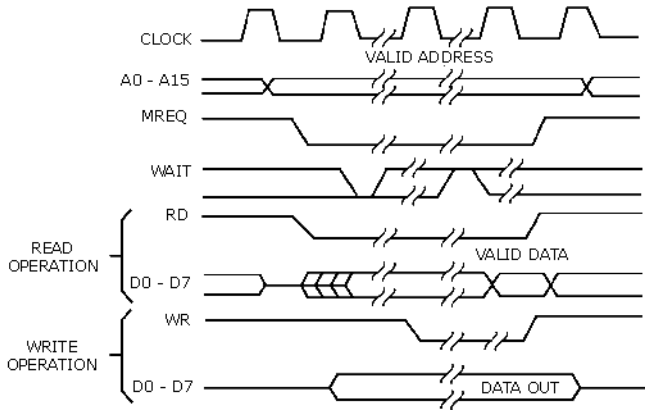
Sampling in the state acquisition mode

[Tutorial Home (see page 56)] [Next Topic (see page 331)] [Previous Topic (see page 330)]

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).



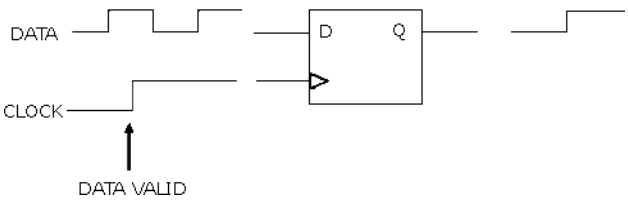
Triggering the state analyzer

[Tutorial Home (see page 56)] [Next Topic (see page 56)] [Previous Topic (see page 331)]

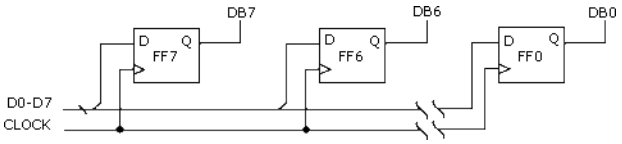
Similar to a timing analyzer, a state analyzer has the capability to qualify the data we want to store. If we are looking for a specific pattern of highs and lows on the address bus, we can tell the analyzer to start storing when it finds that pattern and to continue storing until the analyzer's memory 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.



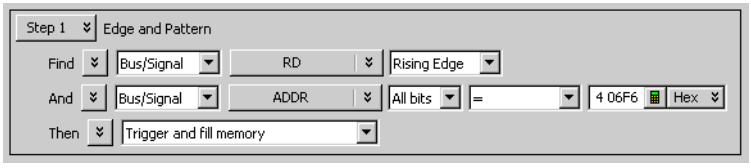
When a high level occurs on the clock line, all eight capture data at their "D" inputs. Again, a valid state occurs each time there is a positive level on the clock line.

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
		=, XXX$	1
	-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.



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 **Equals406F6 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 497)

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* (see page 492) 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 492) 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 492) 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.
- The module is part of a logic analyzer that has been split. In this case, the pods are in the module that is the other half of the split analyzer.

See Also

- Memory Depth and Channel Count Trade-offs (see page 335)
- Configuring Logic Analyzer Modules (see page 88)

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 335)
- Transitional Timing Sampling Mode (see page 336)

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 528) 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.

Splitting an Analyzer

In the Split Analyzer Setup dialog (see page 429) you can:

- Specify whether pod pairs should be reserved for time tag storage.

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 528) 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 88)
- To set acquisition memory depth (see page 115)
- Choosing the Sampling Mode (see page 106)
- Logic Analyzer Notes, Channels and Memory Depth (see page 492)

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.

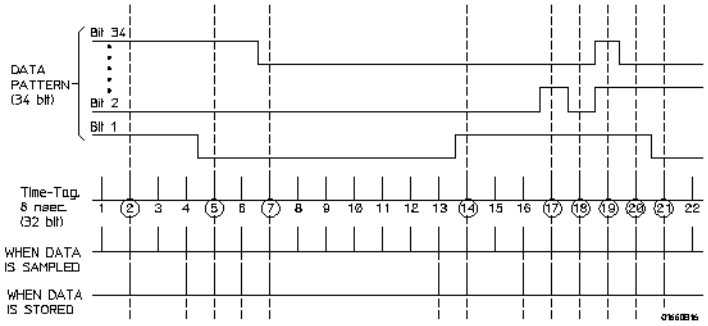
More on Storing Transitions

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 392), 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

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 *defined* buses/signals (that haven't been excluded).

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.

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.

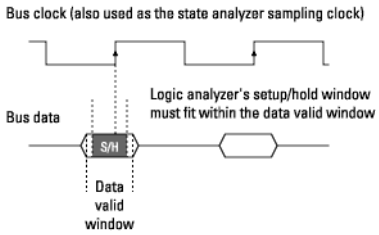
Understanding State Mode Sampling Positions

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.

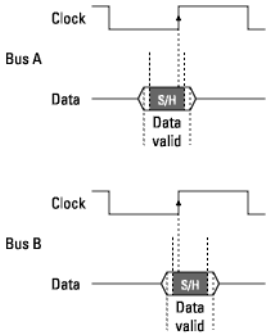
A device under test (because of its own setup/hold requirements) specifies that data be valid on a bus for a certain length of time. This is known as the data valid window. The data valid window on most buses is generally less than half of the bus clock period.

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 U4154A/B Logic Analyzer" on page 508 to know more about the eyescan feature.**

See Also

- To automatically adjust state sampling positions and threshold voltages (see page 114)
- To manually adjust state sampling positions (see page 114)
- Selecting the State Mode (Synchronous Sampling) (see page 108)
- "Setting up and Running Eyescans in U4154A/B Logic Analyzer" on page 508

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.

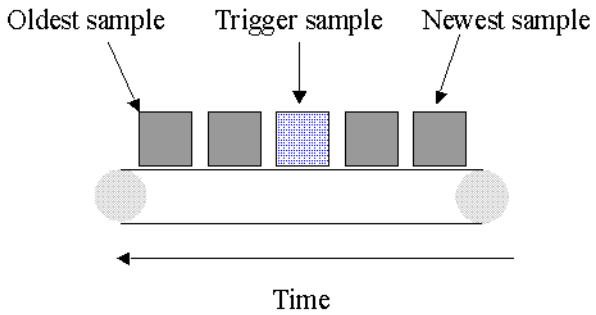
- The Conveyor Belt Analogy (see page 341)
- Summary of Triggering Capabilities (see page 342)
- Sequence Steps (see page 343)
- Boolean Expressions (see page 344)
- Branches (see page 345)
- Edges (see page 345)
- Ranges (see page 345)
- Flags (see page 345)
- Occurrence Counters and Global Counters (see page 345)
- Timers (see page 346)
- Storage Qualification (see page 347)
- Strategies for Setting Up Triggers (see page 348)
- Conclusions (see page 351)

See Also

- Capturing Data from the Device Under Test (see page 127)

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 342)

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	`If there is rising edge on SIG1 then Trigger` `If there is falling edge on SIG1 then Trigge r`
Boolean expressions	`If ADDR = 1000 and DATA = 2000`
Ranges	`If ADDR in range 1000 to 2000`

Storage qualification	`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 th en`
	`Don't Store Sample`
	`Go to 1`

Counters	`1. If DATA = 1000 Then`
	`Increment Counter 1`
	`Go to 2`
	`2. If Counter 1 > 2 Then`
	`Trigger`

Timers	`1. If DATA = 1000 Then`
	`Start Timer 1`
	`Go to 2`
	`2. If Timer 1 > 500 ns Then`
	`Trigger`

Next: Sequence Steps (see page 343)

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 344)

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 345)

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 345)

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 345)

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 345)

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: Occurrence Counters and Global Counters (see page 345)

Occurrence Counters and Global 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
```

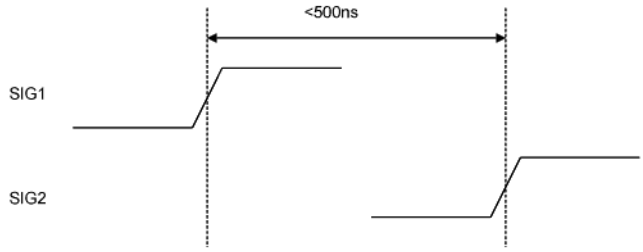
Global Counters are like integer variables. They are more flexible than Occurrence Counters because they can be used to count complex events such as an edge followed by another edge. Global Counters can be incremented, tested, and reset. By default, Global Counters begin with zero and don't need to be 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.

Next: Timers (see page 346)

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 347)

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.

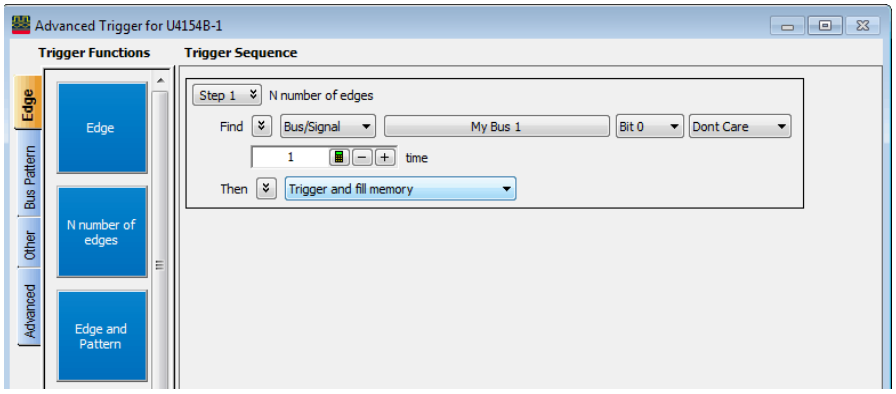
Next: Strategies for Setting Up Triggers (see page 348)

Strategies for Setting Up Triggers

- Trigger Functions (see page 348)
- Setting Up Complex Triggers (see page 350)
- Save and Document Your Trigger Sequences (see page 350)

Trigger Functions

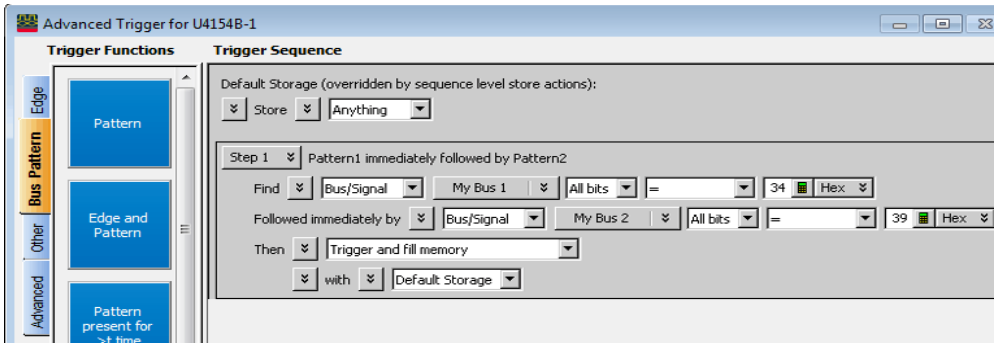
While setting up logic analyzer triggers can be difficult, *trigger functions* can greatly simplify the process. Trigger functions are commonly-needed building blocks that can be combined to set up a trigger. Because the functions cover most common triggers, you can set up your trigger simply by selecting 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

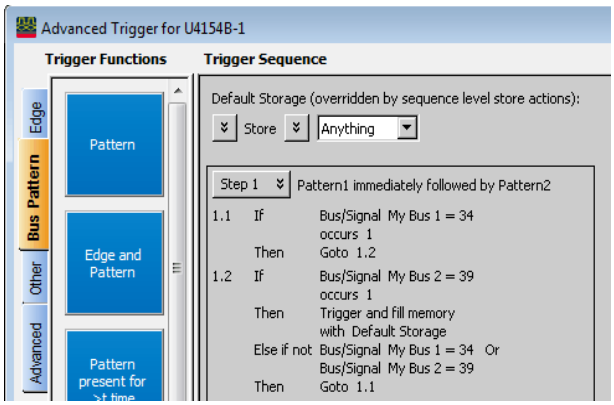
Note that a picture (which corresponds to the selected function) is provided by hovering over the trigger function button.

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

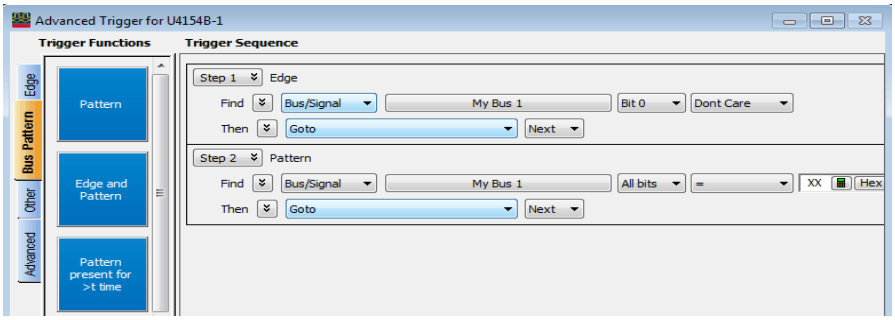
Once you have selected this function, you simply fill in the names of the buses and the patterns. Contrast the previous figure with the following figure, which is the same trigger created using If/Then statements. The trigger function is easier to use because the additional details of the If/Then statements 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 350)

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.

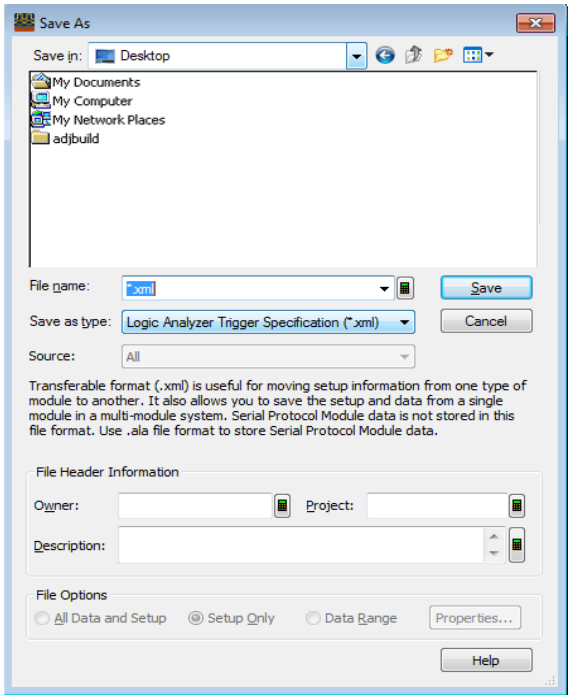
Next: Save and Document Your Trigger Sequences (see page 350)

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 161.

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 172 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 351)

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 *offline* mode, you can read ALA format configuration files in any instance of the *Keysight Logic Analyzer* 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 *online* 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 *offline* 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 176)
- To open a configuration file (see page 188)
- To transfer module setups to/from multi-module systems (see page 190)
- ALA Format (see page 466)
- "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.

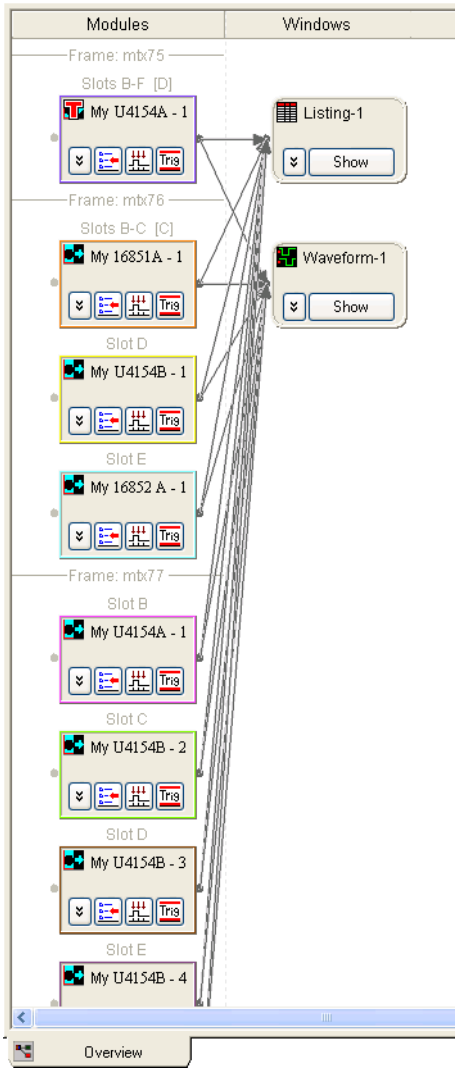
Remotely hosted systems may have their own boot sequence requirements. Usually this is to boot the chassis first and then the host computer.

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.

12 Reference

Menus / 358
Toolbars / 371
Marker Measurement Display Bar / 376
Windows / 377
Dialogs / 391
Trigger Functions / 440
Data Formats / 466
Object File Formats Supported by the Symbol Reader / 475
General-Purpose ASCII (GPA) Symbol File Format / 476
Product Overviews / 482
- U4154A Logic Analyzer Product Overview / 482
- U4154B Logic Analyzer Product Overview / 484
- 16850-Series Logic Analyzer Product Overview / 485
- Keysight Logic and Protocol Analyzer Application Product Overview / 488

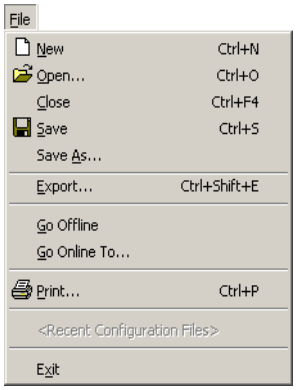
Logic Analyzer Notes / 492
- U4154A Logic Analyzer Notes / 492
- U4154B Logic Analyzer Notes / 494
- 16850-Series Logic Analyzer Notes / 526

Specifications and Characteristics / 528
- U4154A Logic Analyzer Specifications and Characteristics / 528
- U4154B Logic Analyzer Specifications and Characteristics / 531
- 16850-Series Logic Analyzer Specifications and Characteristics / 531

Menus

- File Menu (see page 358)
- Edit Menu (see page 359)
- View Menu (see page 360)
- Setup Menu (see page 361)
- Tools Menu (see page 362)
- Markers Menu (see page 364)
- Run/Stop Menu (see page 365)
- Overview Menu (see page 366)
- Listing Menu (see page 366)
- Waveform Menu (see page 366)
- Compare Menu (see page 367)
- Source Menu (see page 368)
- Protocol Viewer Menu (see page 368)
- Window Menu (see page 368)
- Help Menu (see page 369)

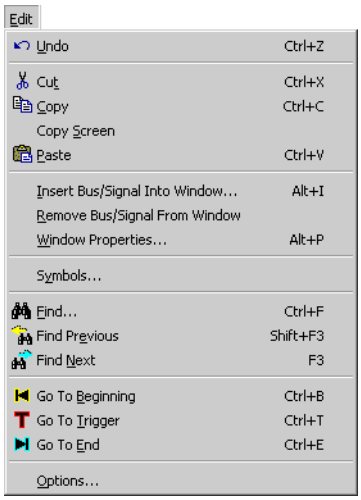
File Menu



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 176)	Saves changes to the currently open configuration file.
Save As... (see page 176)	Saves the currently open configuration file to a new name.
Export... (see page 177)	Saves captured data to comma-separated value (CSV) files.
Go Offline	Disconnects the *Keysight Logic Analyzer* application from the currently connected frame.

Menu	Description
Go Online To Local Frame	Connects the *Keysight Logic Analyzer* application to the local frame. If there is no local frame, the Offline Startup Options dialog (see page 411) opens.
Go Online To...	Opens the Select System to Use dialog (see page 421) for choosing a frame to connect the *Keysight Logic Analyzer* application to.
Print...	Opens the Printing Data dialog (see page 415) for printing displayed data within a defined range.
Recent Configuration Files (see page 190)	Lists recently opened files for quick reference or access.
Exit	Closes the logic analyzer user interface window.

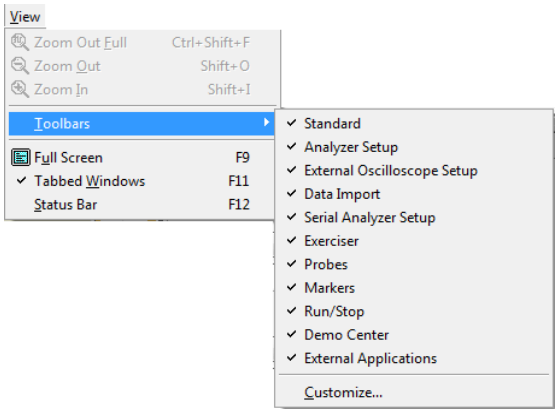
Edit Menu



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 490) (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 490) (accelerator keys). Copied selections are pasted to the clip board.
Copy Screen (see page 291)	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 490) (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 415)	Accesses the window properties dialog.
Symbols...	Opens the Symbols dialog (see page 433) for setting up symbols for the selected bus/signal.
Find... (see page 240)	Locates specific data in the acquisition.
Find Previous (see page 240)	Locates the previous occurrence of the specified data.
Find Next (see page 240)	Locates the next occurrence of the specified data.
Go To Beginning (see page 226)	Places the beginning of the captured data trace at center screen.
Go To Trigger (see page 226)	Places the trigger point at center screen.
Go To End (see page 226)	Places the end of the captured data trace at center screen.
Options...	Accesses the System Options dialog (see page 412).

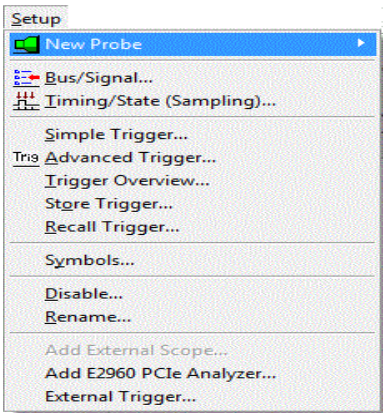
View Menu



Menu	Description
Zoom Out Full	Zooms out on an active window as far as possible.
Zoom Out (see page 198)	Zooms out on an active window.
Zoom In (see page 198)	Zooms in on an active window.
Toolbars (see page 371)	Access the Toolbar dialog window.

Menu	Description
Full Screen	Enables or disables full screen display.
Tabbed Windows (see page 289)	Enables or disables Listing and Waveform tabs.
Status Bar	Enables or disables the status bar.

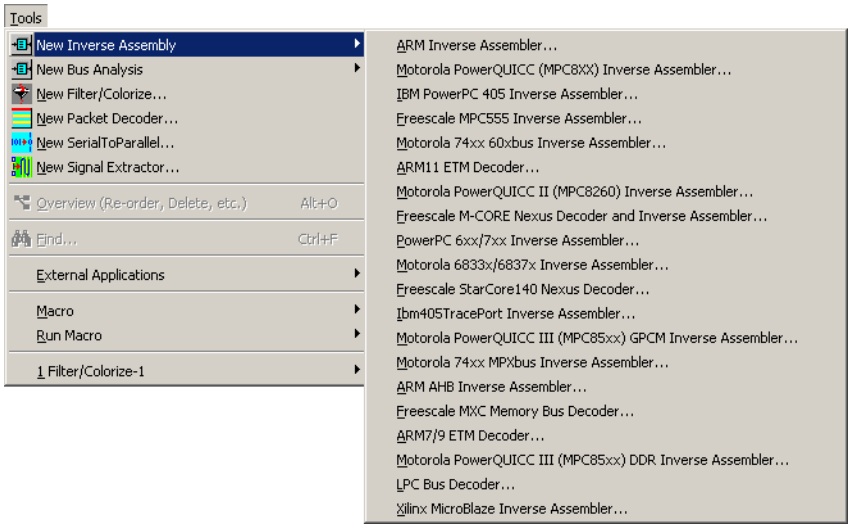
Setup Menu

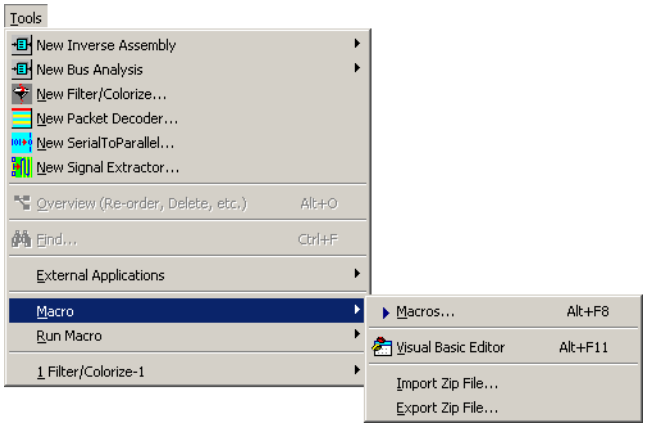


Menu	Description
New Probe	Lets you set up the FPGA Dynamic Probe (for probing signals internal to an FPGA) or set up definitions for other probes that are used (see "Setting Up Probes" (in the online help)).
Bus/Signal...	Accesses the Buses/Signals (see page 395) tab of the Analyzer Setup dialog (see page 394).
Timing/State (Sampling)...	Accesses the Sampling (see page 396) tab of the Analyzer Setup dialog (see page 394).
Simple Trigger...	See Specifying Simple Triggers (see page 132).
Advanced Trigger...	Accesses the Advanced Trigger (see page 392) dialog.
Store Trigger...	Stores current trigger.
Recall Trigger... (see page 417)	Accesses a list of most recently used triggers.
Symbols...	Opens the Symbols dialog (see page 433) for setting up symbols for the selected bus/signal.
Pod Assignment...	Opens the Pod Assignment dialog (see page 414) for reserving pods or pod pairs for time tag storage. When a *module* (see page 540) has been split into two modules, this dialog can also be used to re-assign channels to the split modules.
Split Analyzer...	Opens the Split Analyzer Setup dialog (see page 429) for splitting a logic analyzer module into two modules.

Menu	Description
Unsplit Analyzer...	This item appears when a logic analyzer module has been split into two modules; it re-combines the split modules into one module.
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.
" Add External Scope..." (in the online help)	Runs the *Add External Oscilloscope* 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 404) for setting up triggers between the logic analyzer and other, external instruments.
Target Control Port...	Opens the Target Control Port dialog (see page 437) for outputting signals on the logic analysis system frame's *target control port*.
Skew & System Trigger...	Opens the Module Skew and System Trigger dialog (see page 410) for specifying which *module* (see page 540) is the *system trigger* (that is, which module's trigger reference point is *Time=0*) and for specifying the trigger reference point skew for modules that are not the *system trigger*.

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 283)	Creates a new inverse assembly, bus analysis, filter/colorize, packet decoder, serial to parallel, or signal extractor tool.
Overview (see page 283)	Lets you manage the active tools.
Find... (see page 240)	Locates specific data in the acquisition.
External Applications>	Lets you run external applications from the *Keysight Logic Analyzer* application's menu. **Setup...** on the submenu opens the External Application Setup dialog (see page 403) that lets you add, edit, arrange, or remove items from the submenu.

Menu	Description
Macro>	Lets you: · 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.

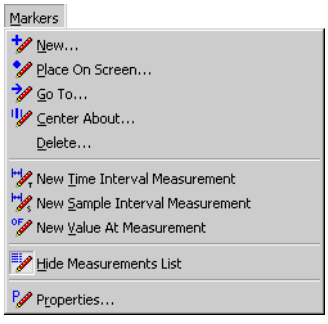
Run Macro>	Opens the Add-In Manager dialog which lets you register an add-in (a customized tool that adds capabilities to the Visual Basic development environment), load or unload it, and set its load behavior. For more information, click **Help** in the Add-In Manager dialog.
	Runs a sample macro. As shipped from the factory, the submenu contains:
	· 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).
	· SendToPatternGeneratorModule (macro for sending logic analyzer data to a pattern generator module as stimulus vectors).
	You can add your own COM/DCOM macros to this submenu by placing COM/DCOM project code .zip files in the directory:
	`<Drive letter>:\<Install directory>\COM/DCOM\`
	For example:
	`C:\Program Files\Keysight Technologies\Logic Analyzer\COM/DCOM\`

1 *tool name* (see page 286)	Edit an existing tool.

See Also

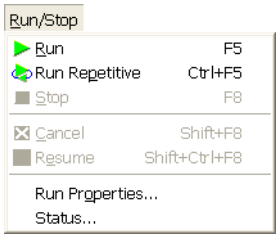
- To add a new tool (see page 283)
- To change a tool (see page 286)
- To delete a tool (see page 283)
- "Using the Filter/Colorize Tool" (in the online help)

Markers Menu



Menu	Description
New... (see page 223)	Creates a new marker.
Place on Screen... (see page 225)	Places a new or selected (existing) marker at the middle of the screen.
Go To... (see page 226)	Goes to a selected marker.
Center About... (see page 226)	Centers the display around a selected marker.
Delete...	Deletes selected markers.
New Time Interval Measurement (see page 228)	Creates a new time interval measurement.
New Sample Interval Measurement (see page 229)	Creates a new sample interval measurement.
New Value At Measurement (see page 230)	Creates a new value at measurement.
Hide/Show Measurements List	Hides or shows the marker measurement display bar.
Properties... (see page 232)	Accesses the markers properties dialog

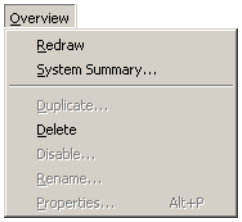
Run/Stop Menu



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.
(Pattern Generator Module)>	Provides access to the " pattern generator module's run/stop commands" (in the online help).
Cancel	When searching, using a filter, or exporting captured data, Cancel stops the operation.

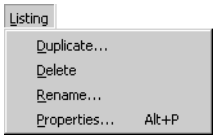
Menu	Description
Resume	If you have used Cancel to stop a filter tool operation, Resume continues the filter operation.
Run Properties...	Opens the Run Properties dialog (see page 418) which lets you enable, and set the options for, saving captured data after each run and stopping after a certain number of repetitive runs.
Status...	Opens the Status dialog (see page 430).

Overview Menu



Menu	Description
Redraw	Re-paints the Overview window.
System Summary...	Opens the System Summary dialog (see page 434) which displays information about the *frames* (see page 539), *modules* (see page 540), *cards* (see page 538), and *slots* (see page 542) in the logic analysis system.
Delete	Closes the Overview window.

Listing Menu

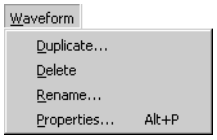


Menu	Description
Duplicate...	Adds a new Listing window with the same properties as the window being displayed.
Delete	Closes the Listing window.
Rename...	Lets you rename the Listing window.
Properties...	Lets you change Listing window properties.

See Also

- Analyzing Listing Data (see page 212)
- To set listing window properties (see page 215)

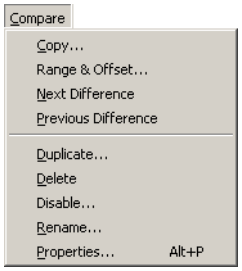
Waveform Menu



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 197)
- To set waveform window properties (see page 204)

Compare Menu

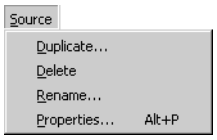


Menu	Description
Copy...	Copies data to the reference buffer.
Range & Offset...	Lets you compare a range of samples and offset the reference data.
Next Difference	Finds the next difference (below the center reference).
Previous Difference	Finds the previous difference (above the center reference).
Duplicate...	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 384)

- Comparing Captured Data to Reference Data (see page 250)
- To set Compare window properties (see page 252)

Source Menu

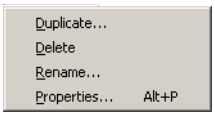


Menu	Description
Duplicate...	Adds a new Source window with the same properties as the window being displayed.
Delete	Closes the Source window.
Rename...	Lets you rename the Source window.
Properties...	Lets you change Source window properties.

See Also

- Source Display Window (see page 385)
- Viewing Source Code Associated with Captured Data (see page 253)
- To set Source window properties (see page 257)

Protocol Viewer Menu

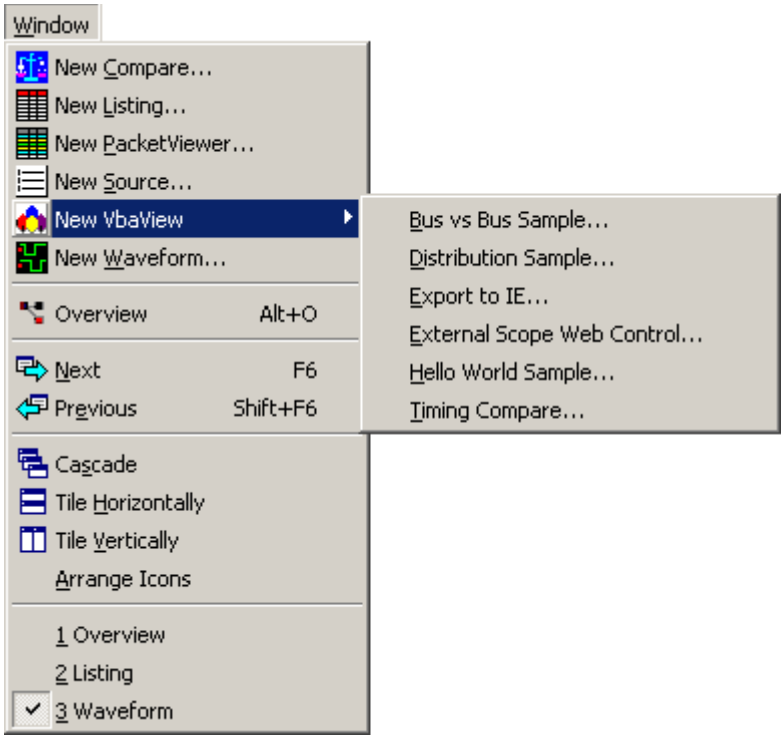


Menu	Description
Duplicate...	Adds a new Protocol Viewer window with the same properties as the window being displayed.
Delete	Closes and deletes the Protocol Viewer window.
Rename...	Lets you rename the Protocol Viewer window.
Properties...	Lets you change Protocol Viewer window properties.

See Also

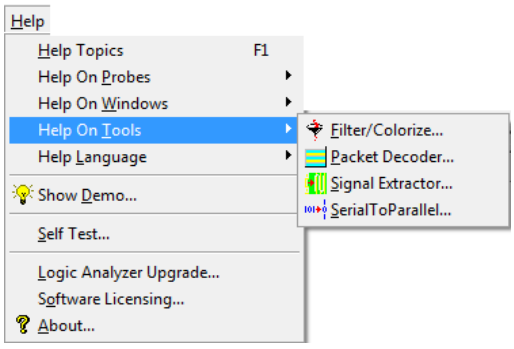
- Protocol Viewer Display Window (see page 386)
- Analyzing Packet Data (see page 259)
- Changing Protocol Viewer Window Properties (see page 278)

Window Menu



Menu	Description
New Compare...	Creates a new Compare window (see page 384).
New Listing...	Creates an additional Listing window (see page 382).
New Protocol Viewer...	Creates an additional Protocol Viewer window (see page 386).
New Source...	Creates a new Source window (see page 385).

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 (Windows XP) (see page 317)).
Show Demo...	Launches the Demo Center (see page 71) application that demonstrates logic analysis system features.
Self Test...	Accesses the Logic Analyzer Self-Tests dialog (see Running Self Tests (see page 315)).
Logic Analyzer Upgrade...	Accesses the Keysight Logic Analyzer Upgrade dialog (see page 393) which provides information for upgrading hardware in logic analyzer modules.
Software Licensing...	Opens the Software Licensing dialog (see page 423) for managing software licenses used by the logic analysis system.
Software Update...	Opens the Add or Remove Keysight Logic Analyzer Software (see page 299) 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 371)
- Pattern Generator (see page 372)
- Analyzer Setup (see page 372)
- External Oscilloscope Setup (see page 373)
- Probes (see page 373)
- Markers (see page 373)
- Run/Stop (see page 374)
- Visual Basic (see page 374)
- Customize... (see page 375)

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 415) - Prints displayed data within a defined range.
	Find (see page 240) - Locates specific data in the acquisition.
	Find Previous (see page 240) - Locates the previous occurrence of the specified data.
	Find Next (see page 240) - Locates the next occurrence of the specified data.
	Go to Beginning (see page 226) - Centers the beginning of the acquisition data.
	Go to Trigger (see page 226) - Centers the trigger point of the acquisition.
	Go to End (see page 226) - Centers the end of the acquisition data.
	Zoom Out (see page 198) - Zooms in on an active window.
	Zoom In (see page 198) - Zooms out on an active window.
	Overview - Opens or displays the Overview window (see page 389).

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 *Keysight Logic Analyzer* 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 375)

Pattern Generator Toolbar

Menu	Description
	Sets up the pattern generator bus/signal attributes.
	Sets up the pattern generator clocking attributes.
Seq	Sets up the pattern generator sequence.
Mac	Sets up the pattern generator macros.
	Runs the pattern generator.
	Runs the pattern generator repetitively.
	Stops the pattern generator.
	Steps through the pattern generator vector sequence.
	Resumes pattern generator vector sequence output.
	Resets the pattern generator vector sequence to the beginning.

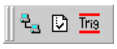
See Also
- "Using the Pattern Generator" (in the online help)

Analyzer Setup Toolbar

Menu	Description
	Bus/Signal - Accesses the Buses/Signals (see page 395) tab of the Setup dialog.
	Timing/State (Sampling) - Accesses the Sampling (see page 396) tab of the Setup dialog.
	Advanced Trigger - Accesses the Advanced Trigger (see page 392) dialog.

See Also
- To create a custom toolbar (see page 375)

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)

Probes Toolbar

Menu	Description
	Opens the properties dialog for a particular probe. · "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 223) - Creates a new marker.
	Go To (see page 226) - Centers the display around the selected marker.
	Creates a new value at a measurement (see page 230).
	Creates a new time interval measurement (see page 228).

NOTE The following are optional markers toolbar icons.

Menu	Description
	Place Maker (see page 225) - Places a new or selected (existing) marker at the middle of the screen.
	Center About (see page 226) - Centers the display around two selected markers.
	Creates a new sample interval measurement (see page 229).
	Hides or shows the marker measurement display bar.
	Accesses the markers properties (see page 232) dialog.

See Also
- To create a custom toolbar (see page 375)

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 375)

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 375)
- To remove icons from a toolbar (see page 375)
- To create a new toolbar (see page 375)
- To restore a toolbar to its original icons (see page 375)

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.

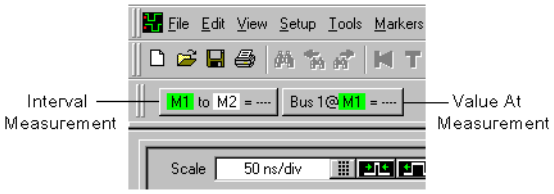
Once you have created a new toolbar, you can add or remove icons as described above.

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.

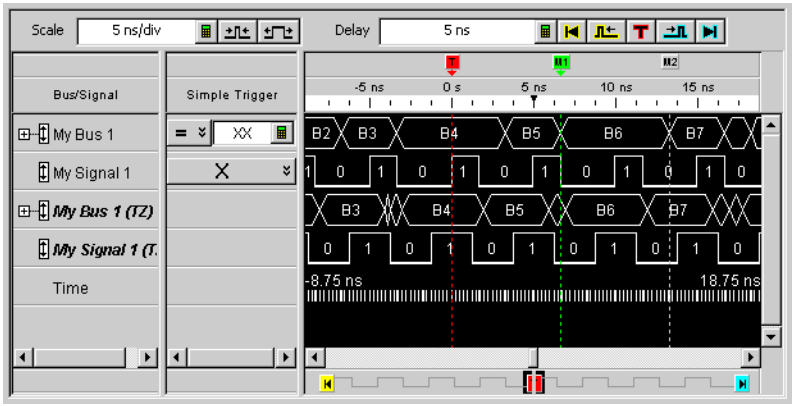


- To create a new time interval measurement (see page 228)
- To create a new sample interval measurement (see page 229)
- To create a new value at measurement (see page 230)
- To hide/show measurement display bar (see page 364)

Windows

- Waveform Display Window (see page 377)
  - Markers Display Bar (see page 380)
  - Markers Overview Bar (see page 381)
- Listing Display Window (see page 382)
- Compare Display Window (see page 384)
- Source Display Window (see page 385)
- Protocol Viewer Display Window (see page 386)
- Overview Window (see page 389)

Waveform Display Window



The Waveform window is accessed through the menu bar's **Window>Waveform**. If you have Tabbed Windows (see page 289) turned on, you can also select a tab at the bottom of the window.

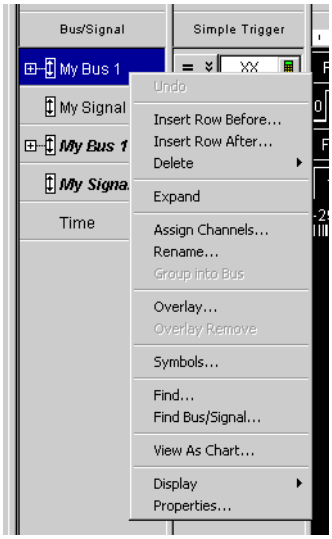
The Waveform window displays captured data as a digital waveform. You can configure the window to display selected buses and signals with time or pattern markers in the data. You can also set up bus pattern triggers and signal trigger options.

The Waveform window consists of the following areas:

- Bus/Signal Configuration (see page 377)
- Simple Trigger (see page 132)
- Markers Display Bar (see page 380)
- Waveform Display Area (see page 379)
- Markers Overview Bar (see page 381)
- Delay Controls (see page 199)
- Scale (time/division) Controls (see page 198)

Bus/Signal Configuration

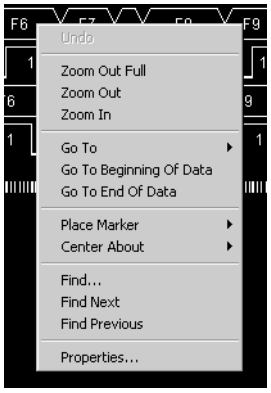
To access the following Bus/Signal configuration options, right-click on any bus or signal name in the Bus/Signal column.



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 204)	Groups highlighted signals into a bus.
Overlay... (see page 200)	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 433) for setting up symbols for the selected bus/signal.
Find...	Opens the Find dialog for searching the captured data (see page 240).
Find Bus/Signal...	Searches for a bus/signal row.
View As Chart... (see page 201)	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 203).
Properties... (see page 204)	Access to properties dialog for waveform window, bus/signal row, bus/signal column, and marker properties.

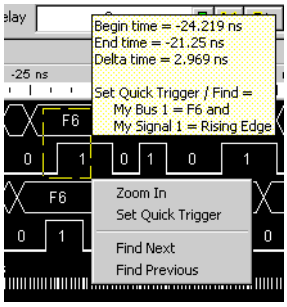
Waveform Display Area

To access waveform display options, right-click anywhere in the display area.



Status	Description
Undo	Undo the last action performed.
Zoom Out (see page 198)	
Zoom In (see page 198)	
Go To (see page 199)	
Place Marker› (see page 224)	
Center About›	Centers the display about a marker pair (see page 226) or waveform edges (see page 198).
Find... (see page 240)	
Find Next (see page 240)	
Find Previous (see page 240)	
Properties... (see page 204)	

Drawing Rectangle in Data

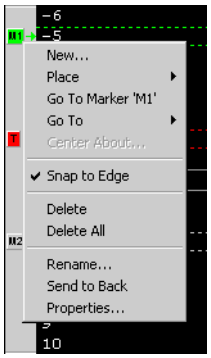


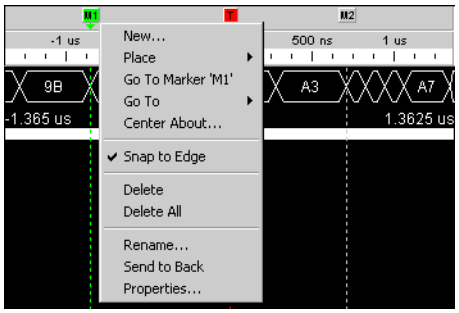
Menu	Description
Zoom In (see page 198)	
Set Quick Trigger (see page 129)	Alternative way to set a Simple Trigger.
Find Next	Data value on left edge of rectangle becomes Find search (see page 240) criteria and next occurrence of that data value is placed at center screen.
Find Previous	Data value on left edge of rectangle becomes Find search (see page 240) criteria and previous occurrence of that data value is placed at center screen.

See Also

- Analyzing Waveform Data (see page 197)
- Marking, and Measuring Between, Data Points (see page 222)
- Specifying Advanced Triggers (see page 138)
- Setting Up Symbols (see page 118)
- To add or delete display windows (see page 289)
- To turn window tabs on/off (see page 289)

Markers Display Bar



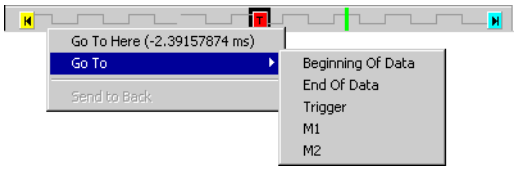


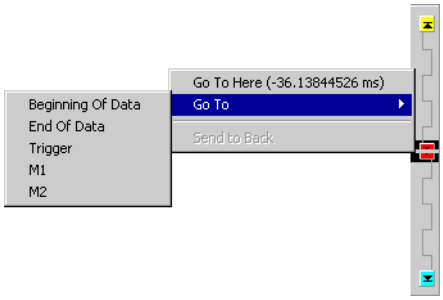
To access these tasks, right-click anywhere in the marker display bar.

Menu	Description
New... (see page 223)	To create new markers.
Place (see page 224)	To place markers in data.
Go To (see page 226)	To go to a marker.
Center About... (see page 226)	To center the display about a marker pair.
Snap to Edge (see page 227)	To toggle a marker's snap to edge property.
Delete (see page 227)	To delete a marker.
Delete All (see page 227)	To delete all markers.
Rename... (see page 231)	To rename a marker.
Send to Back (see page 231)	To send a marker to the back.
Properties... (see page 232)	To set marker properties.

See Also
- To read the markers display and overview bars (see page 222)
- Markers Menu (see page 364)
- Markers Toolbar (see page 373)
- Markers Overview Bar (see page 381)

Markers Overview Bar





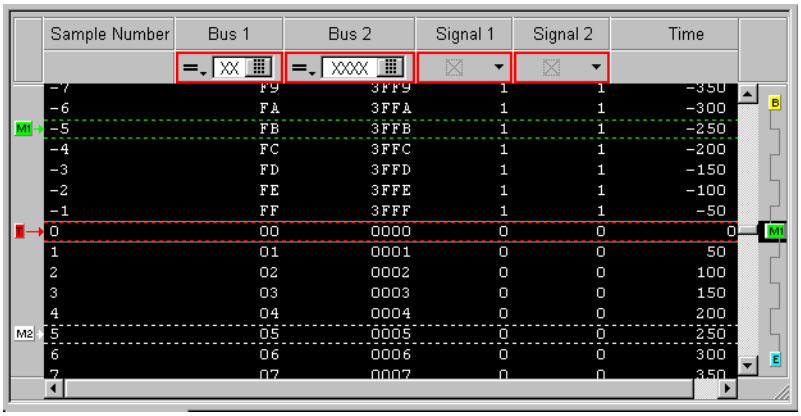
To access these menus, click anywhere in the marker overview bar.

Menu	Description
Go To (see page 226)	To go to a marker.
Send to back (see page 231)	To send a marker to the back.

See Also

- To read the markers display and overview bars (see page 222)
- Markers menu (see page 364)
- Markers toolbar (see page 373)

Listing Display Window



The Listing window is accessed through the menu bar's **Window>Listing**. If you have Tabbed Windows (see page 289) turned on, you can also select a tab at the bottom of the window.

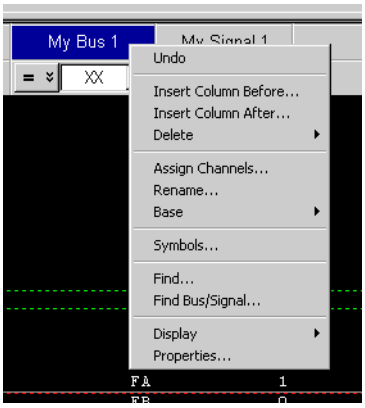
The Listing window displays your captured data as a state listing. You configure the window to display selected buses and signals in columns. Within the listed data, you can insert time or pattern markers. You can also configure the bus pattern triggers and signal trigger options.

The Listing window consists of the following areas:

- Column Configuration (see page 382)
- Simple Trigger (see page 132)
- Markers Display Bar (see page 380)
- Listing Display Area (see page 383)
- Markers Overview Bar (see page 381)

Column Configuration

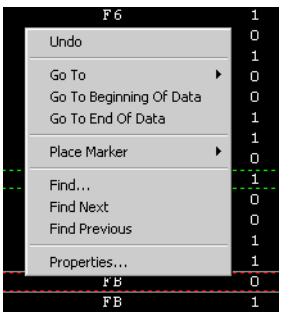
To access the following column configuration options, right-click on any bus or signal name in the column head.



Menu	Description
Undo	Undo the last action performed.
Insert Column Before... (see page 215)	
Insert Column After... (see page 215)	
Delete> (see page 215)	
Assign Channels... (see page 93)	
Rename...	Access a keypad to rename the highlighted bus/signal.
Base> (see page 218)	
Symbols...	Opens the Symbols dialog (see page 433) for setting up symbols for the selected bus/signal.
Find...	Opens the Find dialog for searching the captured data (see page 240).
Find Bus/Signal...	Searches for a bus/signal column.
Display>	Lets you show or hide parts of the Listing window (see page 214).
Properties... (see page 215)	Access to properties dialog for Listing window, bus/signal column, and marker properties.

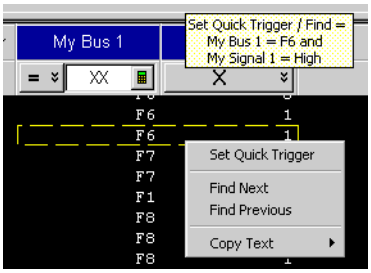
Listing Display Area

To access the Listing display options, right-click anywhere in the display area.



Menu	Description
Undo	Same as Edit>Undo (see page 359).
Go To (see page 213)	
Place Marker> (see page 224)	
Find... (see page 240)	
Find Next (see page 240)	
Find Previous (see page 240)	
Properties... (see page 215)	

Draw Rectangle in Data

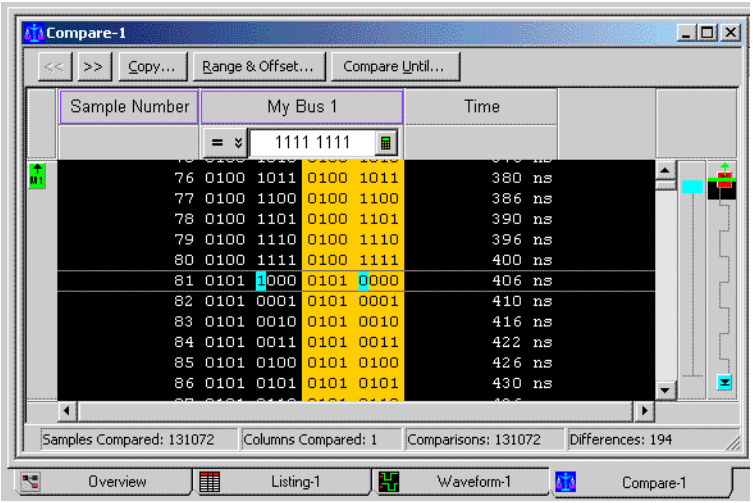


Menu	Description
Set Quick Trigger (see page 129)	Alternative way to set a simple trigger.
Find Next	Data value on top edge of rectangle becomes Find search (see page 240) criteria and next occurrence of that data value is placed at center screen.
Find Previous	Data value on top edge of rectangle becomes Find search (see page 240) criteria and previous occurrence of that data value is placed at center screen.
Copy Text (see page 290)	Copies data as text into the system clip board.

See Also

- Analyzing Listing Data (see page 212)
- Setting Up Symbols (see page 118)
- Marking, and Measuring Between, Data Points (see page 222)
- To add or delete display windows (see page 289)
- To turn window tabs on/off (see page 289)

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.

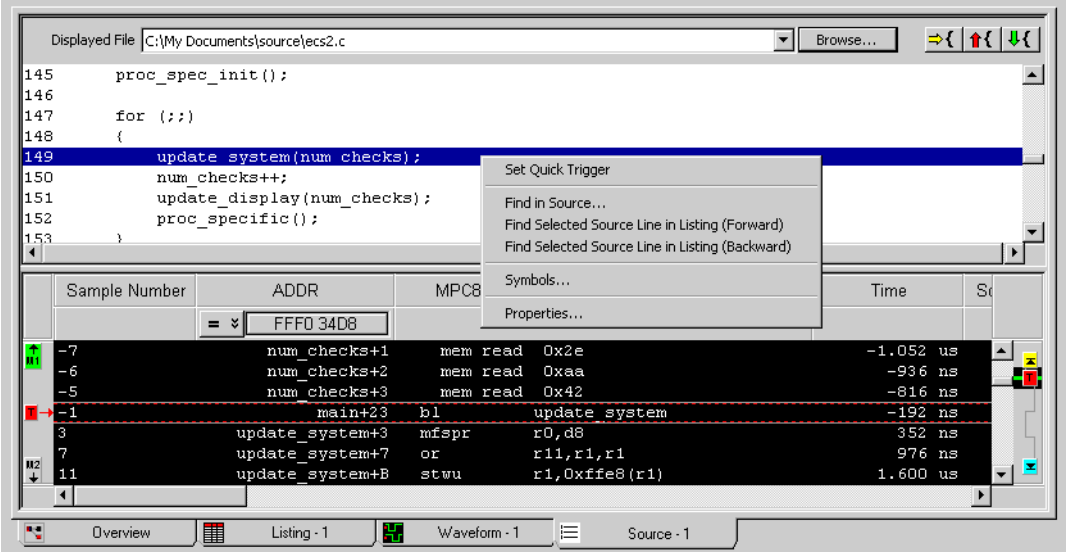
The Compare window is accessed through the menu bar's **Window>Compare**. If you have Tabbed Windows (see page 289) turned on, you can also select a tab at the bottom of the window.

Except for the Compare window's ability to display the differences between captured data and reference data, and its inability display colorized 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 250)
  - To copy data to the reference buffer (see page 250)
  - To find differences in the compared data (see page 250)
  - To compare only a range of samples (see page 251)
  - To offset the reference data (see page 251)
  - To run until a number of compare differences (see page 251)
  - To set Compare window properties (see page 252)
- Analyzing Listing Data (see page 212)

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 289) turned on, you can also select a tab at the bottom of the window.

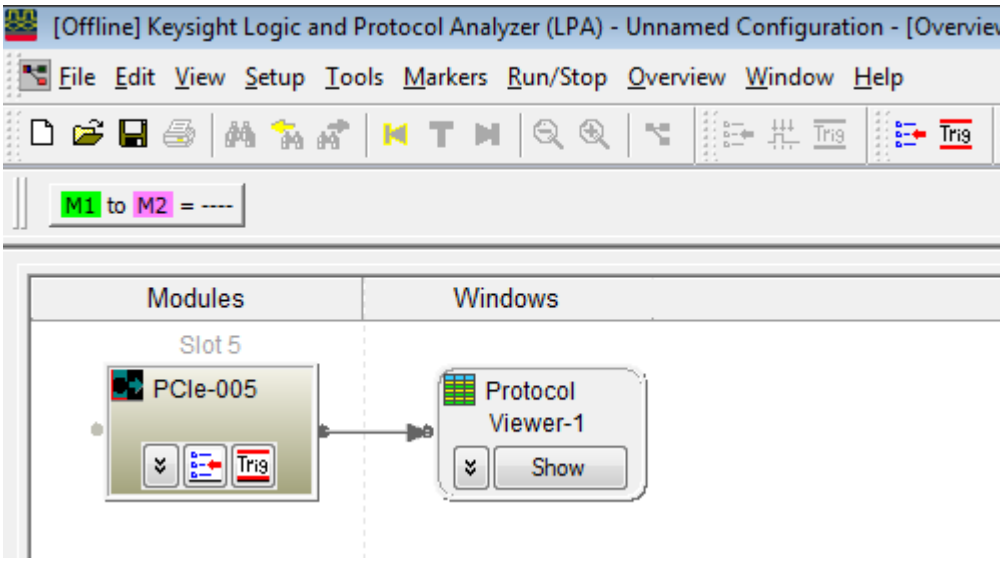
The Source window has two panes: the top pane displays the high-level source code associated with the captured data, and the bottom pane is the same as a Listing window.

See Also

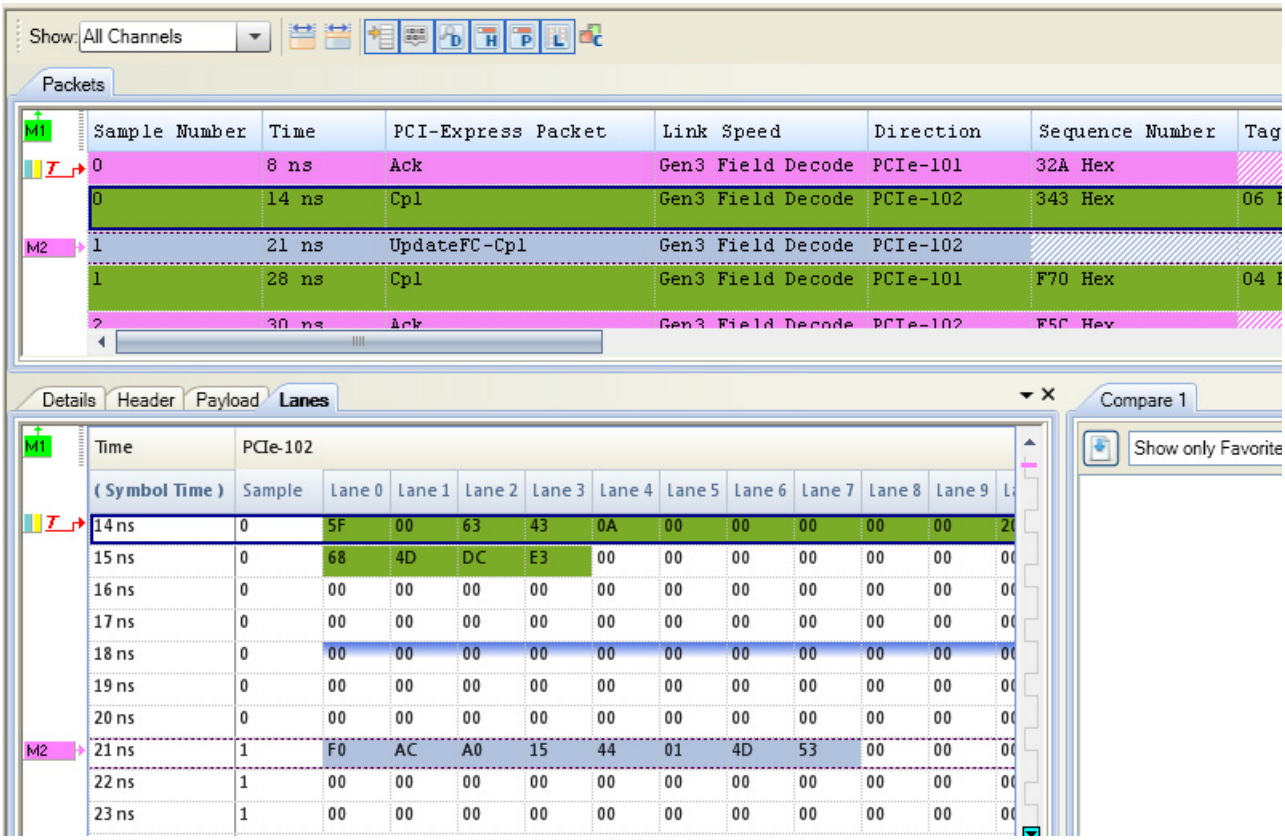
- Viewing Source Code Associated with Captured Data (see page 253)
  - To step through captured data by source lines (see page 253)
  - To go to captured data associated with a source line (see page 254)
  - To browse source files (see page 254)
  - To search for text in source files (see page 255)
  - To set a Quick Trigger in the Source window (see page 131)
  - To go to the source line associated with the listing center rectangle (see page 255)
  - To edit the source code directory list (see page 255)
  - To select the correlation bus (see page 256)
  - Changing Source Window Properties (see page 257)
- Analyzing Listing Data (see page 212)

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 261.

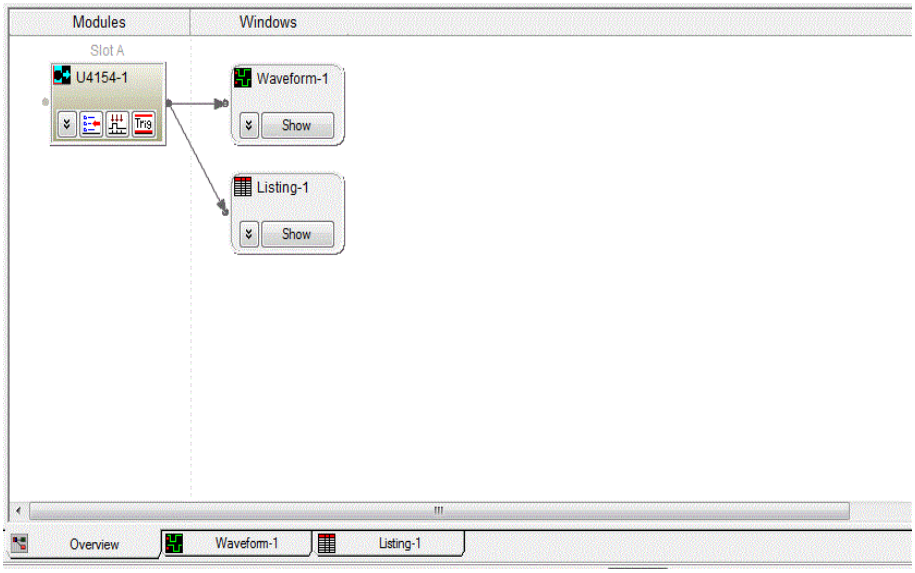
Lower pane of Protocol Viewer

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 270.
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 273.
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 274.
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 275.
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 *U4301A PCIe Gen3 Analyzer module online help*.
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 *U4301A PCIe Gen3 Analyzer module online help*.

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 CSV toolbar button in the Protocol Viewer window.

Overview Window



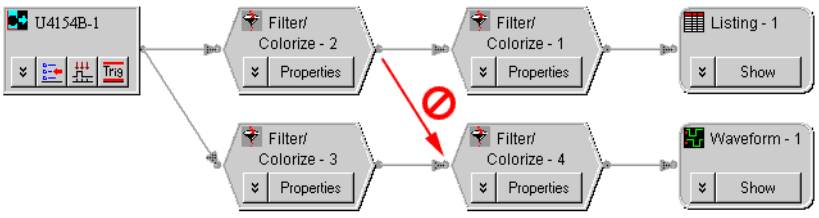
The Overview window lets you specify how the data is sent from the logic analyzer data acquisition module to post-processing tools and display windows.

The Overview window is accessed through the menu bar's **Window>Overview** command. If you have Tabbed Windows (see page 289) turned on, you can also select a tab at the bottom of the window.

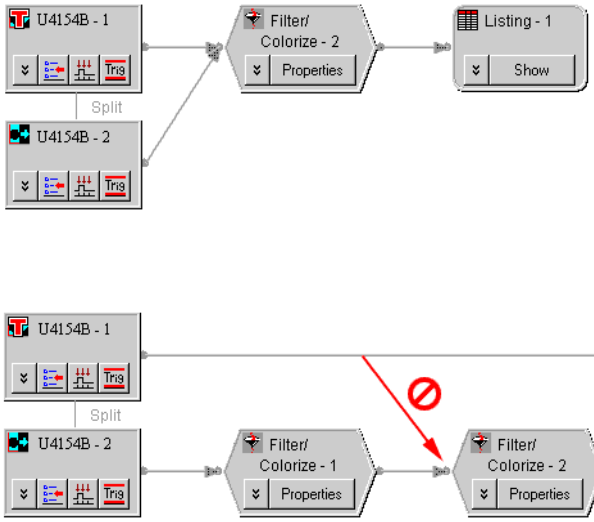
Connection Rules

There are a few rules that govern how you are able to add/connect tools and display data.

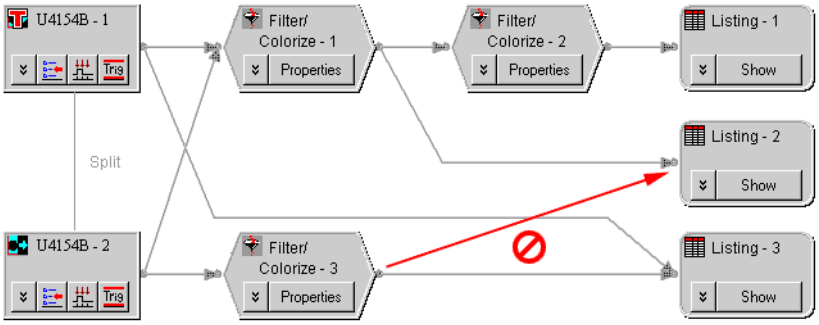
- **Rule 1: Fan out to tools only occurs directly after modules.**



- **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

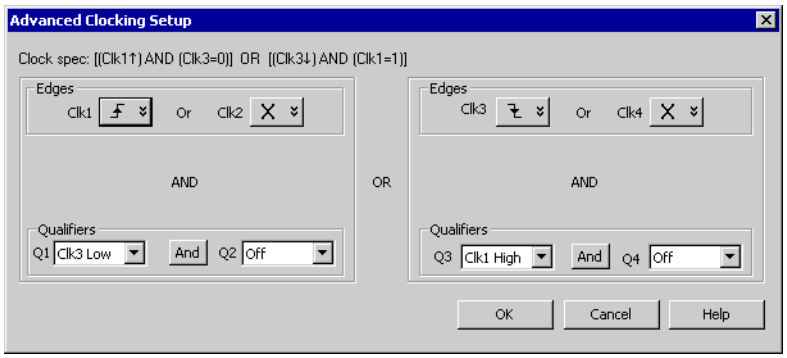
- Analyzing the Same Data in Different Ways (Using the Overview Window) (see page 283)
- To open or display the Overview window (see page 283)
- To add, duplicate, or delete windows and tools (see page 283)
  - To add new windows (see page 284)
  - To delete windows (see page 284)
  - To add new tools (see page 284)
  - To delete tools (see page 285)
  - To duplicate windows (see page 285)
  - To delete connections (see page 285)
  - To add connections (see page 286)
- To edit window or tool properties (see page 286)
- To rename windows, tools, and modules (see page 287)
- To redraw the Overview window (see page 287)
- To delete the Overview window (see page 287)

Dialogs

- Advanced Clocking Setup Dialog (see page 391)
- Advanced Trigger Dialog (see page 392)
- Keysight Logic Analyzer Upgrade Dialog (see page 393)
- Analyzer Setup Dialog (see page 394)
- Chat Dialog (see page 397)
- Chat Select Destination Dialog (see page 398)
- Choose a Protocol Family and Bus Dialog (see page 398)
- Create a New Configuration Dialog (see page 399)
- E-mail Dialog (see page 399)
- Event Editor Dialog (see page 401)
- Export Dialog (see page 401)
- Export File Selection Dialog (see page 402)
- External Application Setup Dialog (see page 403)
- External Trigger Dialog (see page 404)
- Find Dialog (see page 406)
- Frame/Module Information Dialog (see page 407)
- "General Purpose Probe Set Dialog" (in the online help)
- Module Mapping Dialog (see page 409)
- Module Skew and System Trigger Dialog (see page 410)
- Netlist Import Dialog (see page 411)
- Offline Startup Options Dialog (see page 411)
- Options Dialog (see page 412)
- Pod Assignment Dialog (see page 414)
- Printing Data Dialog (see page 415)
- Properties Dialog (see page 415)
- Range Properties Dialog (see page 416)
- Recall Trigger Dialog (see page 417)
- Run Properties Dialog (see page 418)
- Select Symbol Dialog (see page 419)
- Select System to Use Dialog (see page 421)
- Software Licensing Dialog (see page 423)
- Source Viewer Properties Dialog (see page 427)
- Specify Mapping Dialog (see page 429)
- Split Analyzer Setup Dialog (see page 429)
- Status Dialog (see page 430)
- Symbols Dialog (see page 433)
- System Summary Dialog (see page 434)
- Target Control Port Dialog (see page 437)
- TimingZoom Setup Dialog (see page 438)

Advanced Clocking Setup Dialog

The Advanced Clocking Setup dialog lets you specify more complex clock setups than you can with the normal Master or Slave selections. 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.

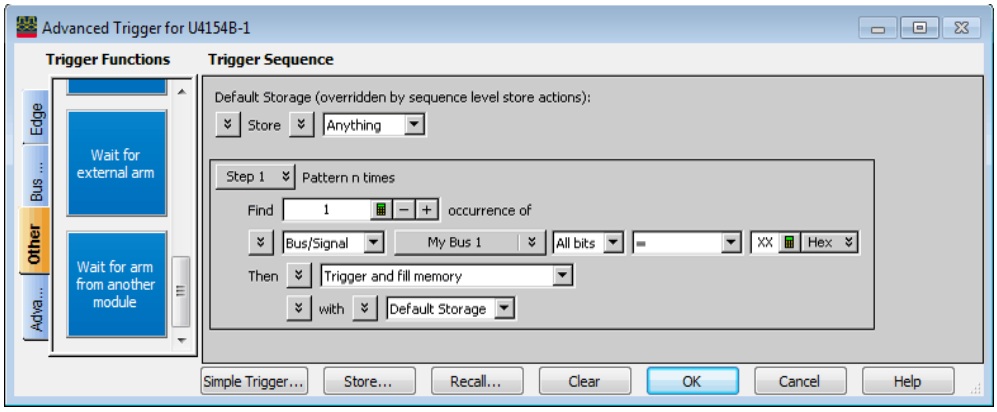


Menu	Description
Clock spec:	A textual description of the clocking setup.
Edges	Lets you choose from **Don't Care, Rising Edge, Falling Edge, or Both Edges** for each of the available clock inputs.
Qualifiers	Lets you turn **Off** clock qualifiers or select **Low** or **High** levels from the available clock inputs for each of the clock qualifier resources (Q1-Q4).
And/Or	Lets you toggle the boolean operators for the clock qualifiers.

See Also
- To set up advanced clocking (see page 114)
- Pod and Channel Naming Conventions (see page 333)

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 *events* to look for in the sampled data and the *actions* 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. *Default storage* specifies which samples to store unless there are overriding *storage control* actions within the trigger sequence steps.
Simple Trigger...	Opens an information dialog that explains how to specify simple triggers (see page 132).
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 417) 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 138)
- Trigger Functions (see page 440)
- Specifying Simple Triggers (see page 132)
- Storing and Recalling Triggers (see page 172)

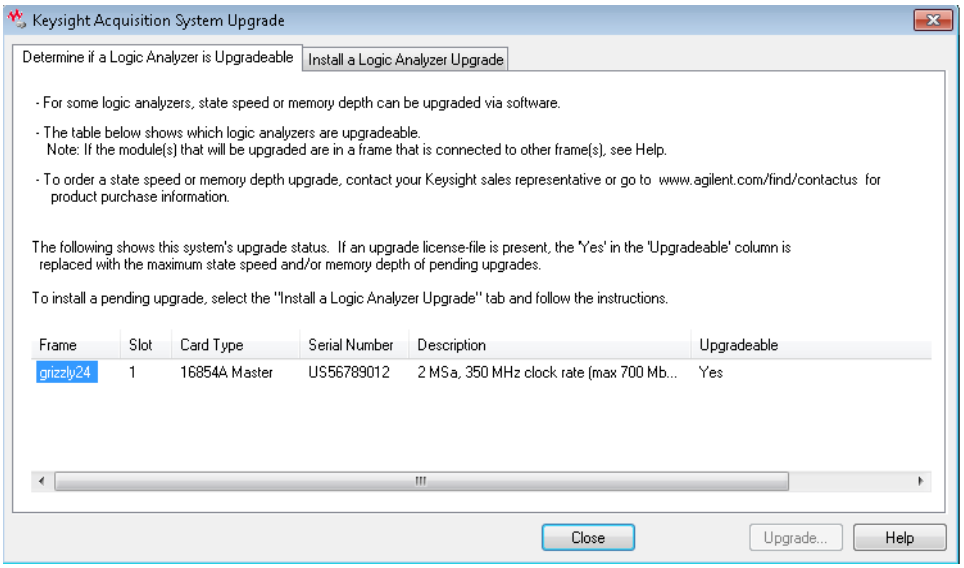
Keysight Logic Analyzer Upgrade Dialog

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:

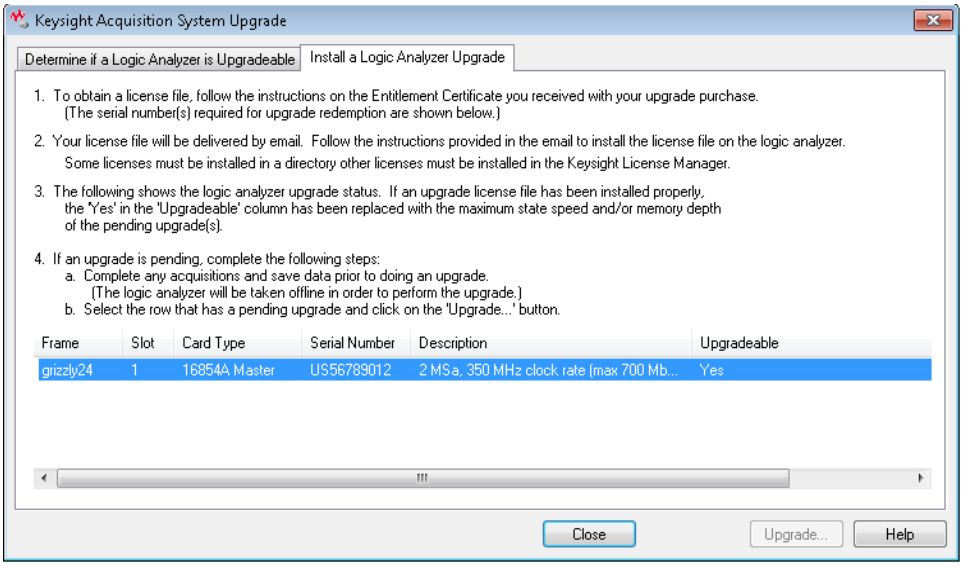
- In a multiframe logic analysis system, you must run the *Hardware Update Utility* program on each frame that has cards to be upgraded.
- You cannot install module upgrades over a remote connection (including remote connections via Remote Desktop, NetOp, or RealVNC).

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

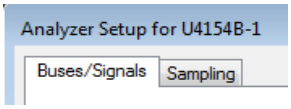


Menu	Description
Upgrade...	Performs the pending upgrade for the selected card.

See Also

- Installing Licensed Hardware Upgrades (see page 124)

Analyzer Setup Dialog



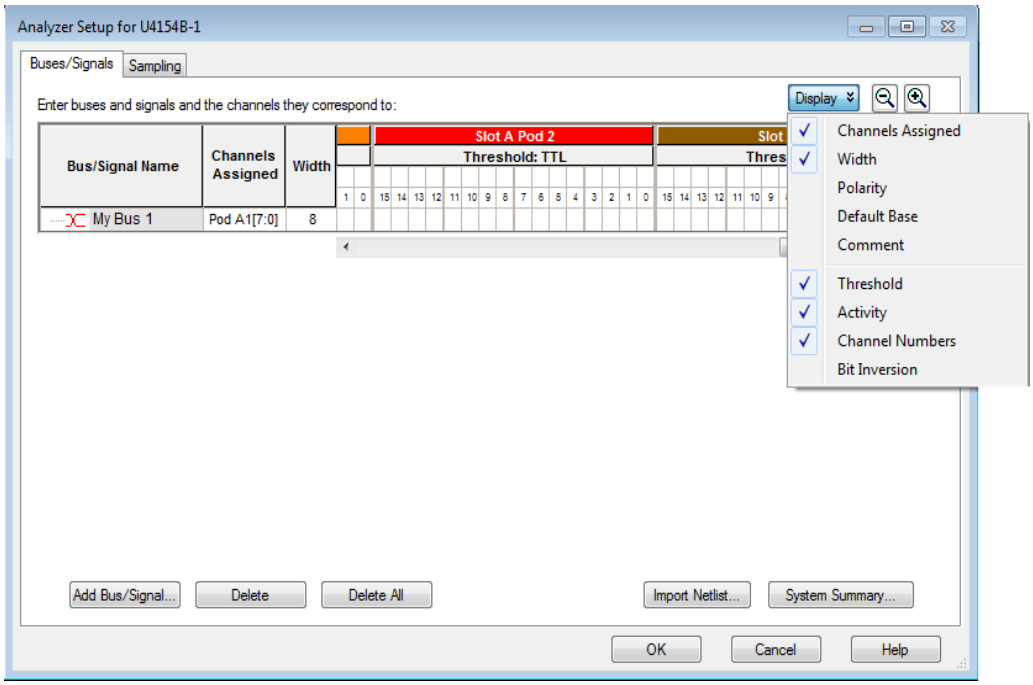
The Analyzer Setup dialog is accessed through the main menu's **Setup>(Logic Analyzer Module)>Bus/Signal...** or **Setup>(Logic Analyzer Module)>Timing/State (Sampling)...** commands.

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 395).
- **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 396).

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. · 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

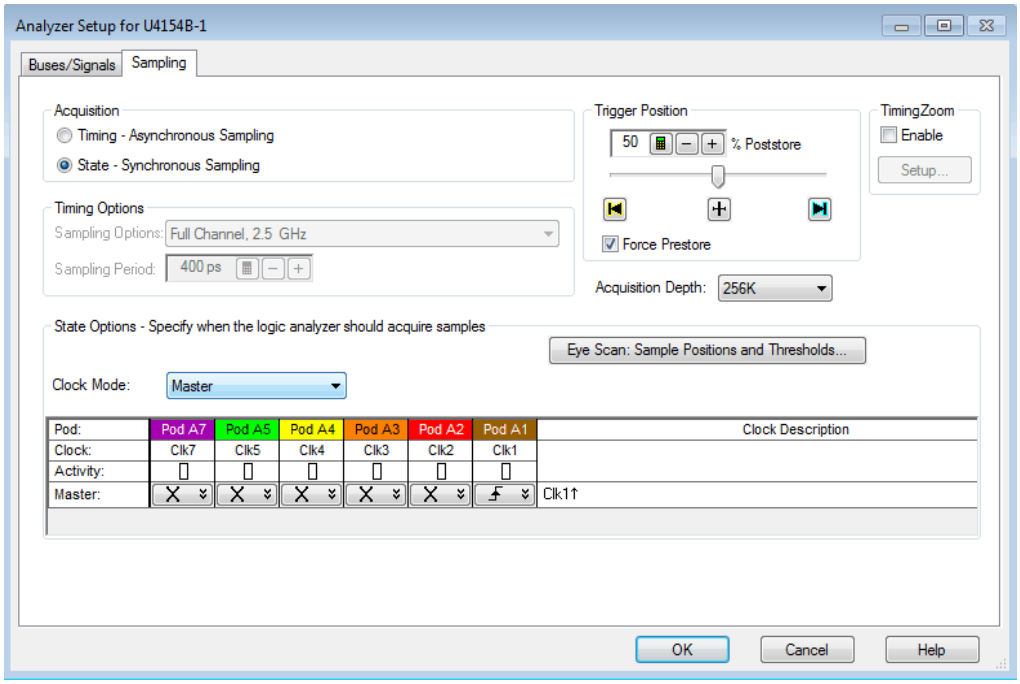
NOTE In previous versions of the *Keysight Logic Analyzer* application, the Buses/Signals setup tab had a Define Probes... button; now, probes are defined differently (see "To define probes" (in the online help)).

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 U4154A/B Logic Analyzers" on page 498.

See Also
- Defining Buses and Signals (see page 93)
  - To add a new bus or signal (see page 94)
  - To delete a bus or signal (see page 95)
  - To rename a bus or signal (see page 95)
  - To assign channels in the default bit order (see page 98)
  - To assign channels, selecting the bit order (see page 99)
  - To define buses and signals by importing netlist files (see page 100)
  - To reorder bits by editing the Channels Assigned string (see page 101)
  - To set the default number base (see page 103)
  - To set polarity (see page 104)
  - To add user comments (see page 104)
  - To add a folder (see page 105)
  - To alias a bus/signal name (see page 105)
  - To sort bus/signal names (see page 105)
- Pod and Channel Naming Conventions (see page 333)
- Why Are Pods Missing? (see page 334)
- Logic Analyzer Notes (see page 492)

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.



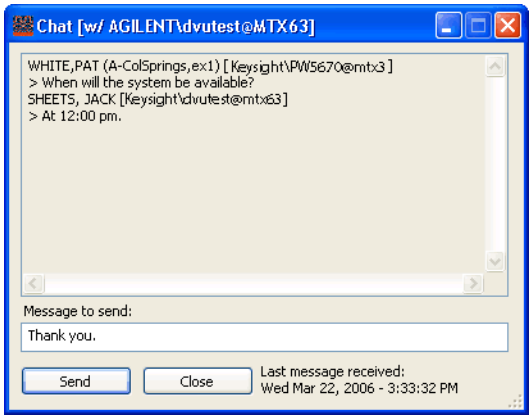
Menu	Description
Acquisition	Lets you select the Timing or State acquisition mode (see Choosing the Sampling Mode (see page 106)).
Timing Options	When the Timing acquisition mode is selected, you can specify its options (see Selecting the Timing Mode (Asynchronous Sampling) (see page 106)).
State Options	When the State acquisition mode is selected, you can specify its options (see Selecting the State Mode (Synchronous Sampling) (see page 108)).
Options	Lets you specify options that apply to both the Timing and State acquisition modes (see To specify the trigger position (see page 115) and To set acquisition memory depth (see page 115)).
TimingZoom	Lets you turn the timing zoom feature on or off and specify its settings (see Using Timing Zoom (see page 116)).

See Also

- Choosing the Sampling Mode (see page 106)
- Logic Analyzer Notes (see page 492)

Chat Dialog

The Chat dialog lets you enter and send messages to other logic analysis system users.

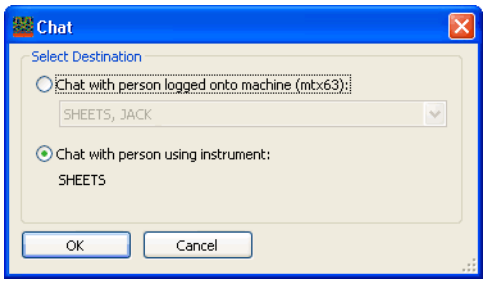


Menu	Description
Message to send	Lets you enter a message to send.
Send	Sends the message.
Close	Closes the dialog and the chat session.

See Also
- Select System to Use Dialog (see page 421)
- Chat Select Destination Dialog (see page 398)

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.

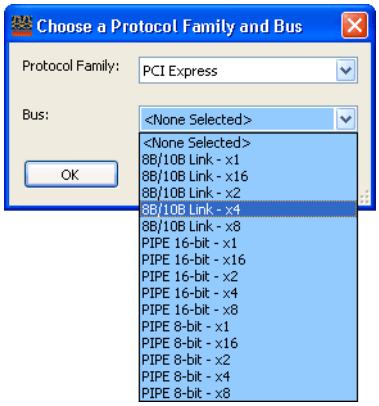


Clicking **OK** opens the Chat dialog (see page 397) where you can enter and send your message.

See Also
- Select System to Use Dialog (see page 421)

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.

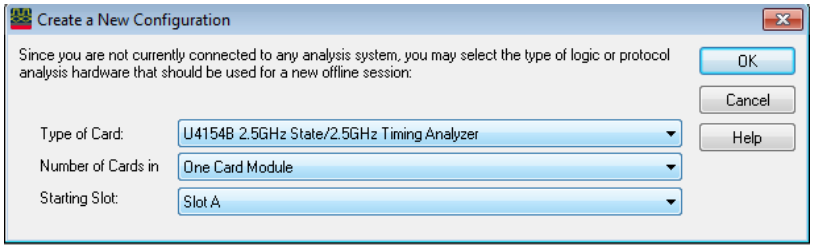


Menu	Description
Protocol Family	Lets you select the protocol family.
Bus	Lets you select the type of bus within the protocol family.

See Also
- Find a packet (see page 458) trigger function
- To specify packet events (in "Find a packet" trigger function) (see page 148)

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.

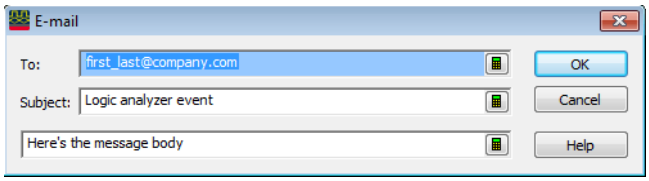
See Also
- Offline Analysis (see page 194)
- To set up multiple-modules with XML-format configurations (see page 191)

E-mail Dialog

NOTE

E-mail must be set up on the computer running the *Keysight Logic Analyzer* application before this feature will work. Refer to the operating system online help or your mail application's online help for information on setting up e-mail. See also Setting Up Outlook Express on the Logic Analysis System (see page 400).

Some mail applications block e-mail that is sent by running programs, so test this feature with your mail application before you use it.



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.

See Also

- To specify a trigger sequence step's goto or trigger action (see page 150)
- To create a new time interval measurement (see page 228)
- To create a new sample interval measurement (see page 229)
- To change a marker's position property (see page 225)
- To run until a number of compare differences (see page 251)

Setting Up Outlook Express on the Logic Analysis System

Outlook Express is installed on logic analyzers and logic analysis systems along with Internet Explorer and the Windows XP operating system. If you want to use the logic analysis system's "e-mail on" features (on trigger, on measured interval, on number of compare differences), you can use Outlook Express.

To set up Outlook Express on the logic analysis system:

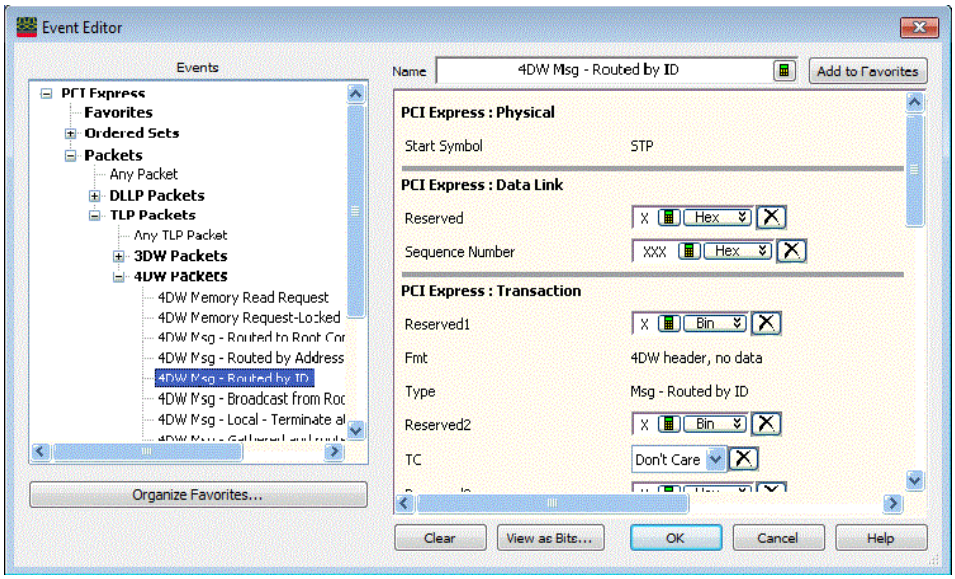
1 Make Outlook Express the default e-mail program in Internet Explorer:
   a In Internet Explorer, choose **Tools>Internet Options...**.
   b In the Internet Options dialog's Programs tab, for the **E-mail** service, select **Outlook Express**.
2 Get e-mail server and account information from your network or system administrator, and configure a mail account in Outlook Express:
   a In Outlook Express, choose **Tools>Accounts...**.
   b In the Internet Accounts dialog's Mail tab, click **Add>Mail...**.
   c In the **Internet Connection Wizard**, follow the instructions, using the e-mail account information from your system or network administrator.
3 Make sure the Security tab settings are set correctly:
   a In Outlook Express, choose **Tools>Options...**.
   b In the Options dialog's Security tab, uncheck **Warn me when other applications try to send mail as me**.

4 Send a test e-mail from Outlook Express.
5 Test one of the logic analysis system's "e-mail on" features.

See Also
- E-mail Dialog (see page 399)

Event Editor Dialog

The Event Editor dialog lets you specify packet events in the "Find a packet" trigger function.

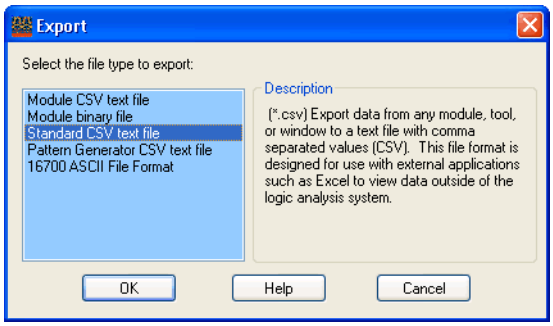


Menu	Description
Events	Hierarchically lists protocol event types that you can select.
(Fields)	(On the right side of the dialog). Lets you enter, select, or clear ( ) packet field values.
Name	Lets you enter or modify the name of the packet event.
Add to Favorites	Adds the packet event to the favorites list.
Organize Favorites...	Opens the Organize Favorites dialog for organizing the packet event favorites list (see To organize favorite packet events (see page 149)).
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 149)).

See Also
- Using the Packet Event Editor (see page 148)
- Find a packet (see page 458) trigger function
- To specify packet events (in "Find a packet" trigger function) (see page 148)
- To find packet patterns in the captured data (see page 243)
- "To specify packet patterns to filter" (in the online help)

Export Dialog

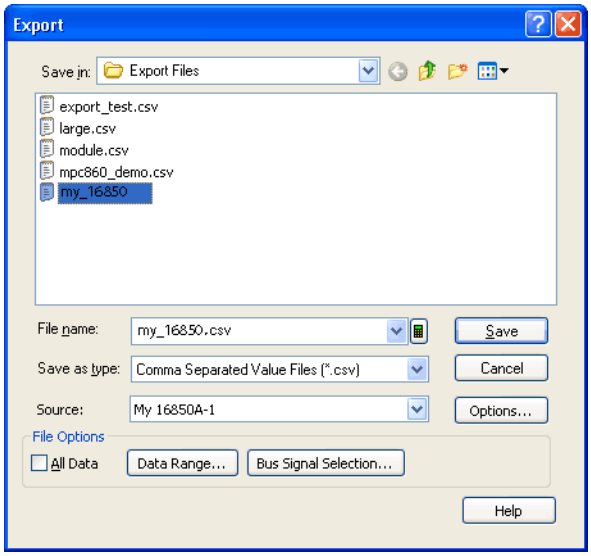
The Export dialog lets you export certain kinds of data into the *Keysight Logic Analyzer* application.



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 179).
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 181).
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 177).
Pattern Generator CSV text file	Lets you export pattern generator vector sequences to a comma separated value (CSV) format file (see "Exporting Vector Sequences to CSV Format Files" (in the online help)).
16700 ASCII File Format	Lets you export captured data to a text file that matches the output of the 16700 logic analysis system's "Print to File" format (see "To export data to 16700 ASCII format files" on page 183).

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.

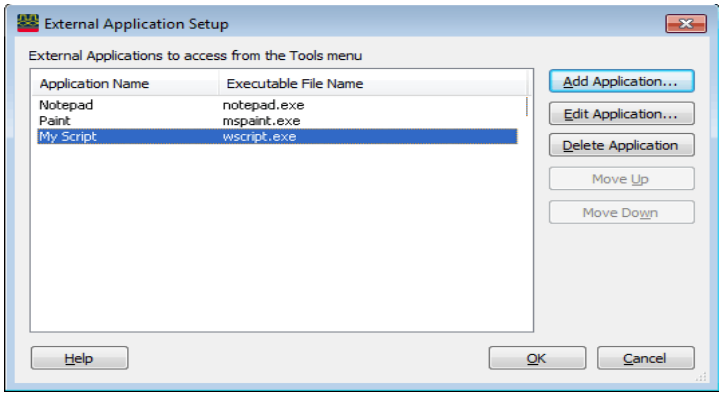


Menu	Description
File name	Lets you select the file to which data is exported.
Save as type	The appropriate type is already selected, based on the file type selected in the previous Export dialog (see page 401).
Source	Lets you select the data source. The list of sources you can select from is determined by the file type selected in the previous Export dialog (see page 401).
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 401).
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 177)
- To export data to module CSV format files (see page 179)
- To export data to module binary (ALB) format files (see page 181)
- "Exporting Vector Sequences to CSV Format Files" (in the online help)

External Application Setup Dialog

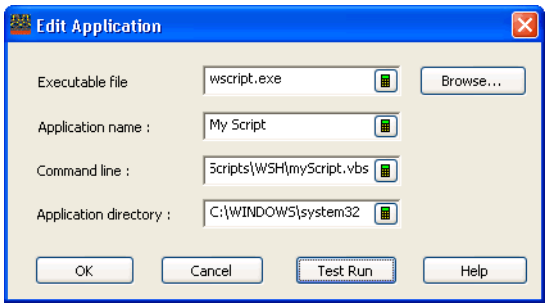
The External Application Setup dialog lets you add, edit, arrange, or remove items from the **Tools>External Applications>** menu.



Menu	Description
Add Application...	Opens the Add Application dialog (see page 404) for specifying a new menu item's parameters.
Edit Application...	Opens the Edit Application dialog (see page 404) 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.



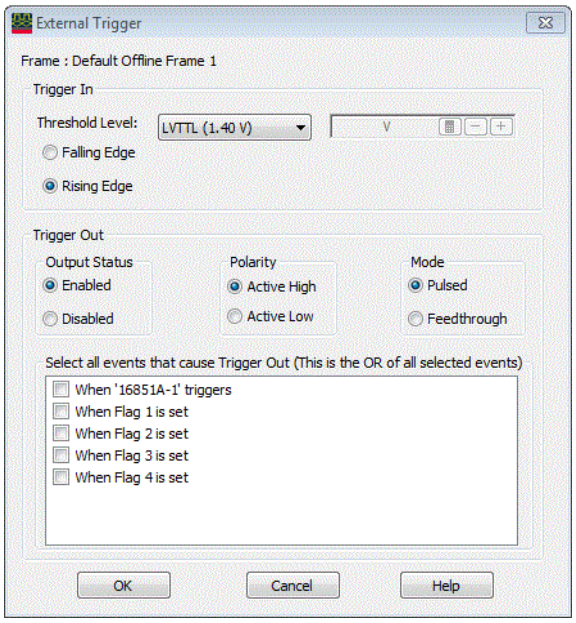
Menu	Description
Executable file	The name of the file to execute when the menu item is selected.
Application name	The name that appears on the menu item.
Command line	Command line options for the executable file.
Application directory	The directory that contains the executable file.
Test Run	For quick 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.

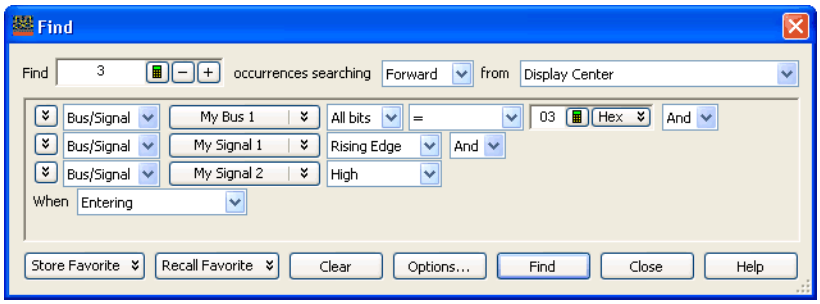
Output Signal Characteristic	16850-Series
VOH (output high level):	>2.0 V (3.3 V avg.)
VOL (output low level):	<0.5 V (0 V avg.)
Pulse width:	Approx. 80-160 ns

See Also

- To trigger other instruments - trigger out (see page 169)
- To trigger analyzer from another instrument - trigger in (see page 170)

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.



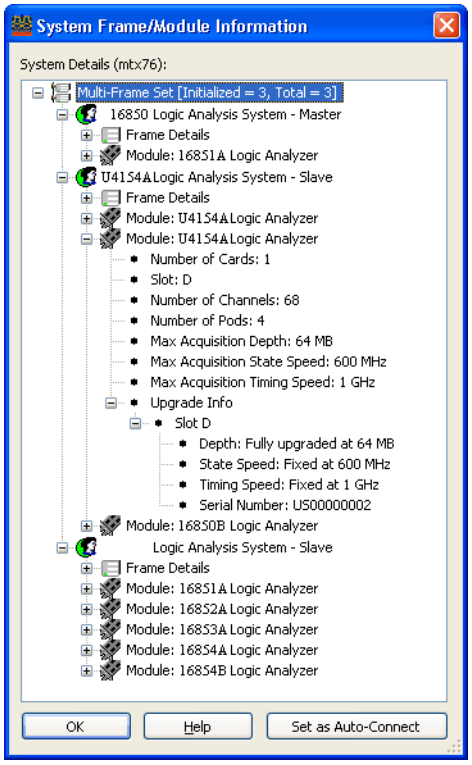
Menu	Description
Find N occurrences	Specifies the number of occurrences to search for.
searching	Specifies whether to search Forward or Backward.
from	Specifies the starting location (Display Center, Beginning Of Data, End Of Data, Trigger, or a marker).
(pattern event)	Specifies the pattern event you 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.

Menu	Description
When	A 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** 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. The other qualifiers (**Present**, **Not Present**, **Entering**, **Exiting**, and **Transitioning**) do not allow a time duration.
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 240)
- To find bus/signal patterns in the captured data (see page 241)
- To find packet patterns in the captured data (see page 243)
- To find complex patterns in the captured data (see page 246)
- To store, recall, or delete favorite find patterns (see page 247)
- To specify "found" marker placement (see page 248)

Frame/Module Information Dialog

The Frame/Module Information dialog displays detailed information about a logic analysis system *frame* (see page 539).



Menu	Description
System Details	Displays the detailed information about the logic analysis system. This includes both information on the frame as a whole and information on each module. 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. If module depth or speed is upgradeable (via hardware licensing), module upgrade information is also provided. 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.
Set as Auto-Connect	Selects the logic analysis system frame as the one to use when the *Keysight Logic Analyzer* 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 421). This button will change to "Clear Auto-Connect" if this frame is already set as an auto-connect.
Clear Auto-Connect	When pressed, specifies that the logic analysis system frame not be automatically connected to when the *Keysight Logic Analyzer* application starts.

Notes:

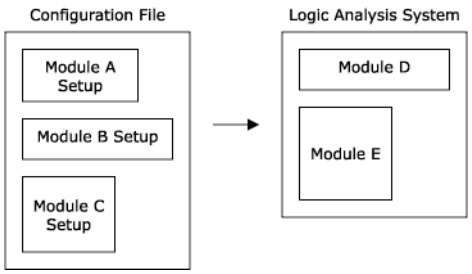
- When no auto-connect is established, the *Keysight Logic Analyzer* application attempts to go online with the local frame or starts offline if there is no local frame.
- The auto-connect frame selection is stored on a per-user basis-so each logged on user can establish a different auto-connect frame.

- 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.

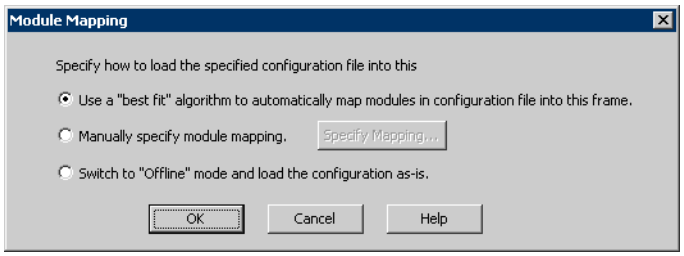
See Also
- To view logic analysis system details (see page 80)

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 409)	Automatically maps configuration file modules into the current frame.
Manually map	The **Specify Mapping...** button opens the Specify Mapping dialog (see page 429) to specify how configuration file modules should be mapped into the current frame.
Offline mode	Loads the configuration file as-is in the *offline mode*.

See Also
- To transfer module setups to/from multi-module systems (see page 190)

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 Is the module split?

2 Look for a module in the Overview window with the same name. If one is found, load the configuration into that module.

3 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.

4 Look for the first module of the same *general* type as the configuration file module.

In this case, "general" means, for example, "logic analyzer" vs. "pattern generator").

5 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:

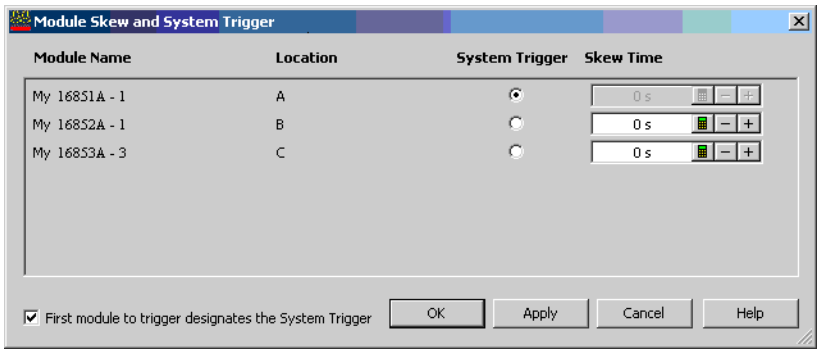
When loading an ALA format configuration file, there is *always* a "clear" performed in the Overview window. Therefore, no matching algorithm is needed.

When loading an XML format configuration file, the XML best-fit algorithm is used.

See Also

- Module Mapping Dialog (see page 409)

Module Skew and System Trigger Dialog



This dialog is available when there are multiple *modules* (see page 540) 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*.

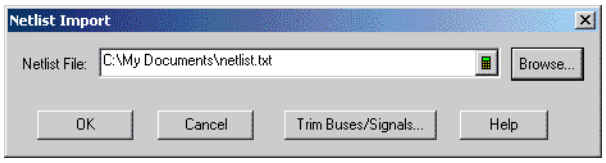
Disabled modules are grayed out.

See Also
- Setting the System Trigger and Skew Between Modules (see page 288)
- To disable and enable modules (see page 88)

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.**

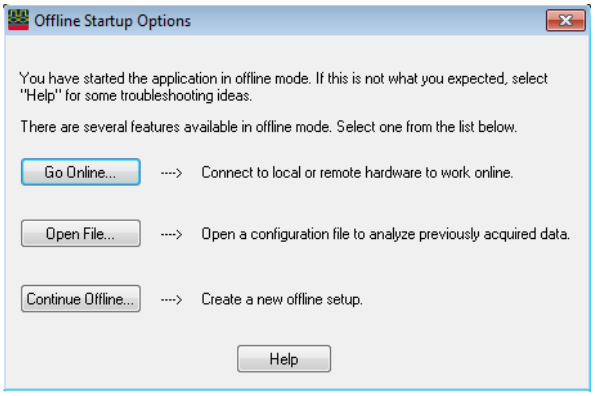


Menu	Description
Netlist File	Lets you enter or browse for the name of the netlist file to import.
Trim Buses/Signals...	Opens the Trim Bus/Signal Names dialog which lets you trim the bus/signal names imported from the netlist file.

See Also
- To define buses and signals by importing netlist files (see page 100)

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 194) (if this is unexpected, see If starting in offline mode is unexpected (see page 304)). This dialog presents options for the tasks you can perform in the offline mode.



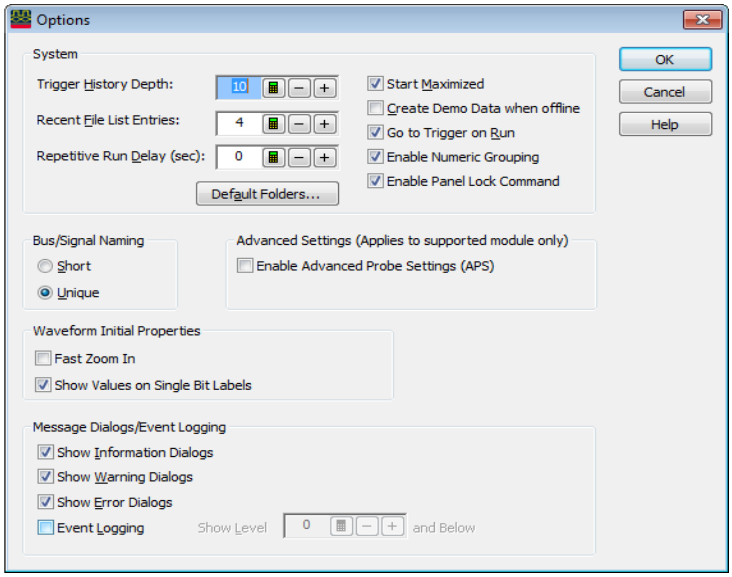
Menu	Description
Go Online...	Opens the Select System to Use dialog (see page 421) for choosing a frame to connect the *Keysight Logic Analyzer* application to.
Open File...	Lets you open a logic analyzer configuration file (see page 188).
Continue Offline...	Opens the Create a New Configuration dialog (see page 399) for creating a new logic analyzer configuration file.

See Also
- If starting in offline mode is unexpected (see page 304)

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 413)
- Bus/Signal Naming Options (see page 413)
- Advanced Settings Options (see page 413)
- Message Dialogs/Event Logging Options (see page 414)



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 172) 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 *Keysight Logic Analyzer* 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 Option

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)	The Advanced Probe Settings are supported on U4154A/B logic analyzers and PCIe Gen3 analyzer. Checking this option enables: · 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 U4154A/B Logic Analyzers" on page 498.

Message Dialogs/Event Logging Options

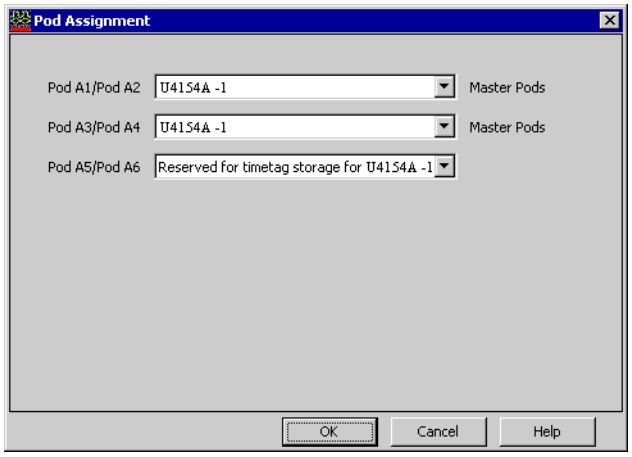
As with any other program, the *Keysight Logic Analyzer* application generates messages about events. You can choose which messages are displayed. Check the appropriate box to indicate you wish the dialogs displayed.

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 through **Help>Status Log**. 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 540) or *pod pair* (see page 540) for time tag storage. When a logic analyzer *module* (see page 540) has been split into two modules, you can also use this dialog to re-assign pods or pod pairs.

If you choose the highest acquisition memory depth (see page 115) 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.

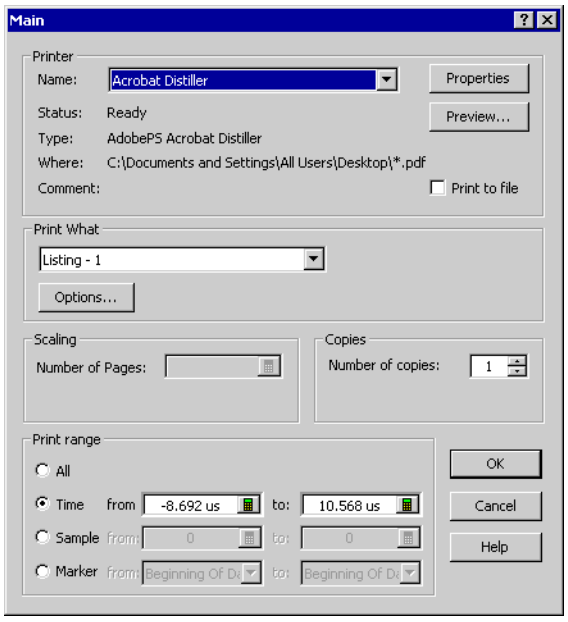
See Also

- Split Analyzer Setup Dialog (see page 429)
- Memory Depth and Channel Count Trade-offs (see page 335)

- Pod and Channel Naming Conventions (see page 333)

Printing Data Dialog

After choosing **File>Print...**, this dialog lets you print the current measurement data from a display window.

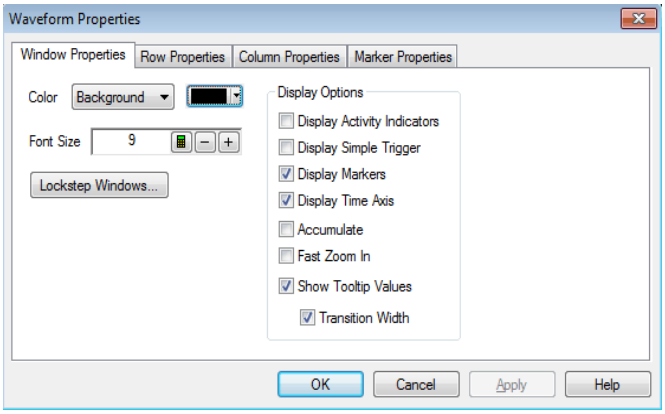


NOTE The first time you access the print dialog, you are asked to install a printer. Follow the directions in the printer install dialogs that appear.

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.
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 290)
- To install a printer (see page 291)
- To connect a LAN (see page 291)

Properties Dialog



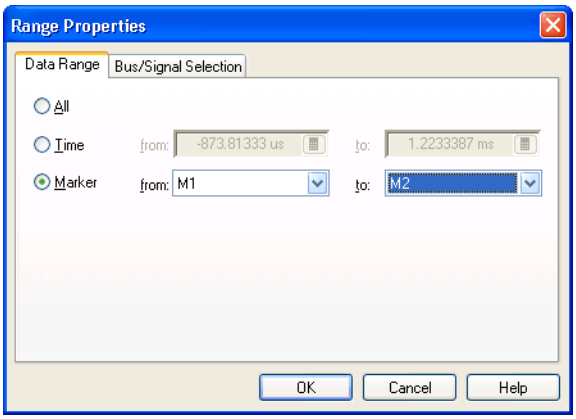
The Properties dialog is accessed through the menu bar's **Window>Properties...**. Use it to set up how the window and the displayed data appear.

- Changing Waveform Window Properties (see page 204)
- Changing Listing Window Properties (see page 215)
- To set Compare window properties (see page 252)
- Changing Source Window Properties (see page 257)
- Changing Marker Properties (see page 232)

Range Properties Dialog

Data Range Tab

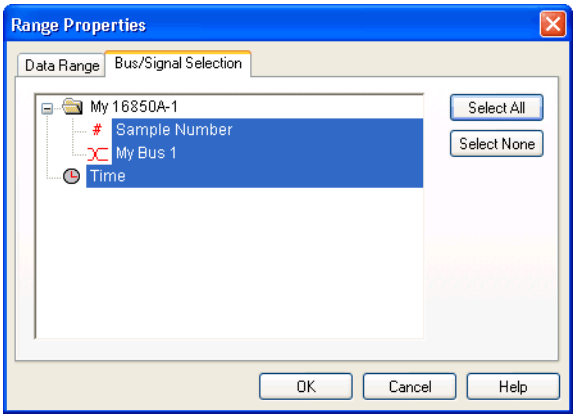
Specifies the range of data to export.



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.

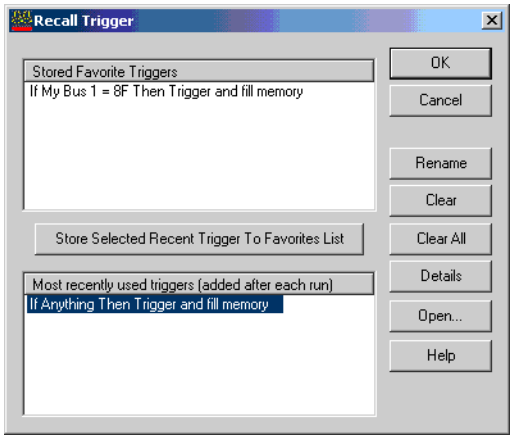
See Also
- To export data to CSV format files (see page 177)

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.

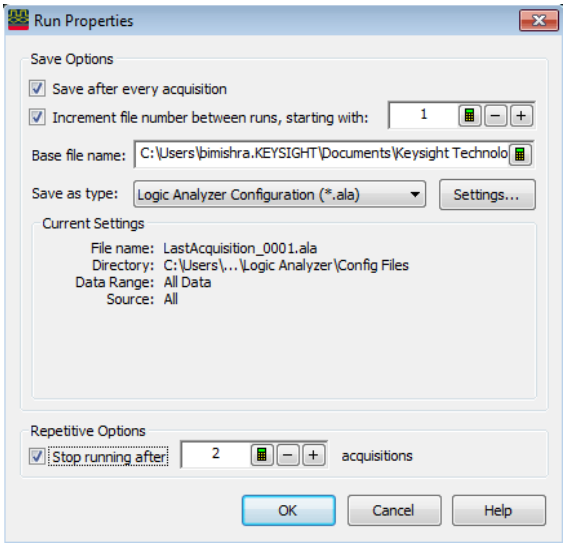


Menu	Description
Rename	Lets you edit the name of the highlighted trigger.
Clear	Clears the highlighted trigger from the list.
Clear All	Clears all triggers from recall lists.
Details	Shows complete definition of the highlighted trigger.
Open...	Lets you recall a trigger that was previously saved to 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 172)
- To recall a trigger (see page 172)
- To set the trigger history depth (see page 173)

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: · 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 176).
Settings...	Opens the Save As dialog (see page 176) 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

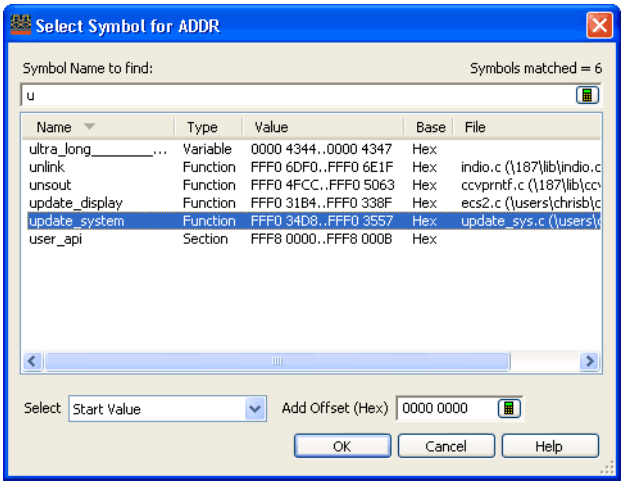
Menu	Description
Stop running after	Lets you stop repetitive runs after a certain number of acquisitions. When enabled, you can enter the number of acquisitions.

See Also

- Running/Stopping Measurements (see page 174)

Select Symbol Dialog

Use the Select Symbol dialog to choose a symbol to use when the numeric base (see page 218) is set to Symbols.



The Select Symbol dialog becomes available when you use the Symbols number base in the following dialogs:

- Find (see page 241)
- "Filter/Colorize" (in the online help)
- Specifying Simple Triggers (see page 132)
- Advanced Trigger Dialog (see page 392)

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: · * (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 118)).
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)	Lets you add an offset to the selected symbol value, for example, in the case where code is relocated.
	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:
	·
	`Symbol Value`
	`---------- ------`
	`RangeSymbol1 00..10`
	`RangeSymbol2 11..20`
	`Symbol3 21`
	`Symbol4 22`
	`Symbol5 FF`
	· 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 118)
- Symbols Dialog (see page 433)

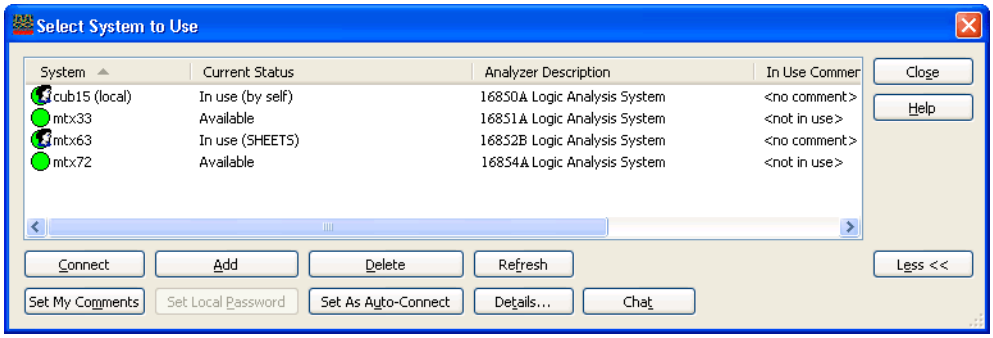
Select System to Use Dialog

The Select System to Use dialog lets you select a logic analysis system *frame* (see page 539) 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 *Keysight Logic Analyzer* 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 84). The icon next to a *remote* 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 *Keysight Logic Analyzer* 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 407) 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 398) 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 75)

Software Licensing Dialog

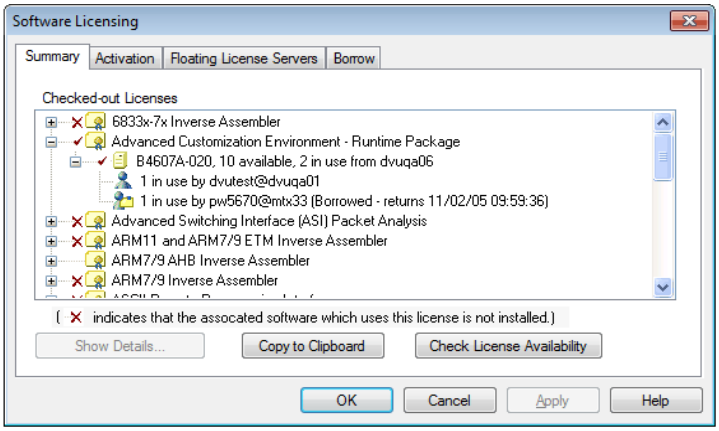
The Software Licensing dialog is used to manage the software licenses used by a logic analysis system. This dialog has four tabs:

- Summary Tab (see page 423)
- Activation Tab (see page 424)
- Floating License Servers Tab (see page 425)
- Borrow Tab (see page 426)

See Also
- Managing Software Licenses (see page 293)

Summary Tab

The Software Licensing dialog's Summary tab displays information about the licenses that can be used in the logic analysis system.



Folders in the **Active Software Licenses** hierarchy show the licensed software that can be used. Red check marks show floating licenses are in use. A red "X" next to a folder shows that the software is not installed.

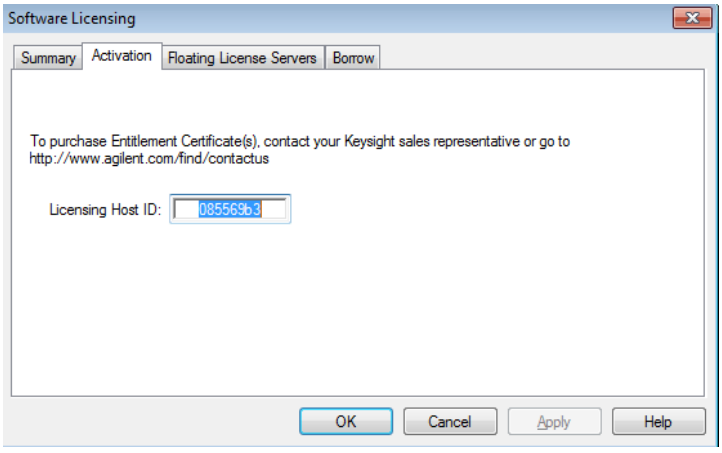
Within a folder, the status of individual licenses is displayed.

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.

See Also
- To view active software license information (see page 294)

Activation Tab

The Software Licensing dialog's Activation tab contains the Licensing Host ID which is needed to activate software licenses.

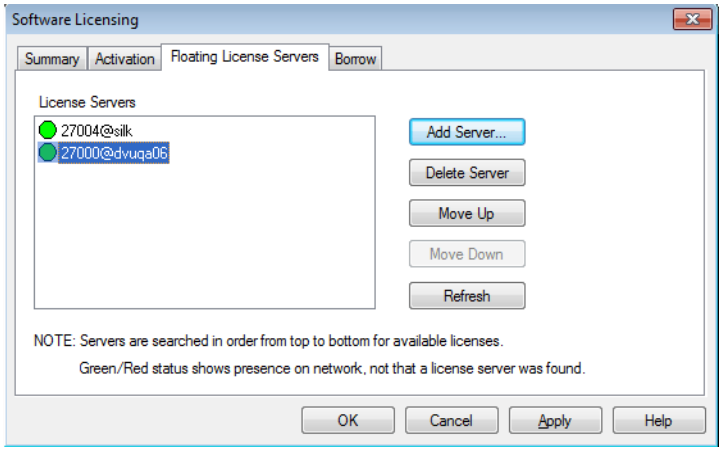


Menu	Description
Licensing Host ID	This ID is used when obtaining license files for software tools.

See Also
- To activate software licenses (see page 295)

Floating License Servers Tab

The Software Licensing dialog's Floating License Servers tab lets you identify and order the servers that floating licenses can be obtained from.



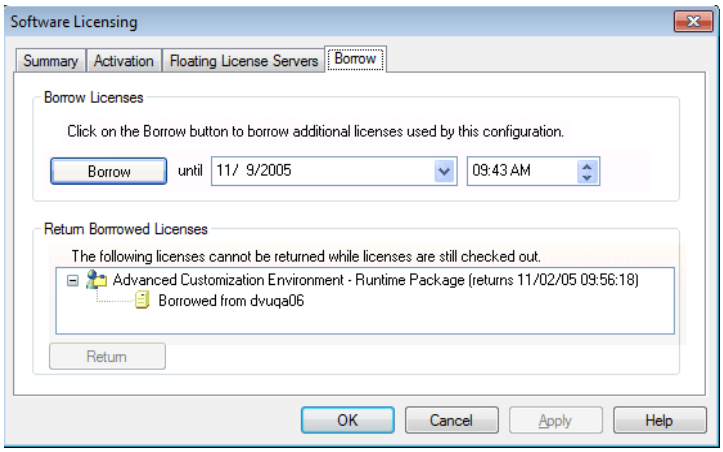
When licenses need to be checked out, the list of **License Servers** is searched in order, and the license is checked out from the first server having an available license.

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 296)

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 426).
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 427).

See Also

- To borrow floating licenses and return them early (see page 298)

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.

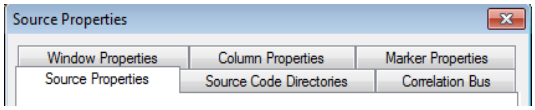
The following licenses cannot be returned while licenses are still checked out.

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



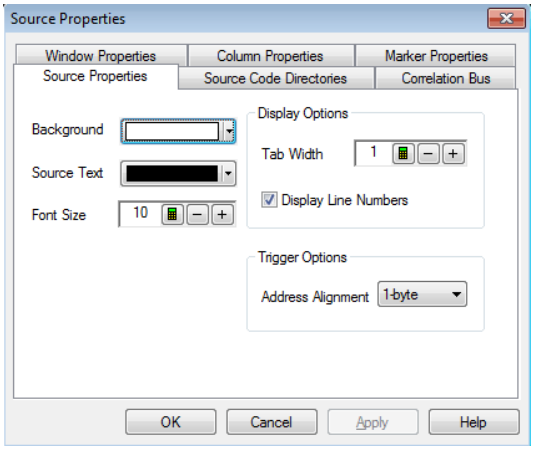
- Source Properties Tab (see page 427)
- Source Code Directories Tab (see page 428)
- Correlation Bus Tab (see page 428)

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 215)
- Changing Bus/Signal Column Properties (see page 217)

Source Properties Tab

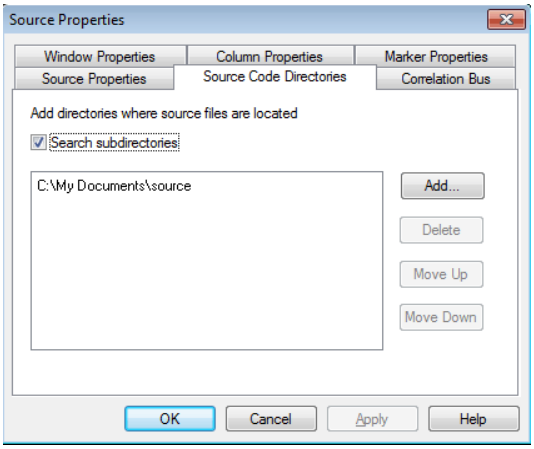
The Source Viewer Properties dialog's Source Properties tab lets you set the background color and font size as well as display options like the number of spaces to use for tabs and whether or not to display line numbers.



See Also
- Changing Source Window Properties (see page 257)
- Source Viewer Properties Dialog (see page 427)

Source Code Directories Tab

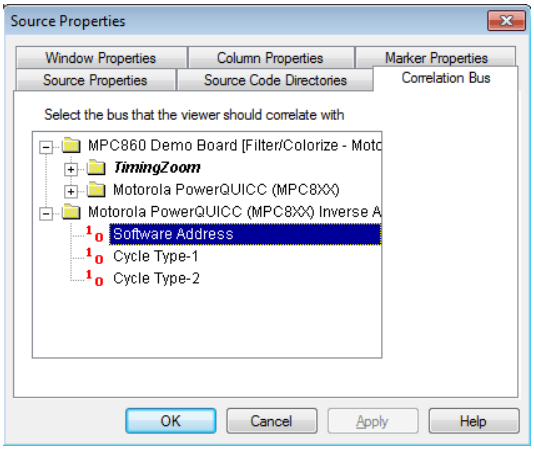
The Source Viewer Properties dialog's Source Code Directories tab lets you specify the directories where the source code is located. This is necessary because source file paths specified in the symbol file may not be valid if you compile on one computer and debug on another. You can specify multiple 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 255)
- Source Viewer Properties Dialog (see page 427)

Correlation Bus Tab

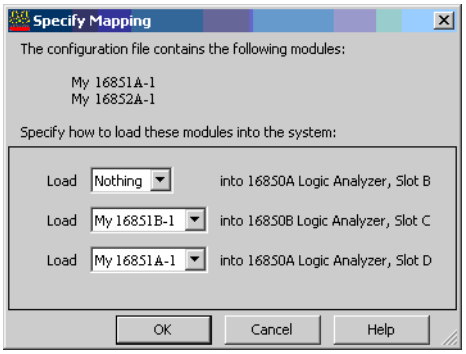
The Source Viewer Properties dialog's Correlation Bus tab lets you specify the bus whose line number symbols will be used for source correlation. Typically, you will select the "software address" bus generated by an inverse assembler tool or another address bus.



See Also
- To select the correlation bus (see page 256)
- Source Viewer Properties Dialog (see page 427)

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 305)).**

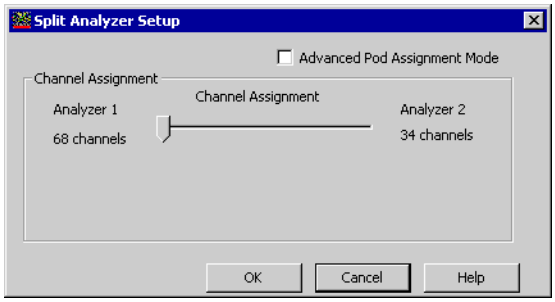
See Also
- Module Mapping Dialog (see page 409)
- To transfer module setups to/from multi-module systems (see page 190)
- ALA vs. XML, When to Use Each Format (see page 352)

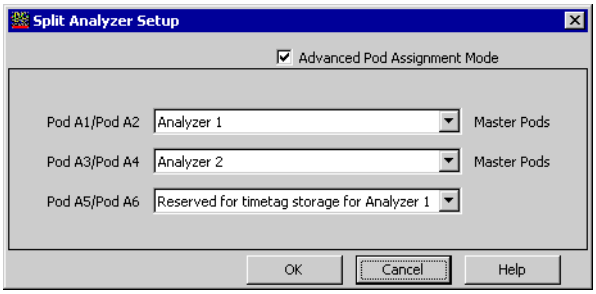
Split Analyzer Setup Dialog

A logic analyzer *module* (see page 540) can be split into two logic analyzer modules, for example, to probe two buses with different clock signals.

NOTE **When you split a logic analyzer module, one of the resulting modules must be in the state sampling mode (they cannot both be in the timing sampling mode).**

The Split Analyzer Setup dialog lets you specify how many channels are used in each module.



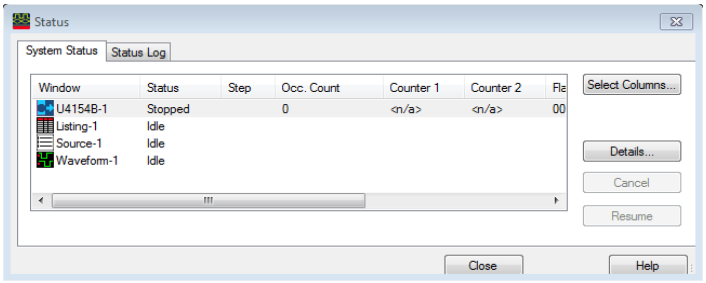
Menu	Description
Channel Assignment	Drag the slider bar to split channels between the logic analyzer modules.
Advanced Pod Assignment Mode	Select this option for finer control over which *pods* (see page 540) or *pod pairs* (see page 540) are assigned to each module or are reserved for time tag storage.  Master Pods indicate that the pod or pod pair is on the master card.

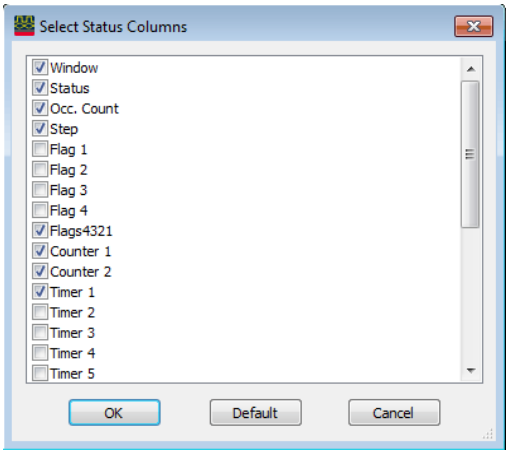
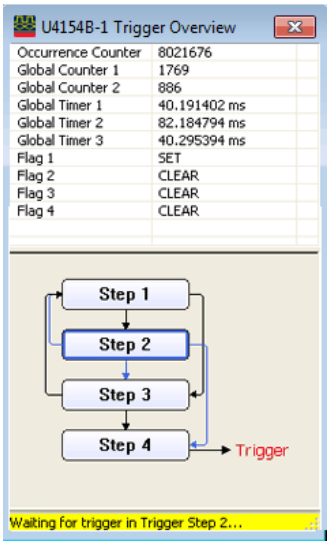
See Also
- Configuring Logic Analyzer Modules (see page 88)
- Memory Depth and Channel Count Trade-offs (see page 335)
- Pod and Channel Naming Conventions (see page 333)

Status Dialog

System Status Tab

Displays the logic analysis system status.

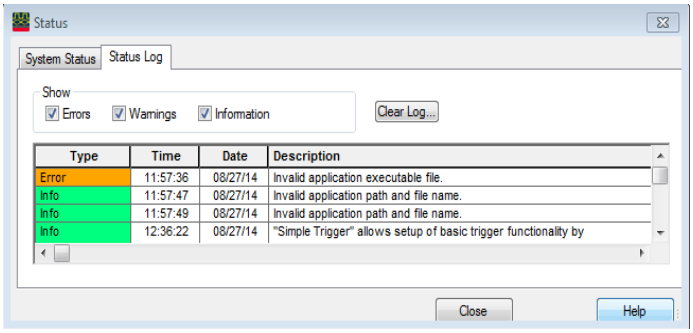

Status
System Status
Status Log
Window
Status
Step
Occ. Count
Counter 1
Counter 2
Fla
U4154B-1
Stopped
0
<n/a>
<n/a>
00
Listing-1
Idle
Source-1
Idle
Waveform-1
Idle
Select Columns...
Details...
Cancel
Resume
Close
Help

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.



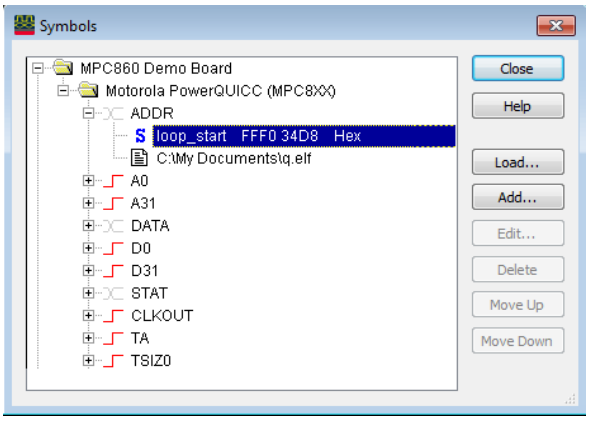
You can select the types of log messages to display, and you can clear the log.

See Also

- Options Dialog (see page 412)

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.

Menu	Description
Edit...	Edits the selected user-defined symbol.
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 118)
- Displaying Names (Symbols) for Bus/Signal Values (see page 220)

System Summary Dialog

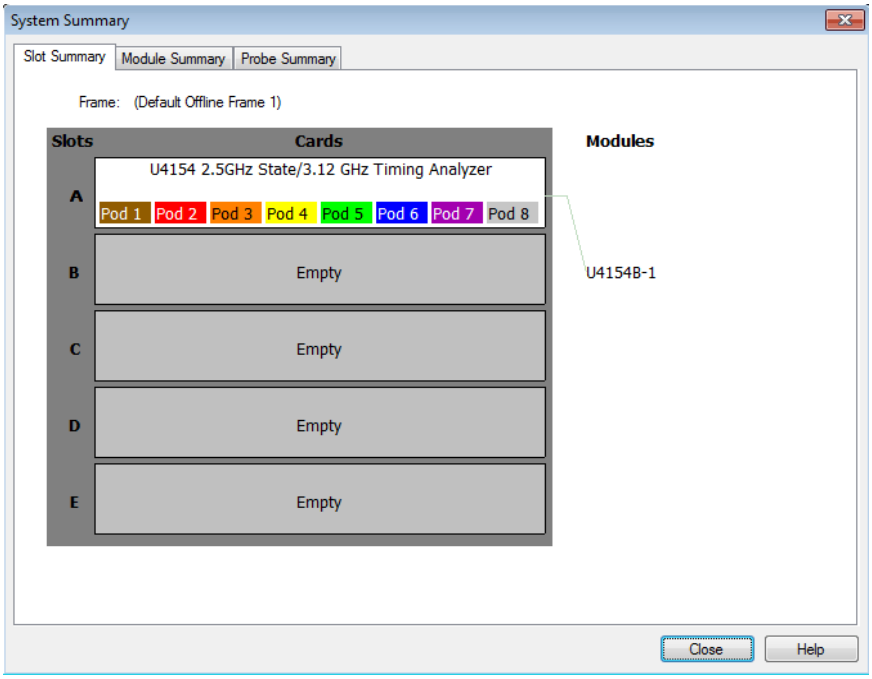
The System Summary dialog is used to see summary information about all modules in the system including a physical view of the cards in the system (Slot Summary (see page 434)), a list of all of the modules and their trigger times (Module Summary (see page 436)), and a list of how to connect all of the probes (Probe Summary (see page 436)).

- Slot Summary Tab (see page 434)
- Module Summary Tab (see page 436)
- Probe Summary Tab (see page 436)

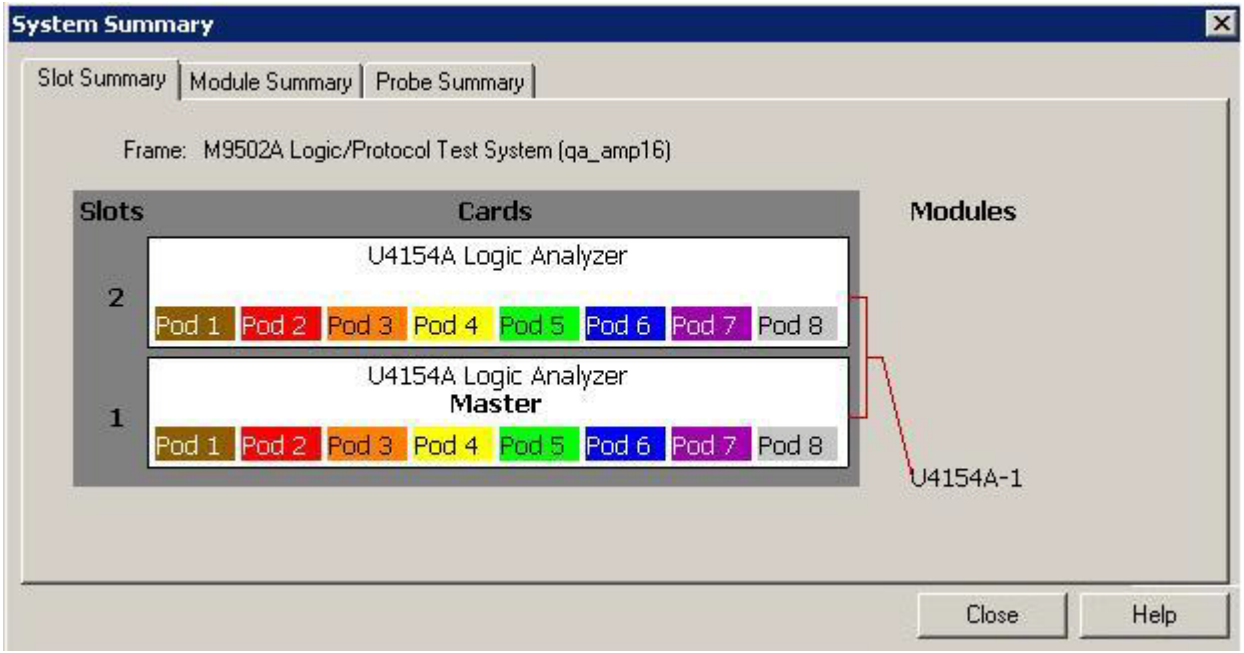
See Also
- Analyzer Setup Dialog (see page 394)

Slot Summary Tab

For the selected *frame* (see page 539) (if there are multiple frames), the Slot Summary tab shows: the *slots* (see page 542), *cards* (see page 538), and *modules* (see page 540) in the logic analysis system frame, or the *cards* (see page 538) and *modules* (see page 540) in the standalone logic analyzer.



U4154A/B Logic Analyzer Slot Summary

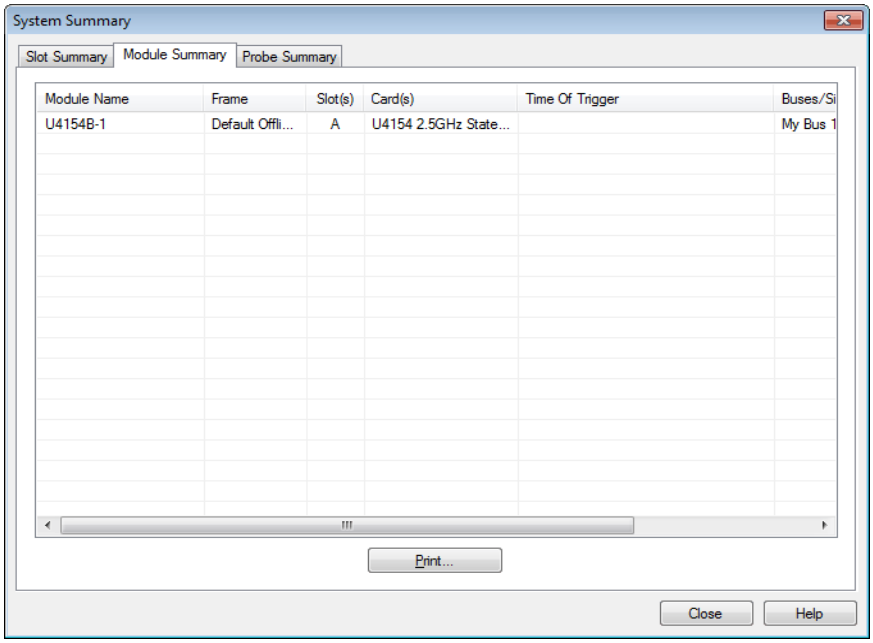


If you are using U4154A/B Logic Analyzer, the Slot Summary tab displays a graphical view of the slots of the AXIe chassis in which you installed the U4154A/B Logic Analyzer. If you are using a two-slot chassis, then the placement of the U4154A/B cards in these two slots is displayed. If you are using 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
- System Summary Dialog (see page 434)

Module Summary Tab

For each *module* (see page 540) in the logic analysis system, the Module Summary tab shows the *frame* (see page 539) in which the module resides, the *cards* (see page 538) that make up the module, the *slots* (see page 542) in which these cards reside, and the time that a trigger occurred.



See Also
- System Summary Dialog (see page 434)

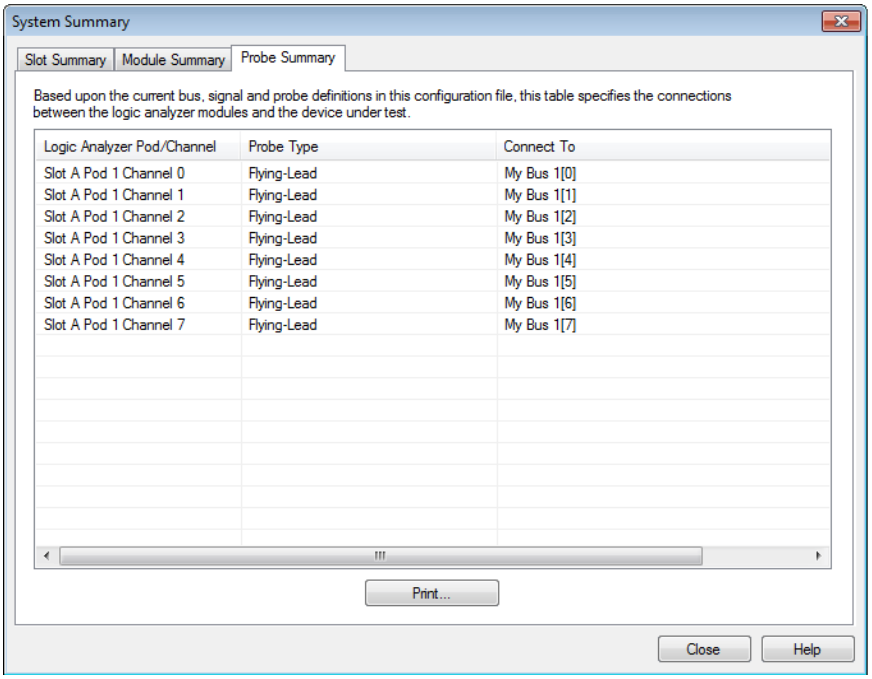
Probe Summary Tab

The Probe Summary tab shows logic analyzer pods and channels, probe types, and the connectors or signals in the device under test to which the pods and channels should be connected.

Probe types can be defined by themselves, in XML configuration files, or while importing netlists (see page 100) (to assign bus/signal names to logic analyzer channels). If you haven't defined probe types, the flying-lead probe type is assumed.

NOTE

When connecting differential probe channel pin/pad/lead pairs to single-ended signals, make sure the negative pin/pad/lead is connected to ground and the positive pin/pad/lead is connected to the single-ended signal.

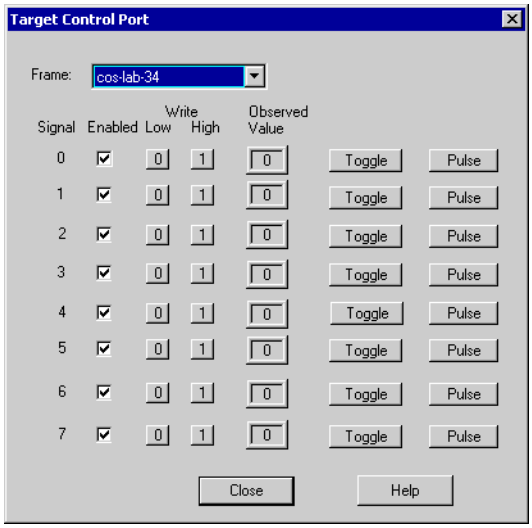


See Also
- "To define probes" (in the online help)
- System Summary Dialog (see page 434)
- "Connecting Probes Using the Probe Summary" (in the online help)

Target Control Port Dialog

The 16850-series logic analysis system *frames* (see page 539) 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.

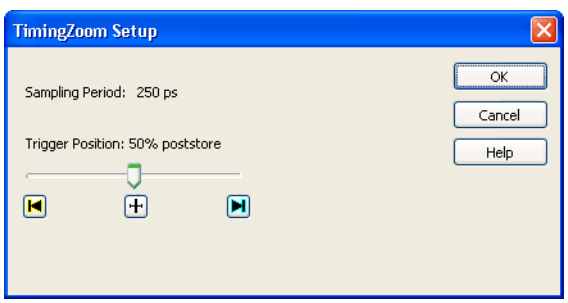


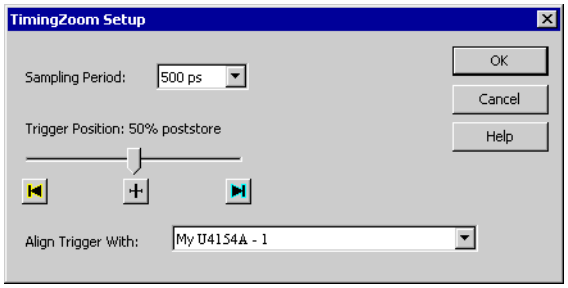
Menu	Description
Signal	Identifies the target control port signal.
Enabled	Lets you enable or disable an output signal on the target control port. When disabled, the signal has a tri-state high-impedance output. When enabled, the signal outputs a standard 3.3V logic level with 1 high and 0 low.
Write	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 74)

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 492)), 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.
Align Trigger With	In some older logic analyzer modules, if the logic analyzer is split, this lets you specify which analyzer's trigger timing zoom should be aligned with.

See Also

- Using Timing Zoom (see page 116)
- Logic Analyzer Notes, Timing Zoom (see page 492)

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 440)
- State Mode Trigger Functions (see page 452)

See Also

- To show a trigger sequence step as Advanced If/Then trigger functions (see page 160)
- To convert a trigger sequence step to Advanced If/Then trigger functions (see page 160)
- Triggering From, and Sending Triggers To, Other Modules/Instruments (see page 164)

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 66).

Edge

- Edge (see page 441)
- "N" number of edges (see page 441)
- Edge and Pattern (see page 442)
- Edge followed by edge (see page 442)
- Edges too far apart (see page 443)
- Edge followed by pattern (see page 443)
- Pattern too late after edge (see page 444)

Bus Pattern

- Pattern (see page 444)
- Edge and Pattern (see page 442)
- Pattern present for > "T" time (see page 445)
- Pattern present for < "T" time (see page 445)
- Pattern absent for > "T" time (see page 446)
- Pattern absent for < "T" time (see page 446)
- Edge followed by pattern (see page 443)
- Pattern too late after edge (see page 444)

Other

- Find anything "N" times (see page 446)
- Width violation on pattern or pulse (see page 447)
- Wait "T" seconds (see page 447)
- Run until user stop (see page 448)
- Wait for external arm (see page 448)
- Wait for arm from another module (see page 449)

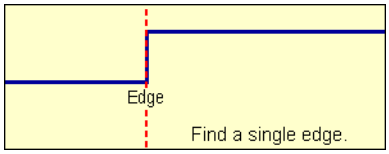
Advanced

- Advanced If/Then (see page 449)
- Advanced 2-Way Branch (see page 450)
- Advanced 3-Way Branch (see page 450)
- Advanced 4-Way Branch (see page 451)
- Pattern "AND" Pattern (see page 451)
- Pattern "OR" Pattern (see page 452)

See Also
- To replace or insert trigger functions into trigger sequence steps (see page 142)
- State Mode Trigger Functions (see page 452)
- To store a trigger (see page 172)
- To recall a trigger (see page 172)

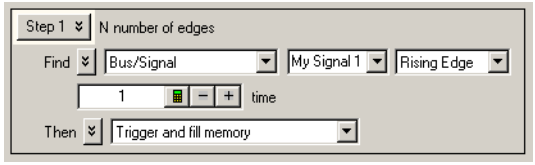
Edge

This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a user-defined edge occurs.

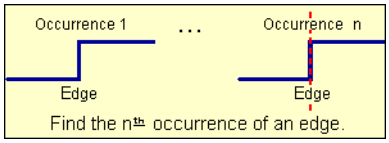


To edit this function
- Specifying Advanced Triggers (see page 138)

"N" number of edges

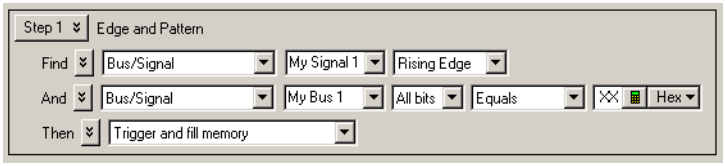


This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when the "Nth" occurrence of a user-defined edge occurs.

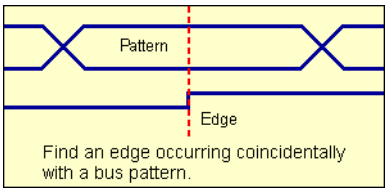


To edit this function
- Specifying Advanced Triggers (see page 138)

Edge and Pattern



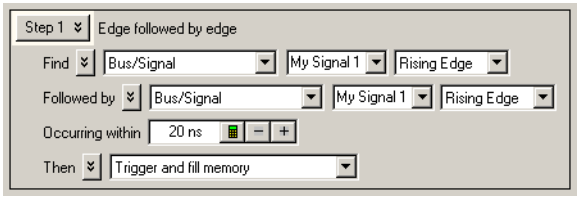
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when both a user-defined edge and bus pattern occur at the same time.



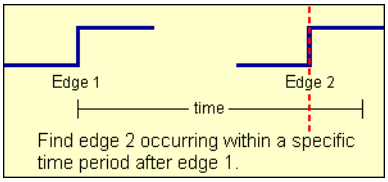
To edit this function

- Specifying Advanced Triggers (see page 138)

Edge followed by edge



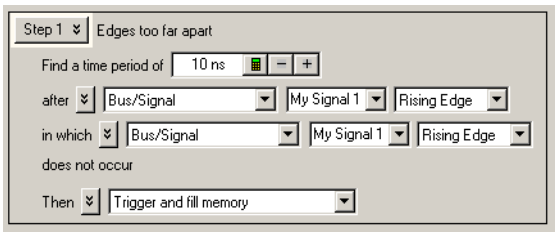
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when edge 2 occurs within a specified time period after edge 1.



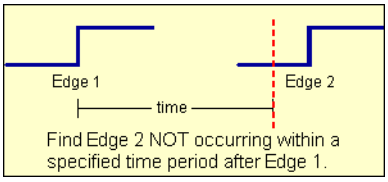
To edit this function

- Specifying Advanced Triggers (see page 138)

Edges too far apart



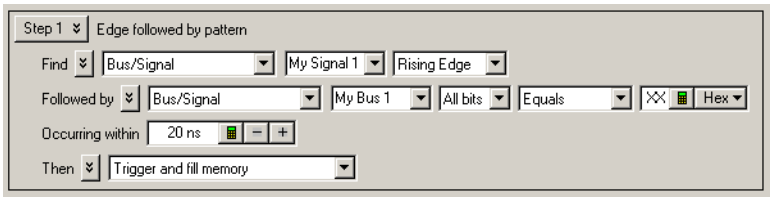
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when edge 2 does not occur within a specified time period after edge 1.



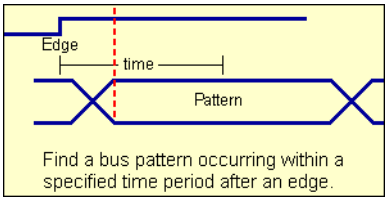
To edit this function

- Specifying Advanced Triggers (see page 138)

Edge followed by pattern



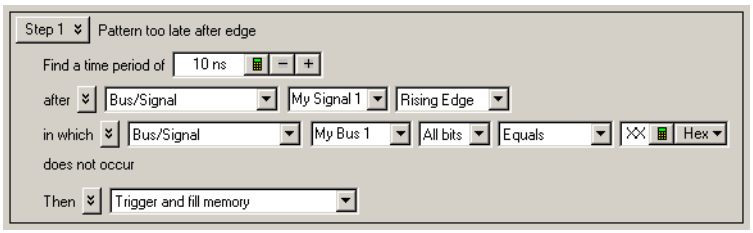
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a bus pattern occurs within a specified time period after an edge.



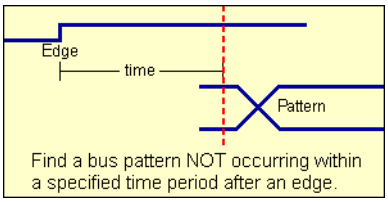
To edit this function

- Specifying Advanced Triggers (see page 138)

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.

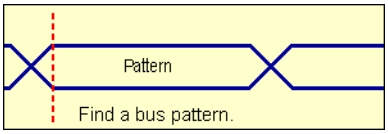


To edit this function

- Specifying Advanced Triggers (see page 138)

Pattern

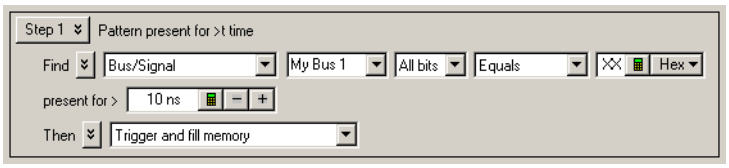
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a designated bus pattern occurs.



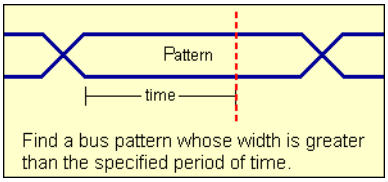
To edit this function

- Specifying Advanced Triggers (see page 138)

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.

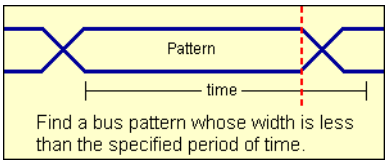


To edit this function

- Specifying Advanced Triggers (see page 138)

Pattern present for < "T" time

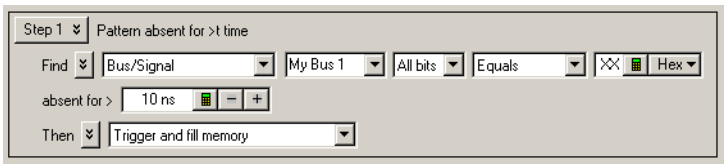
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a user-defined bus pattern is present less than a specified time period.



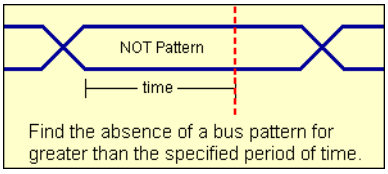
To edit this function

- Specifying Advanced Triggers (see page 138)

Pattern absent for > "T" time



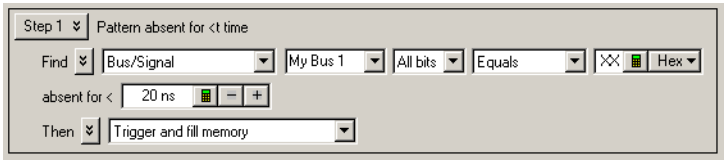
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when a user-defined bus pattern is absent greater than a specified time period.



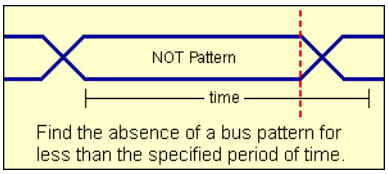
To edit this function

- Specifying Advanced Triggers (see page 138)

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.

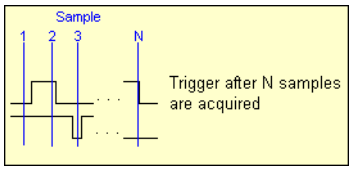


To edit this function

- Specifying Advanced Triggers (see page 138)

Find anything "N" times (timing)

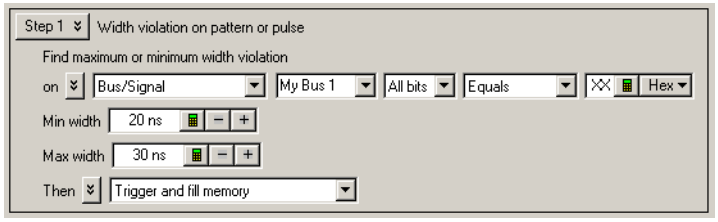
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when it sees any data (Anything) for the Nth time.



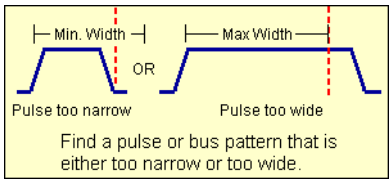
To edit this function

- Specifying Advanced Triggers (see page 138)

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.



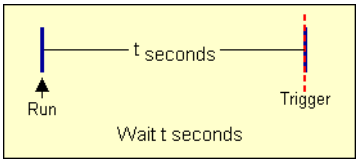
To edit this function

- Specifying Advanced Triggers (see page 138)

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 139) instead.

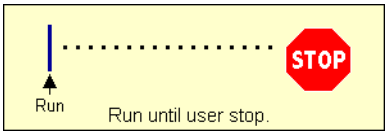


To edit this function

- Specifying Advanced Triggers (see page 138)

Run until user stop (timing)

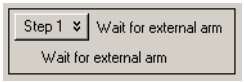
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. This trigger function sets up to never trigger. You must select the stop button to view the captured data.



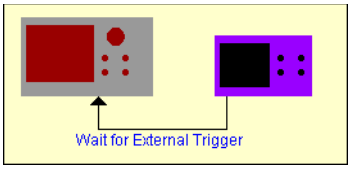
To edit this function

- Specifying Advanced Triggers (see page 138)

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 164)). The **Trigger In** BNC connector is located on the front panel of the U4154A/B Logic Analyzers.

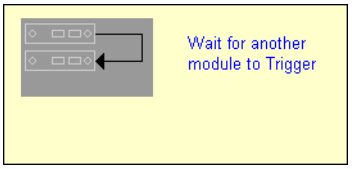


To edit this function

- Specifying Advanced Triggers (see page 138)

Wait for arm from another module (timing)

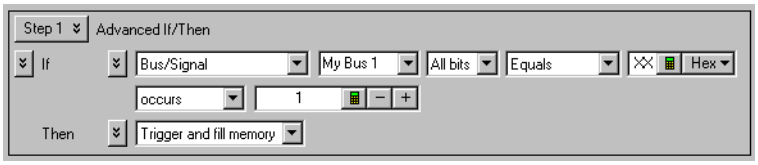
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when an arming signal from another module occurs (see Triggering From, and Sending Triggers To, Other Modules/Instruments (see page 164)).



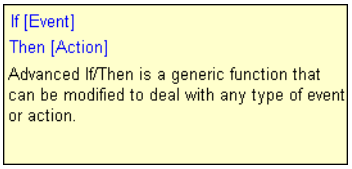
To edit this function

- Specifying Advanced Triggers (see page 138)

Advanced If/Then (timing)



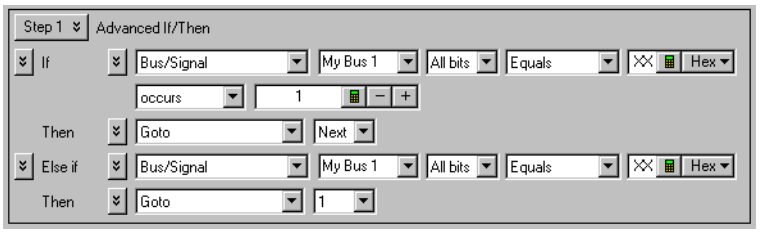
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when the "If" clause becomes true.



To edit this function

- Specifying Advanced Triggers (see page 138)

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.

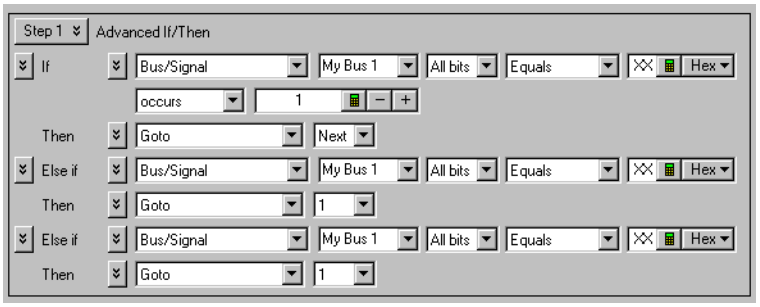
If [Event] then [Action]
Else if [Event] then [Action]

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 138)

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.

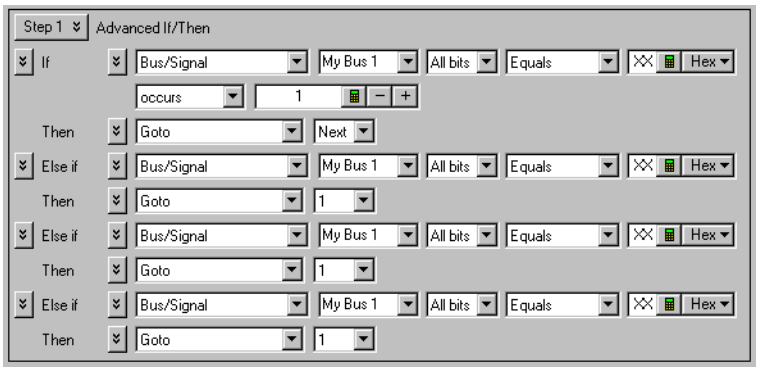
If [Event] then [Action]
Else if [Event] then [Action]
Else if [Event] then [Action]

Advanced 3-Way Branch provides the ability to check for three different events and take a different action for each event.

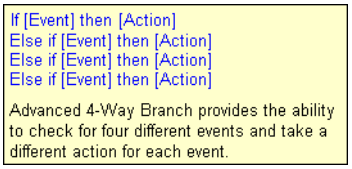
To edit this function

- Specifying Advanced Triggers (see page 138)

Advanced 4-Way Branch (timing)



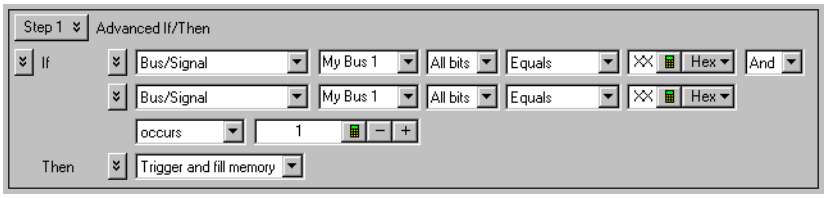
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The four-way branch is evaluated true when either of four patterns (If or Else if) are found. Depending on which pattern is found true, the appropriate "Then" action is executed.



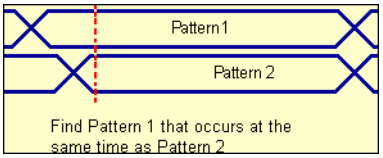
To edit this function

- Specifying Advanced Triggers (see page 138)

Pattern "AND" Pattern (timing)



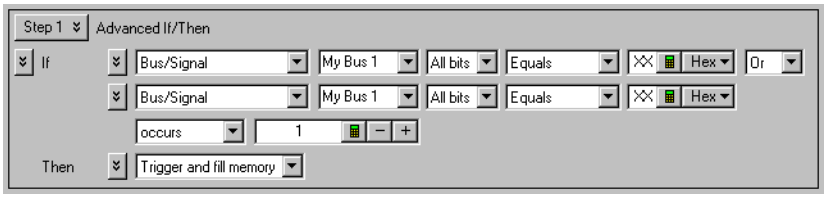
This trigger function is available when the acquisition mode is set to **Timing - Asynchronous**. The analyzer triggers when both pattern1 "AND" pattern2 occur at the same time, and for the specified numbers of samples (occurs).



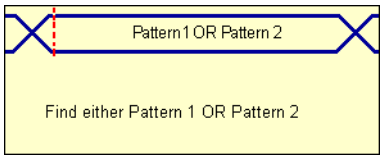
To edit this function

- Specifying Advanced Triggers (see page 138)

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 138)

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 67).

Patterns

- Pattern "N" times (see page 453)
- "N" consecutive samples with Pattern1 (see page 453)
- Pattern1 followed by Pattern2 (see page 454)
- Pattern1 immediately followed by Pattern2 (see page 454)
- Pattern1 followed by Pattern2 before Pattern3 (see page 455)
- Too few states between Pattern1 and Pattern2 (see page 456)
- Too many states between Pattern1 and Pattern2 (see page 456)
- Pattern2 occurring too soon after Pattern1 (see page 457)
- Pattern2 occurring too late after Pattern1 (see page 458)
- Find a packet (see page 458)

Other

- Reset and start timer (see page 459)
- Find anything "N" times (see page 459)
- Run until user stop (see page 460)
- Wait for external arm (see page 460)
- Wait for arm from another module (see page 461)
- Wait "N" external clock states (see page 461)

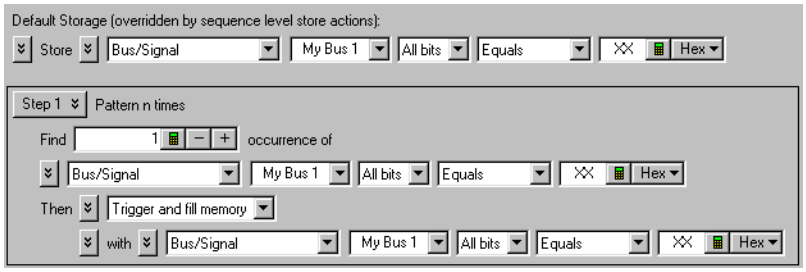
Advanced

- Advanced If/Then (see page 462)
- Advanced 2-Way Branch (see page 462)
- Advanced 3-Way Branch (see page 463)
- Advanced 4-Way Branch (see page 463)
- Pattern "AND" Pattern (see page 464)
- Pattern "OR" Pattern (see page 465)

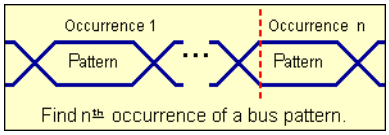
See Also

- To replace or insert trigger functions into trigger sequence steps (see page 142)
- Timing Mode Trigger Functions (see page 440)
- To specify default storage (see page 151)
- To store a trigger (see page 172)
- To recall a trigger (see page 172)

Pattern "N" times



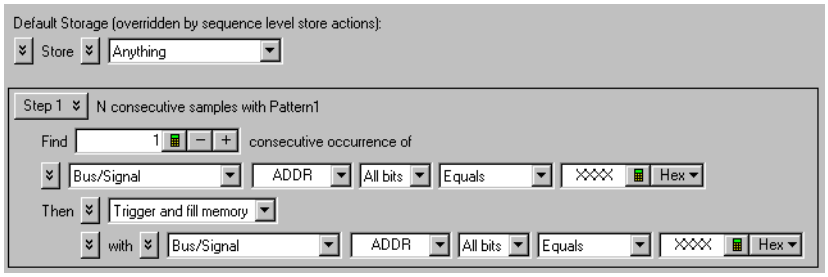
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when it finds the nth occurrence of a bus pattern as shown below.



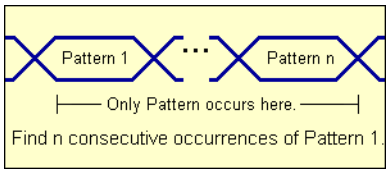
To edit this function

- Specifying Advanced Triggers (see page 138)

"N" consecutive samples with Pattern1



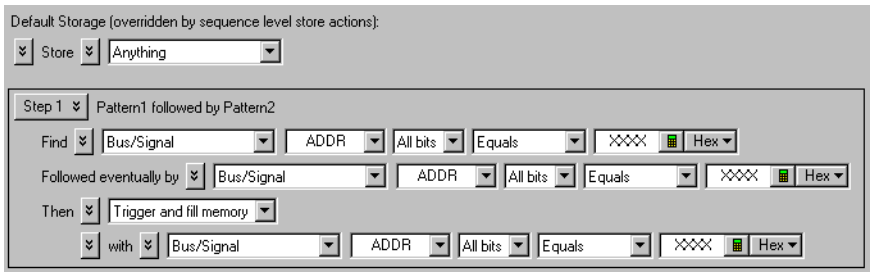
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when a bus pattern occurs a specified number times.



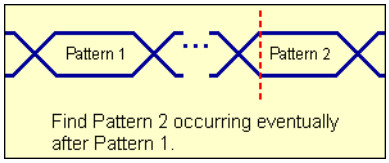
To edit this function

- Specifying Advanced Triggers (see page 138)

Pattern1 followed by Pattern2



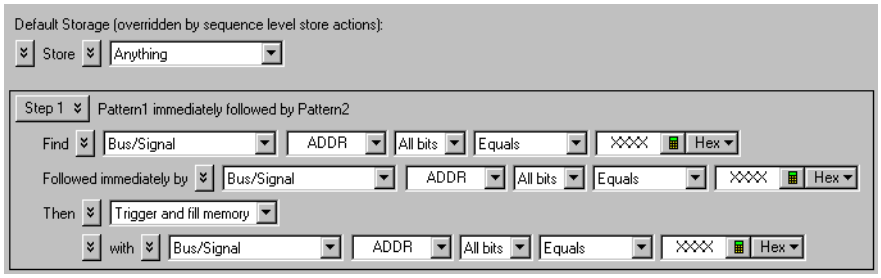
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern2 occurs eventually after pattern 1.



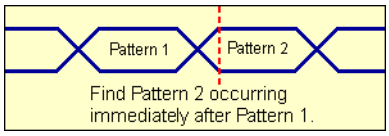
To edit this function

- Specifying Advanced Triggers (see page 138)

Pattern1 immediately followed by Pattern2



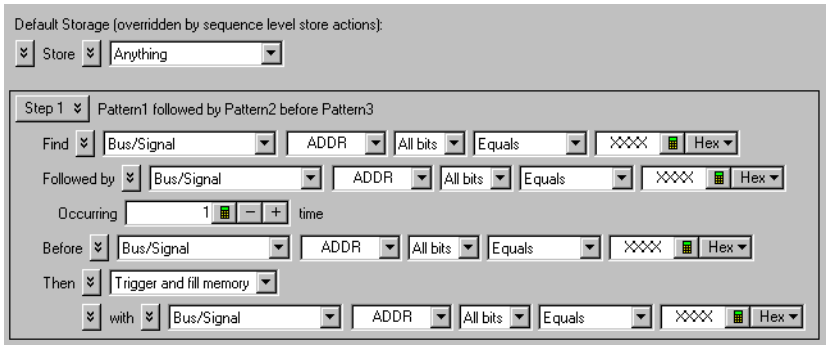
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern 2 is found immediately after exiting pattern 1.



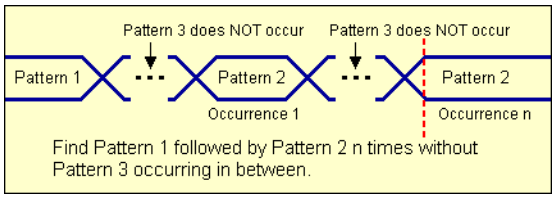
To edit this function

- Specifying Advanced Triggers (see page 138)

Pattern1 followed by Pattern2 before Pattern3



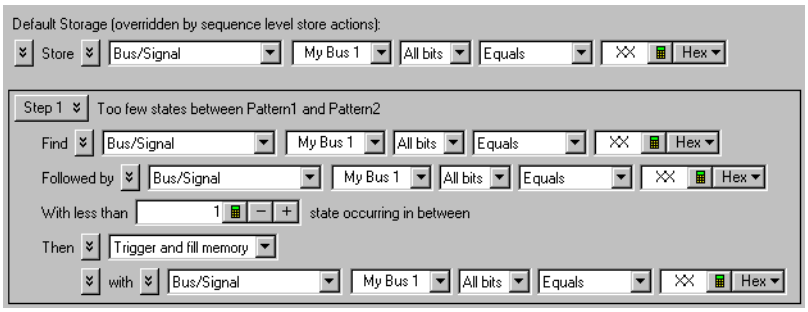
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern2 occurs eventually after pattern1, for a specified number of times, without pattern3 occurring in between.



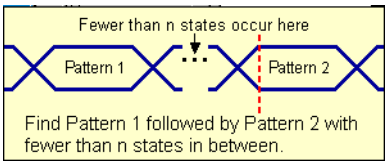
To edit this function

- Specifying Advanced Triggers (see page 138)

Too few states between Pattern1 and Pattern2



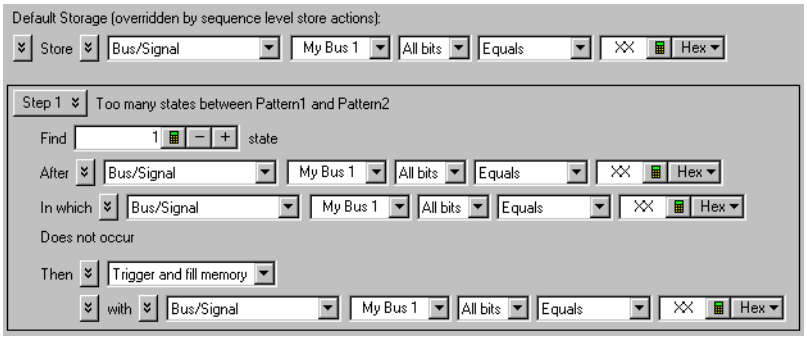
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern1 is followed by pattern2 with fewer than "N" specified states in between.



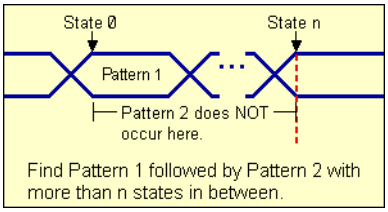
To edit this function

- Specifying Advanced Triggers (see page 138)

Too many states between Pattern1 and Pattern2



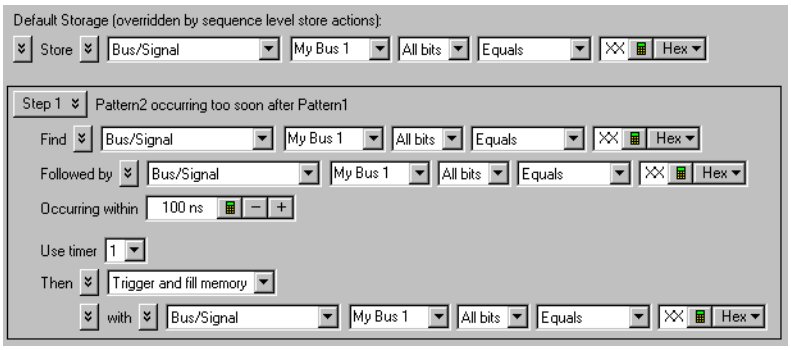
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern1 is followed by pattern2 with more than "N" specified states in between.



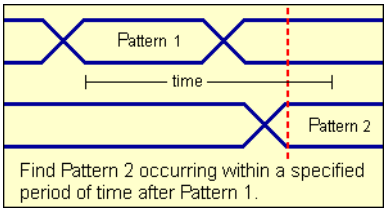
To edit this function

- Specifying Advanced Triggers (see page 138)

Pattern2 occurring too soon after Pattern1



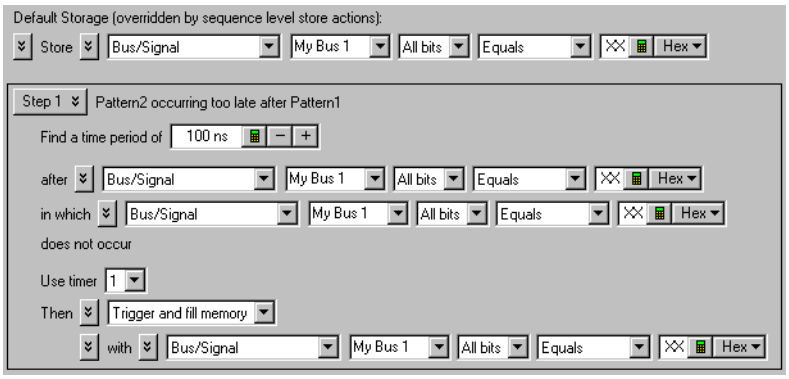
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern2 occurs within a specified time period after pattern1.



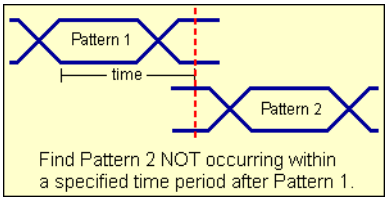
To edit this function

- Specifying Advanced Triggers (see page 138)

Pattern2 occurring too late after Pattern1



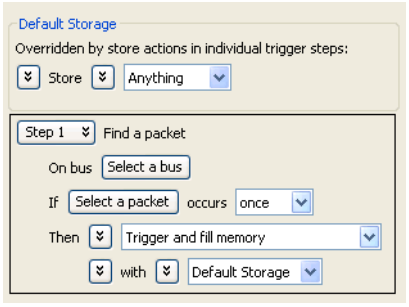
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when pattern2 does not occur within a specified time period after pattern1.



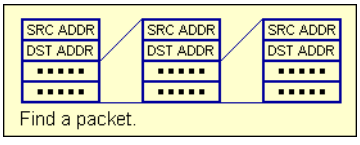
To edit this function

- Specifying Advanced Triggers (see page 138)

Find a packet



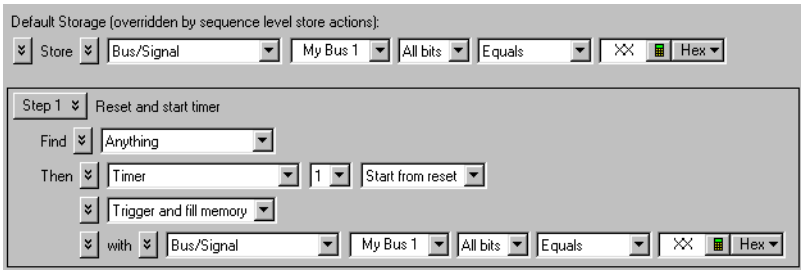
This trigger function is available when the acquisition mode is set to **State - Synchronous**. It will trigger the logic analyzer when the selected packet and occurrence is found.



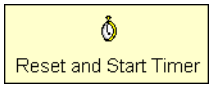
To edit this function

- Specifying Advanced Triggers (see page 138)

Reset and start timer (state)



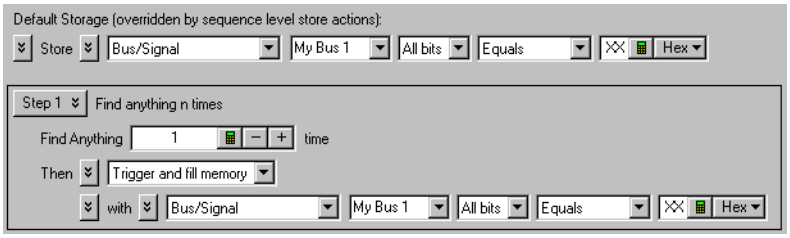
This trigger function is available when the acquisition mode is set to **State - Synchronous**. This trigger function resets a timer, then starts the timer for a specified period of time. This trigger function requires that the timer value be set in either the same trigger step, or another trigger step that follows. When the timer stops, the analyzer triggers. For more information refer to "To configure a timer (see page 139)".



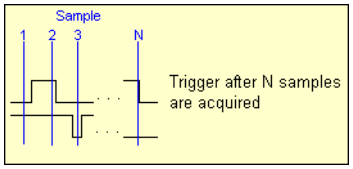
To edit this function

- Specifying Advanced Triggers (see page 138)

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.

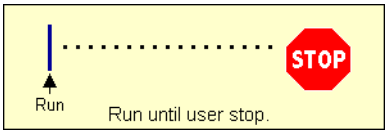


To edit this function

- Specifying Advanced Triggers (see page 138)

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.

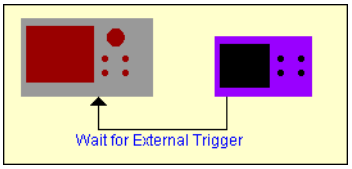


To edit this function

- Specifying Advanced Triggers (see page 138)

Wait for external arm (state)

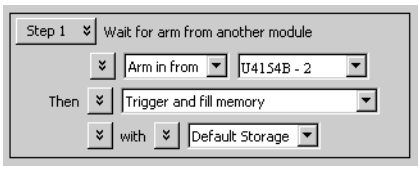
This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers when an external arming signal appears through the external **Trigger In** port (see Triggering From, and Sending Triggers To, Other Modules/Instruments (see page 164)). The **Trigger In** BNC connector is located on the front panel of the U4154A/B Logic Analyzers.



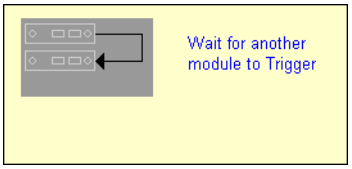
To edit this function

- Specifying Advanced Triggers (see page 138)

Wait for arm from another module (state)



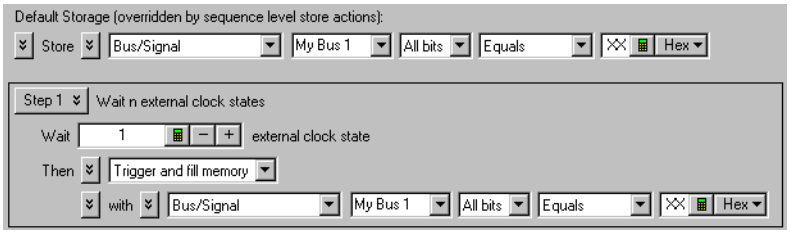
This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers when an arming signal from another module occurs (see Triggering From, and Sending Triggers To, Other Modules/Instruments (see page 164)).



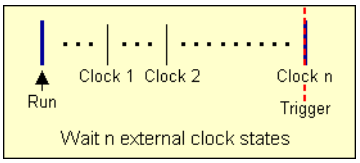
To edit this function

- Specifying Advanced Triggers (see page 138)

Wait "N" external clock states



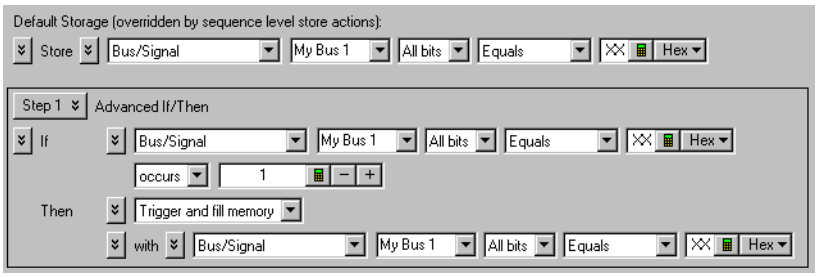
This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers on the "Nth" occurrence of the external clock signal (plus any user-defined clock qualification) from the device under test.



To edit this function

- Specifying Advanced Triggers (see page 138)

Advanced If/Then (state)



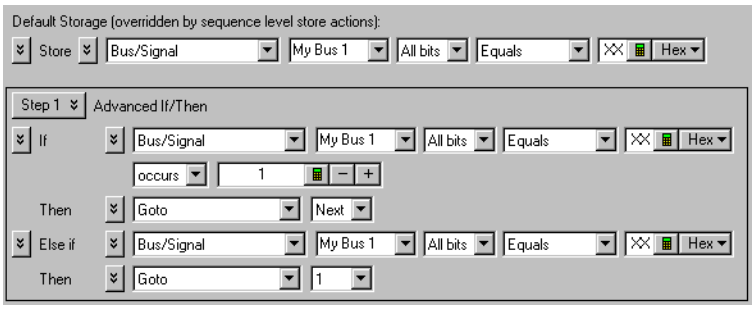
This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers when the "If" clause becomes true.

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 138)

Advanced 2-Way Branch (state)



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.

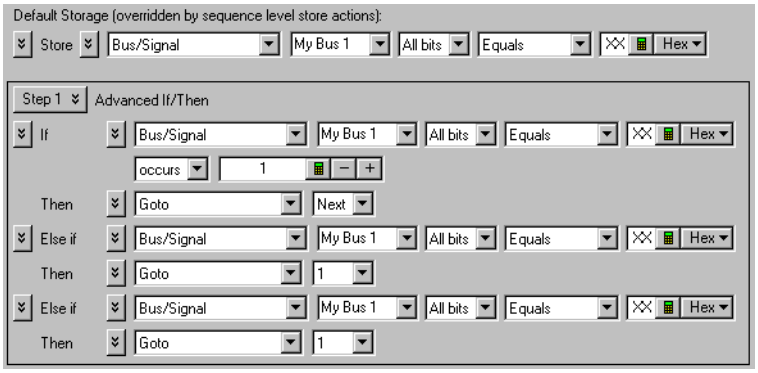
If [Event] then [Action]
Else if [Event] then [Action]

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 138)

Advanced 3-Way Branch (state)



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.

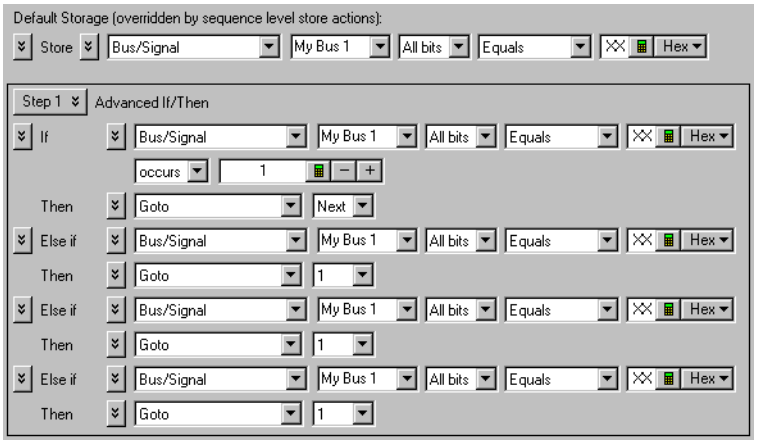
If [Event] then [Action]
Else if [Event] then [Action]
Else if [Event] then [Action]

Advanced 3-Way Branch provides the ability to check for three different events and take a different action for each event.

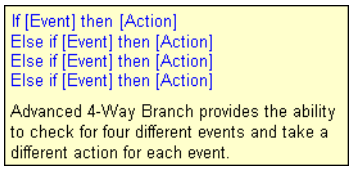
To edit this function

- Specifying Advanced Triggers (see page 138)

Advanced 4-Way Branch (state)



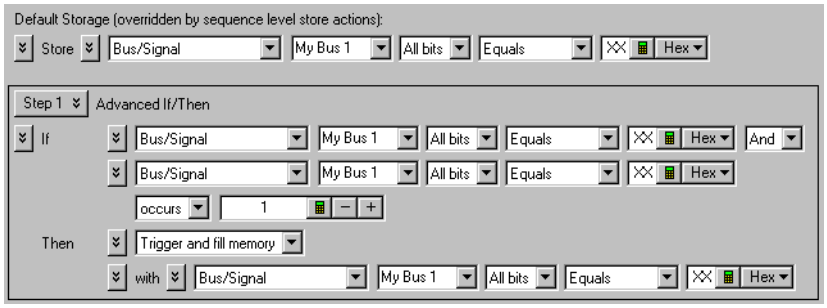
This trigger function is available when the acquisition mode is set to **State - Synchronous**. The four-way branch is evaluated true when either of four patterns (If or Else if) are found. Depending on which pattern is found true, the appropriate "Then" action is executed.



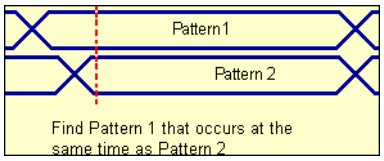
To edit this function

- Specifying Advanced Triggers (see page 138)

Pattern "AND" Pattern (state)



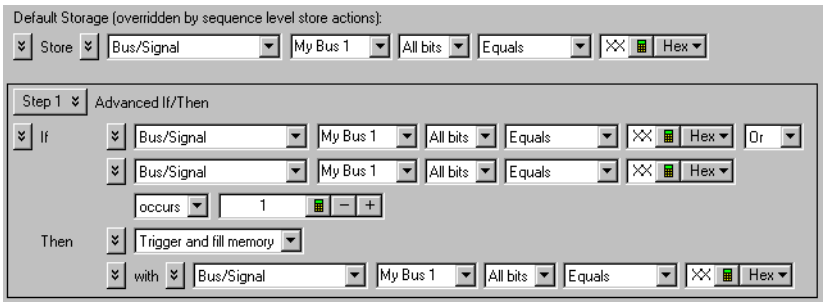
This trigger function is available when the acquisition mode is set to **State - Synchronous**. The analyzer triggers when both pattern1 "AND" pattern2 occur at the same time, and for the specified numbers of samples (occurs).



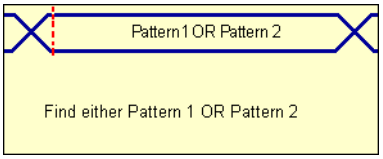
To edit this function

- Specifying Advanced Triggers (see page 138)

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 138)

Data Formats

Configuration File Formats

- ALA Format (see page 466)
- "XML Format" (in the online help)

Data Export Formats

- Standard CSV Format (see page 466)
- Module CSV Format (see page 466)
- Module Binary (ALB) Format (see page 473)

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 176) and opened (see page 188) through the File menu (see page 358).

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 467) 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=0.000000000000000e+000
TRIGGER_ROW=524288
NUM_ROWS=1048576
TIME_SOURCE PERIOD=2.000000000000000e-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 467)
- Module CSV Format File Characteristics (see page 472)

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 = rowNumber.

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 467)

Module Binary (ALB) Format

You can export captured data to module binary files.

Module binary format files contain a header section (see page 467) 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 467)

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 121) file.

See Also

- To load symbols from a file (see page 119)
- To change symbol reader options (see page 121)
- To create an ASCII symbol file (see page 121)

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 476).

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 477)
- FUNCTIONS (see page 478)
- USER (see page 478)
- VARIABLES (see page 480)
- SOURCE LINES (see page 480)
- START ADDRESS (see page 481)
- Comments (see page 481)

General-Purpose ASCII (GPA) Record Format Summary

Format

```
[SECTIONS (see page 477)]
section_name  start..end  attribute

[FUNCTIONS (see page 478)]
func_name  start..end

[USER (see page 478)]
sym_name   value                    base
sym_name   start_value   [size]     base
sym_name   start_value..end_value   base

[VARIABLES (see page 480)]
var_name   start [size]
var_name   start..end
```

```
[SOURCE LINES (see page 480)]
File: file_name
line#  address

[START ADDRESS (see page 481)]
address

#comment text (see page 481)
```

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 s
ize.

[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
section_name	A symbol representing the name of the section.
start	The first address of the section, in hexadecimal.
end	The last address of the section, in hexadecimal.
attribute	(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 reloc_offset - Lets you specify an offset value (up to 32-bit) for a relocated section. SetReloc reloc_value - 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
func_name	A symbol representing the function name.
start	The first address of the function, in hexadecimal.
end	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: · 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 siz
e.
hdontcare    00xF           hex
hvalue       00EF           hex
hrange       0000..00FF     hex
hrange3      0000    99     hex      # The 99 is a decimal value for siz
e.
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 bas
es 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 253)

START ADDRESS

Format
```
[START ADDRESS]
address
```

Menu	Description
address	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

- U4154A Logic Analyzer Product Overview (see page 482)
- U4154B Logic Analyzer Product Overview (see page 484)
- 16850-Series Logic Analyzer Product Overview (see page 485)
- *Keysight Logic and Protocol Analyzer* Application Product Overview (see page 488)

See Also

- Tutorial - Getting to know your logic analyzer (see page 56)

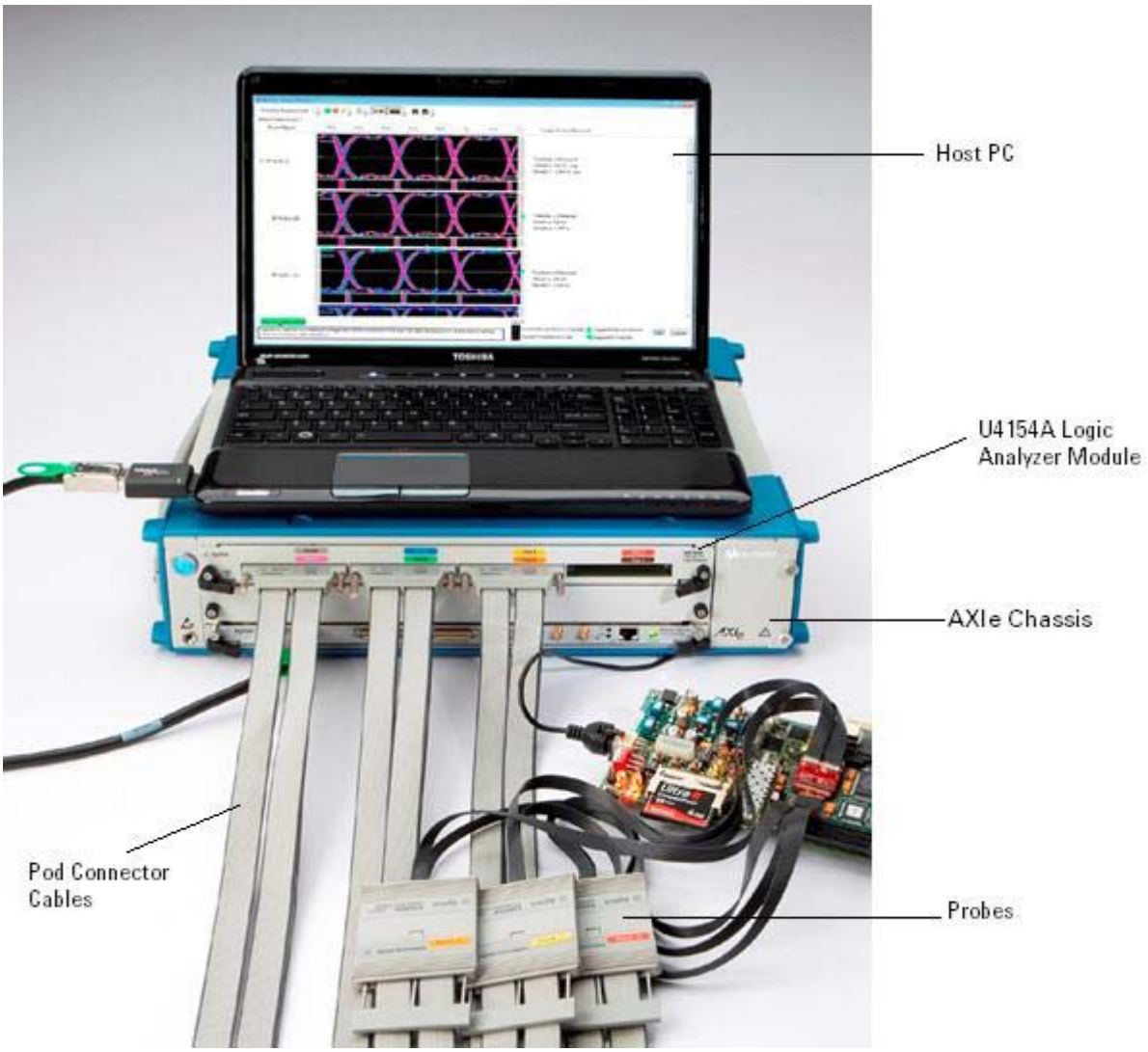
U4154A Logic Analyzer Product Overview

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 56)
- U4154A Logic Analyzer Specifications and Characteristics (see page 528)

U4154B Logic Analyzer Product Overview

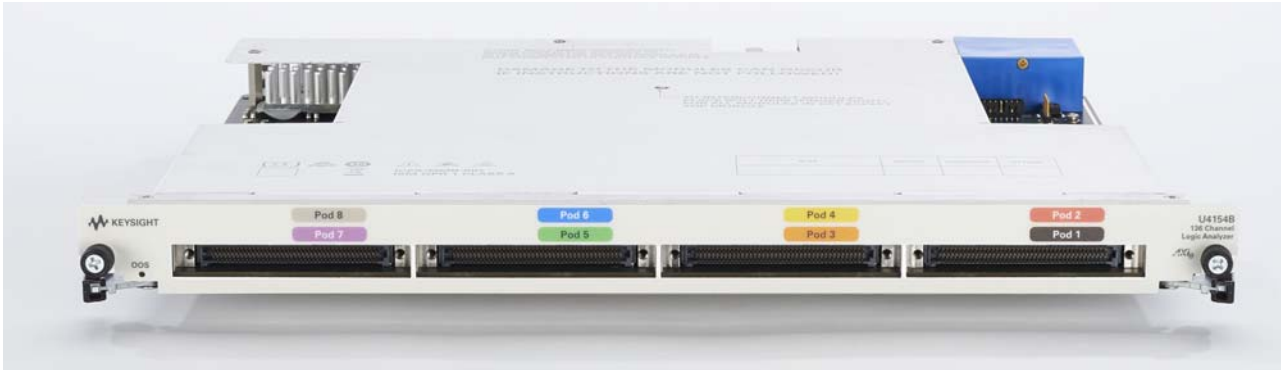
The Keysight U4154B logic analyzer is a 136 channel AXIe based high speed state and timing logic analyzer. It provides reliable data capture with powerful analysis and validation tools for high data rate memory based products. It helps you to perform functional validation and characterization of your 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 AXIe 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 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 U4154B 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 U4154B 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 56)
- U4154B Logic Analyzer Specifications and Characteristics (see page 531)

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.
- 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, 1,024 x 768 (XGA) resolution. Touch screen with Option 103. See Tips for Using the Touch Screen (see page 486).
- Front panel knob and buttons. See 16850 Series Front Panel Operation (see page 487).
- 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 56)
- 16850-Series Logic Analyzer Specifications and Characteristics (see page 531)

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.

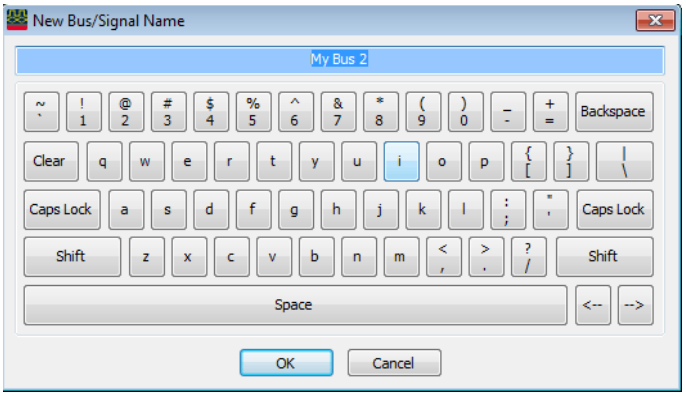
Use the front panel marker knobs to place markers. Placing markers is hard to do accurately using the touch screen. (Any marker can be selected using the Marker Choose button.)

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.

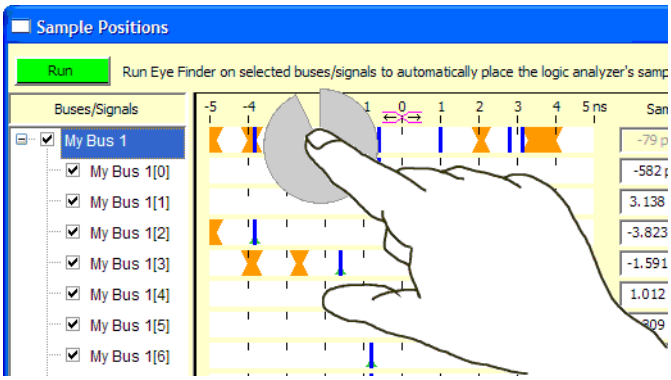
(You can also choose **Start>All Programs>Accessories>Accessibility>On-Screen Keyboard**.)

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 174). 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 174). 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 174) 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 486)

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 489)

Keyboard Commands

- Access Menus (see page 489)
- File Operations (see page 489)
- Edit Operations (see page 490)
- Search Operations (see page 490)
- View operations (see page 490)
- Run/Stop Operations (see page 490)
- Compare Operations (see page 491)
- Listing Operations (see page 491)
- Waveform Operations (see page 491)
- Window Operations (see page 491)
- Help Operations (see page 491)
- Miscellaneous (see page 491)

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

Menu	Description
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 492)
- Timing Mode Sampling Options (see page 492)
- State Mode Sampling Options (see page 492)
- Timing Zoom (see page 492)

Channels and Memory Depth

- U4154A Logic Analyzer Notes, Channels and Memory Depth (see page 493)
- U4154B Logic Analyzer Notes, Channels and Memory Depth (see page 495)
- 16850-Series Logic Analyzer Notes, Channels and Memory Depth (see page 526)

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:

- U4154A Logic Analyzer Notes, Timing Mode Sampling Options/Period (see page 493)
- U4154B Logic Analyzer Notes, Timing Mode Sampling Options/Period (see page 493)
- 16850-Series Logic Analyzer Notes, Timing Mode Sampling Options/Period (see page 526)

State Mode Sampling Options

The state mode sampling options let you choose between *General State Mode* (default) or *Turbo State Mode* (faster sampling). For notes on these modes in a particular logic analyzer, see:

- U4154A Logic Analyzer Notes, State Mode Sampling Options (see page 494)
- U4154B Logic Analyzer Notes, State Mode Sampling Options (see page 494)
- 16850-Series Logic Analyzer Notes, State Mode Sampling Options (see page 526)

Timing Zoom

- U4154A Logic Analyzer Notes, Timing Zoom (see page 494)
- U4154B Logic Analyzer Notes, Timing Zoom (see page 494)
- 16850-Series Logic Analyzer Notes, Timing Zoom

U4154A Logic Analyzer Notes

- Differences from Other Logic Analyzers (see page 492)
- Channels and Memory Depth (see page 493)
- Timing Mode Sampling Options/Period (see page 493)
- State Mode Sampling (see page 494)
- Timing Zoom (see page 494)

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 502.

- 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 U4154A/B Logic Analyzers" on page 498.
- 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 U4154A/B Logic Analyzer" on page 508.
- 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. For more information, refer to "Specifying Advanced Triggers in the U4154A/B Logic Analyzer" on page 506.
- 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

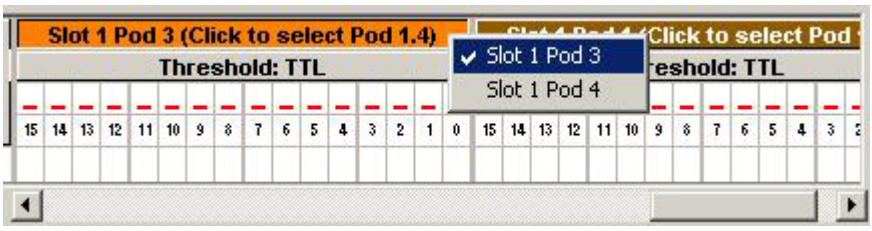
U4154A Memory depth	Option 002	Option 004	Option 008	Option 016	Option 032	Option 064	Option 128	Option 200
	2 M	4 M	8 M	16 M	32 M	64 M	128 M	200 M
Channels	136 channels per U4154A module (This includes 128 data channels and 8 clock channels) In a multi-card set: 136 channels * number of U4154A modules 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 337. 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 U4154A/B Logic Analyzers
- "Specifying Advanced Triggers in the U4154A/B Logic Analyzer
- "Setting up and Running Eyescans in U4154A/B Logic Analyzer
- "U4154A Logic Analyzer Specifications and Characteristics
- "Pod and Channel Naming Conventions in U4154A/B Logic Analyzer

U4154B Logic Analyzer Notes

- Differences from Other Logic Analyzers (see page 495)
- Channels and Memory Depth (see page 495)

- Timing Mode Sampling Options/Period (see page 495)
- State Mode Sampling (see page 496)
- Timing Zoom (see page 496)

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 504.
- 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.						

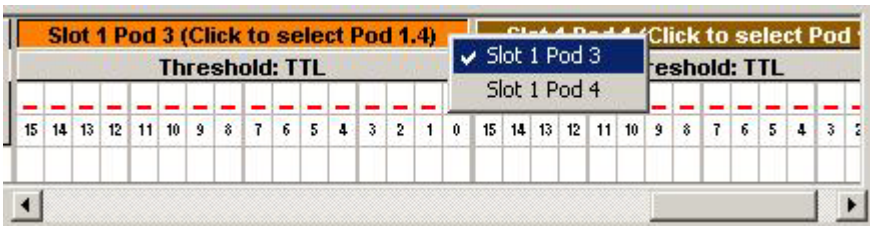
Timing Mode Sampling Options/Period

- **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 337. 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.

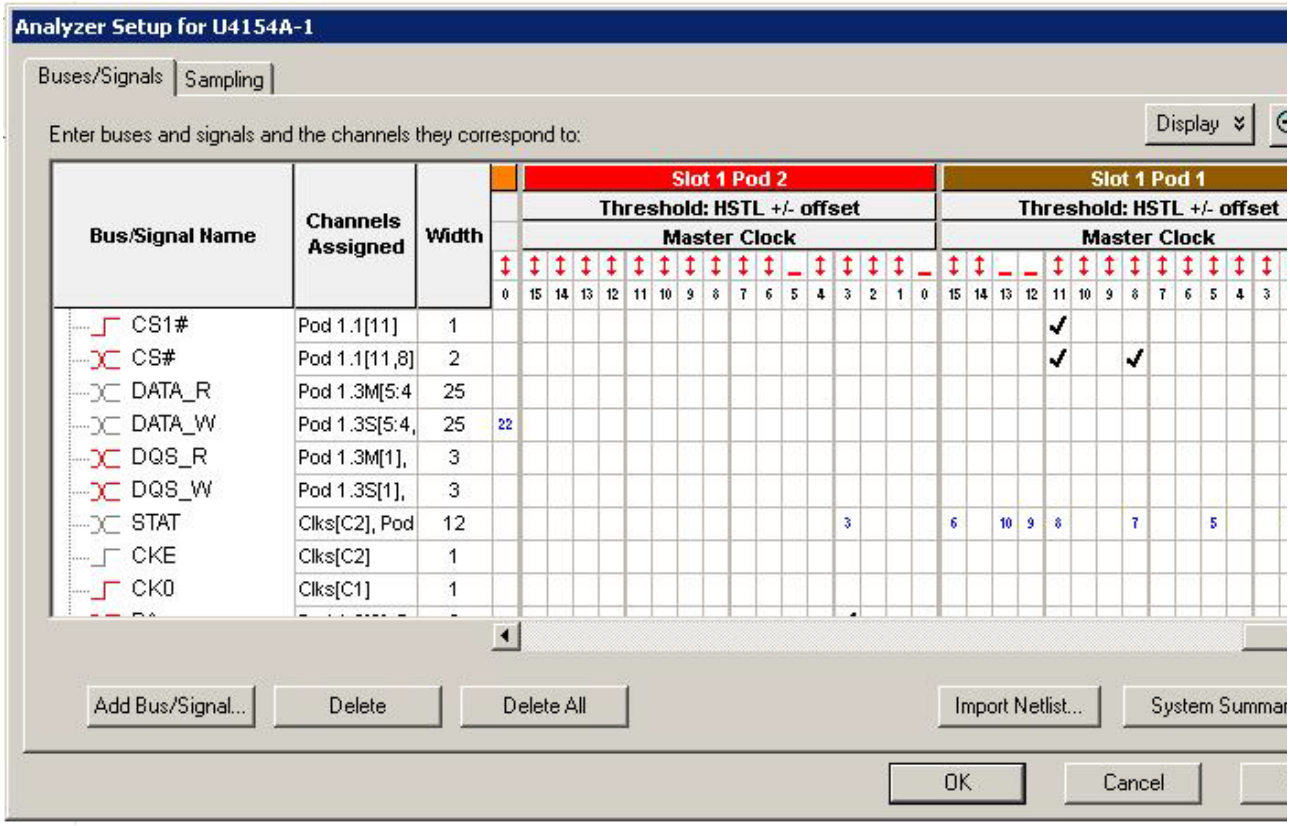
See Also

- "Setting up the State Sampling Options in U4154B Logic Analyzer
- "Changing Advanced Probe Settings for U4154A/B Logic Analyzers
- "Specifying Advanced Triggers in the U4154A/B Logic Analyzer
- "Setting up and Running Eyescans in U4154A/B Logic Analyzer
- "U4154B Logic Analyzer Specifications and Characteristics
- "Pod and Channel Naming Conventions in U4154A/B Logic Analyzer

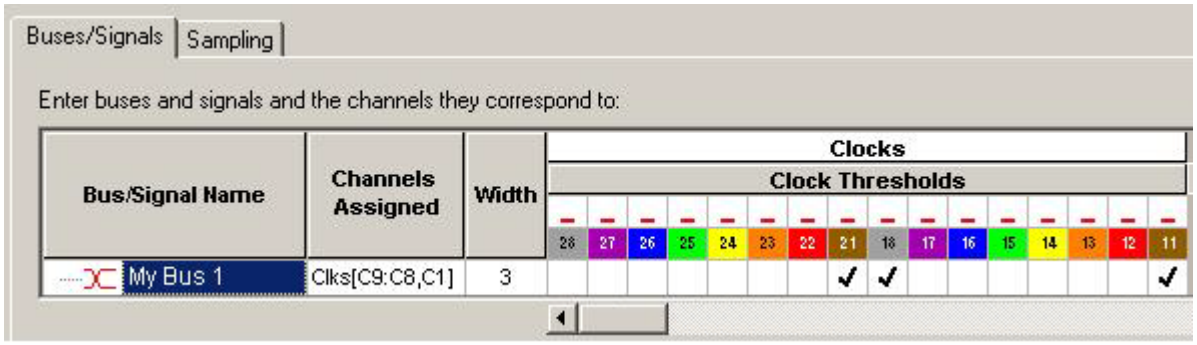
Pod and Channel Naming Conventions in U4154A/B Logic Analyzer

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 AXIe 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 U4154A/B Logic Analyzers

You can change the settings for the probes used with the U4154A/B 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.

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 only supported for U4154A logic analyzers.

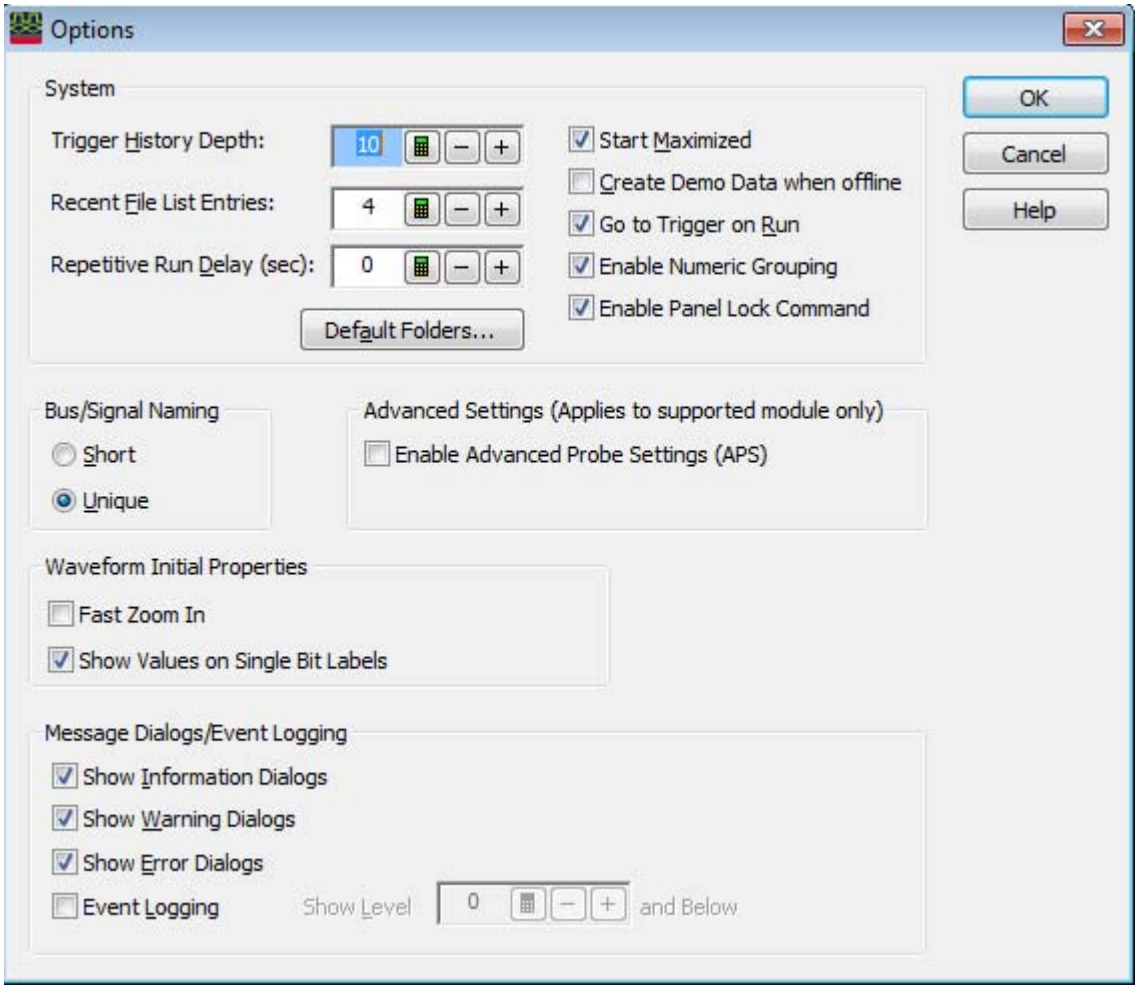
To change probe settings, you must first enable the probe settings and then adjust these settings. These procedures are described below.

To enable Advanced Probe Settings (APS) for the Logic Analyzer Module

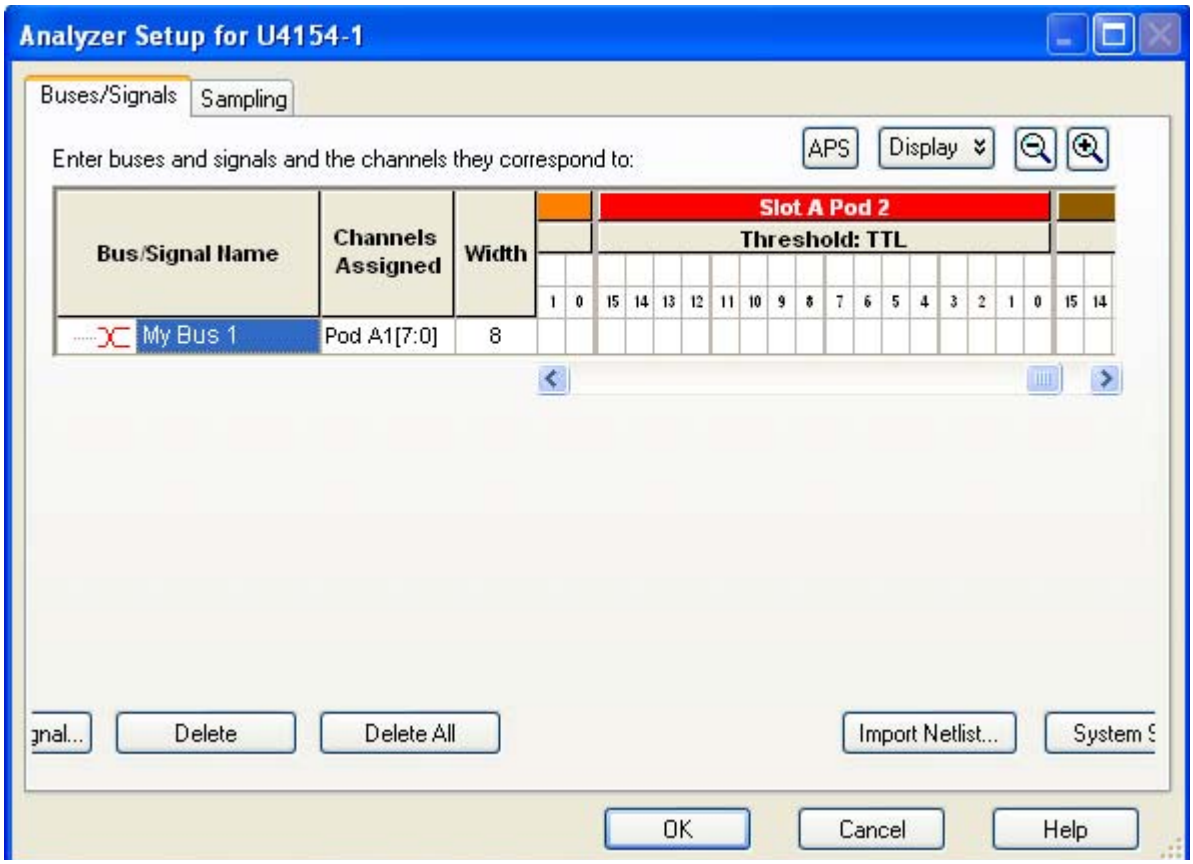
1 From the menu bar, select **Edit** > **Options**.

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.

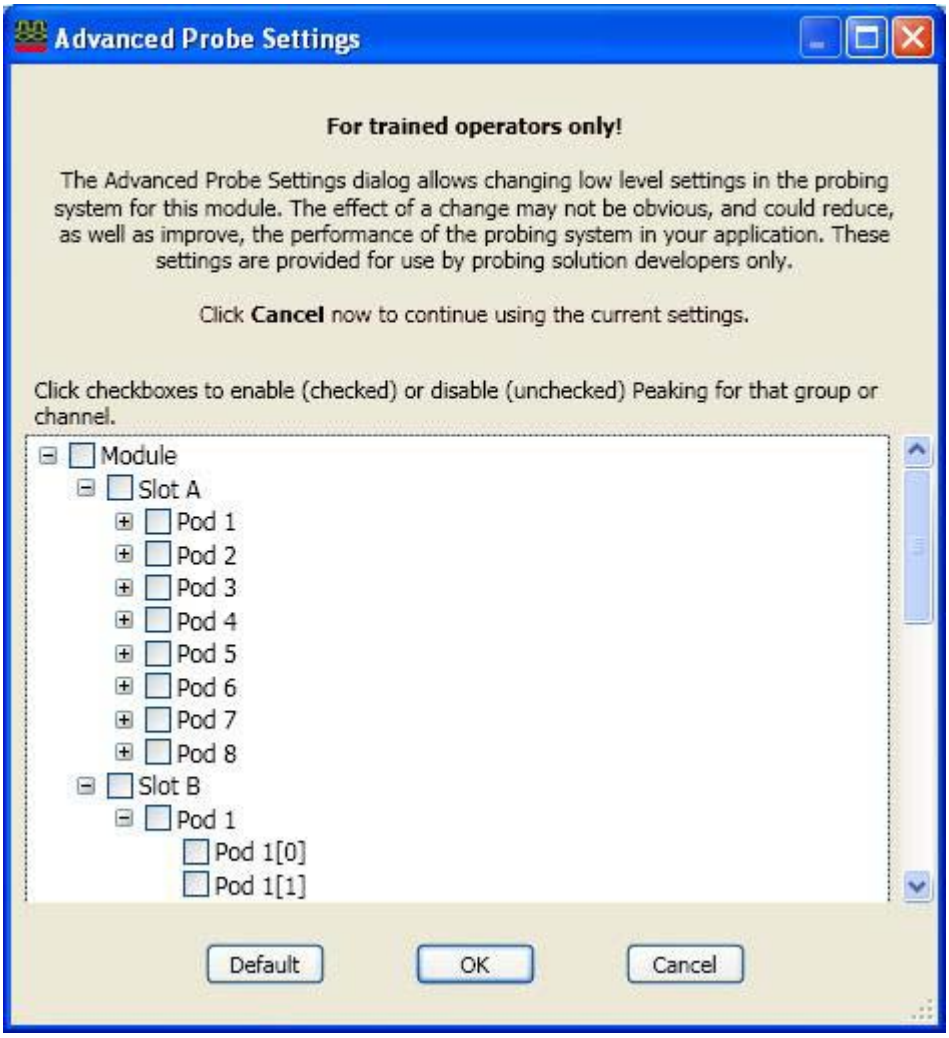


To change the Advanced Probe Settings for the Logic Analyzer Module

1 From the menu bar, select **Setup** > **Bus/Signal**.

2 In the **Buses/Signals** tab of the Analyzer Setup dialog, click the **APS** button.

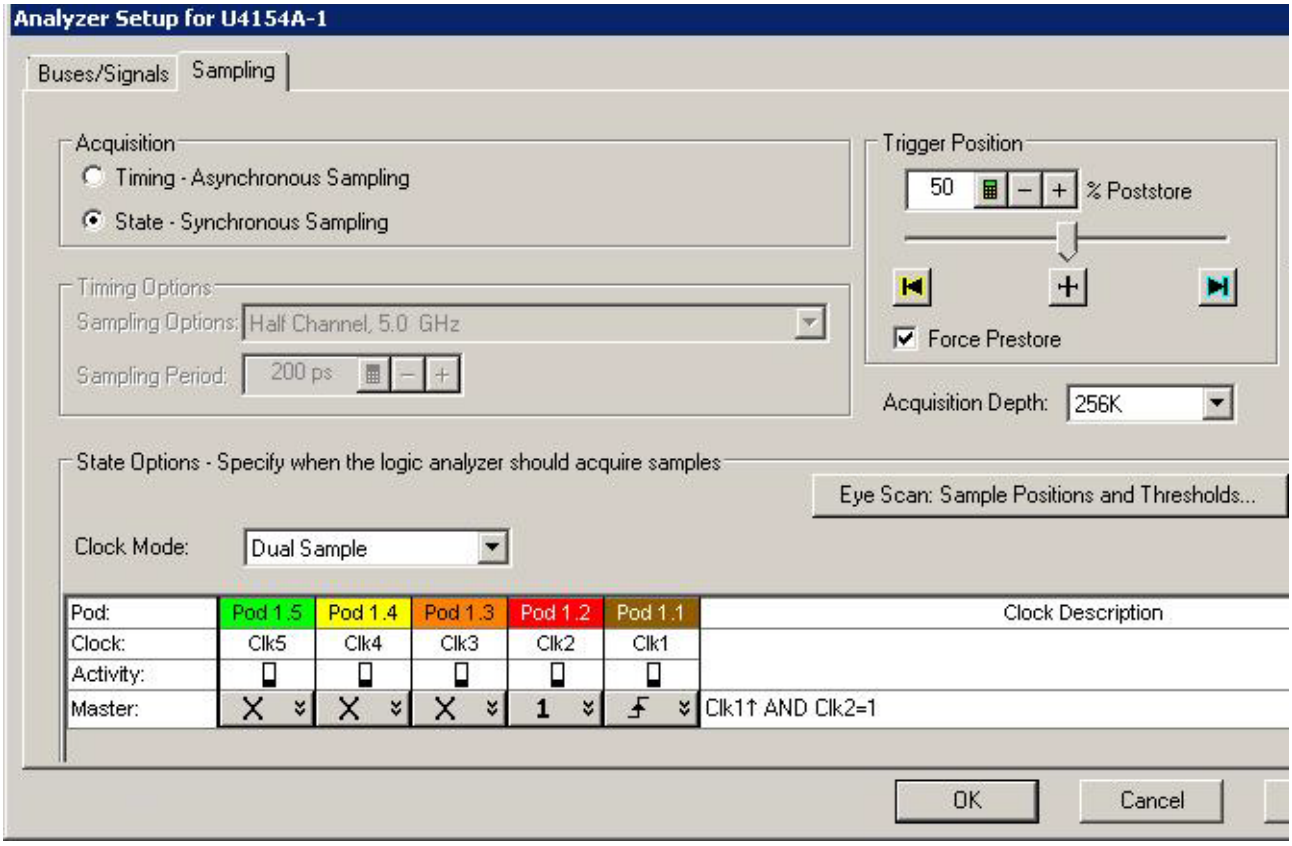
The **Advanced Probe Settings** dialog is displayed. All the logic analyzer cards that you installed in different slots of the chassis to make up the logical module are displayed. On expanding a slot, the pods available for that logic analyzer card are displayed. Each pod further expands to the supported 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 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.

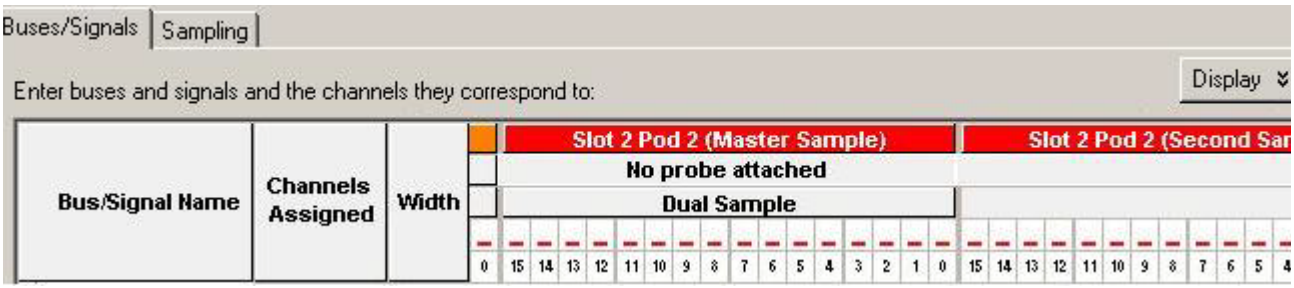
U4154A provides the following two license options for state mode speeds (2.5Gb/s and 1.4Gb/s) at which the state sampling clock matches input clock edges from the device under test.

- **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.

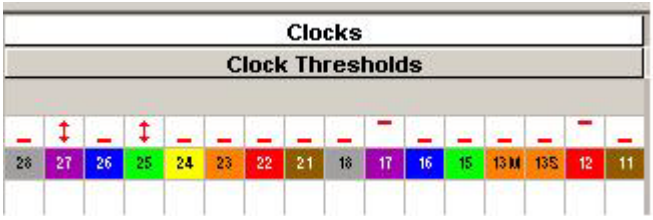
State Sampling Clock Mode

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 Busses/Signals tab of the Analyzer Setup dialog box (see "To set up the dual sample sampling clock mode" on page 112). 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 111
- "To set up the dual sample sampling clock mode" on page 112

State Sampling Clock and Qualifiers

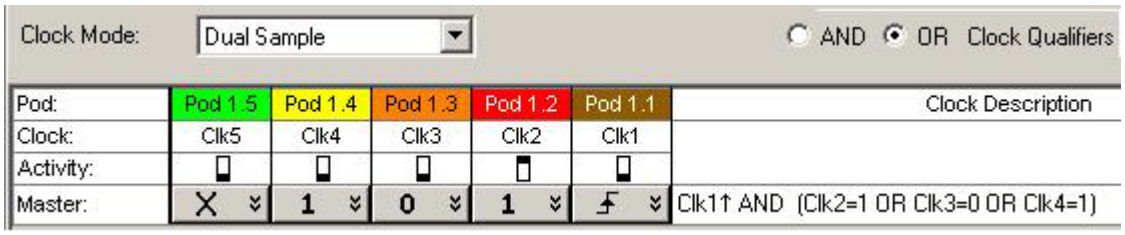
In U4154A, only one state sampling clock is provided. The state sampling clock is the clock of Pod 1 of the master card (the lower middle level card in the set). To know which card is the master card in the set, click **System Summary** button in the Buses/Signals tab of the Analyzer Setup dialog box.

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.



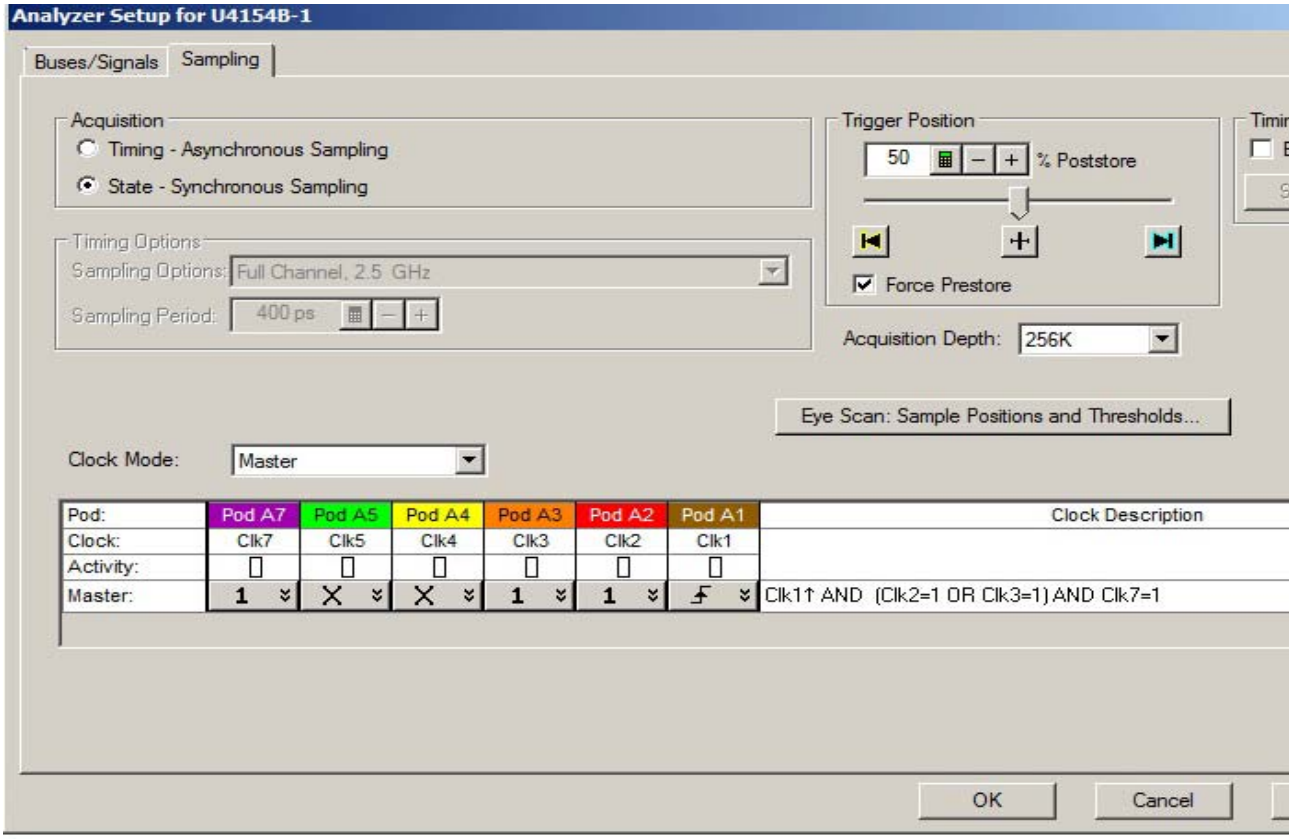
You can use the "AND"/ "OR" conditional operators with the state clock qualifiers to add conditions and completely describe the clock as a combination of edges and highs or lows.

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.

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.

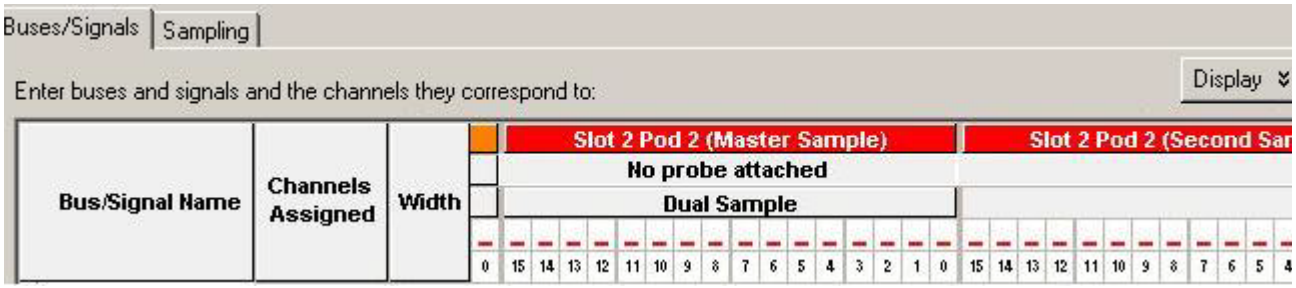
- **2.8 Gb/s speed** - This is the default state speed of U4154B. If, while acquiring data, the clock frequency is found to exceed this maximum available state speed, an error message is displayed and the data is discarded.

- **4 Gb/s speed** - If you have the 02G license for U4154B module, then the maximum available state speed of U4154B is automatically set to 4 Gb/s.

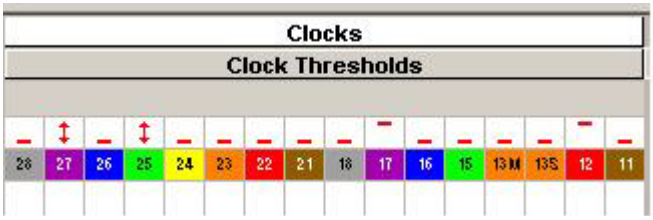
State Sampling Clock Mode

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 **Busses/Signals tab** of the **Analyzer Setup** dialog box (see "To set up the dual sample sampling clock mode" on page 112). 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 111
- "To set up the dual sample sampling clock mode" on page 112

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 Busses/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.

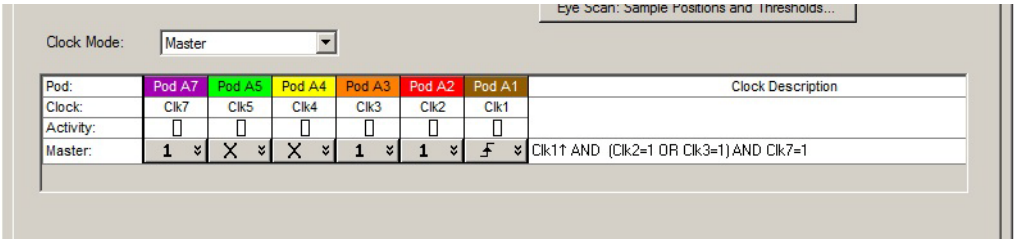
There are following five qualifiers available for the state clock. You cannot use the signals from the other pods clock inputs as the sampling clock qualifiers. You can use the "AND"/ "OR" conditional operators with these state clock qualifiers to add conditions and completely describe the clock as a 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^ AND (Ck4=0 OR Ck5=1) AND 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.



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 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 U4154B channels. Refer to the topic “Setting up and Running Eyescans in U4154A/B Logic Analyzer" on page 508 to learn more.

See Also

- To set up the state sampling clock
- “Pod and Channel Naming Conventions in U4154A/B Logic Analyzer" on page 497

Specifying Advanced Triggers in the U4154A/B Logic Analyzer

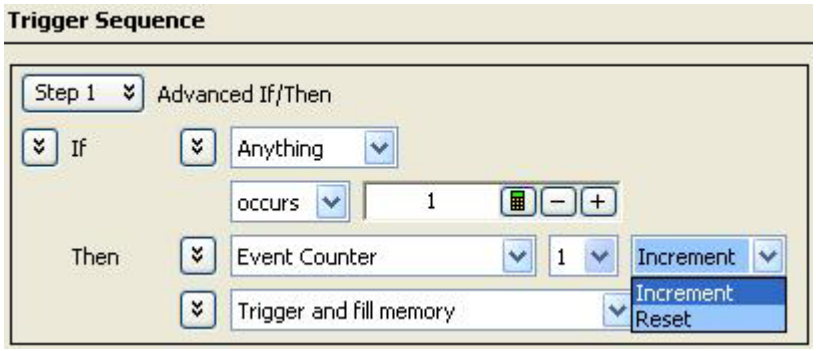
When setting up advanced triggers, the U4154A/B logic analyzer has some differences from previous logic analyzers. This topic highlights these differences:

- Up to eight trigger sequence steps are available.
- 2.5 GHz trigger sequencer enables reliable trigger and capture of DDR3 signals at speed.
- There is one timer available in the timing sampling mode as well as the state sampling mode.

- An Occurrence counter is available in each sequence step to count eventual occurrences. You can achieve a consecutive occurrence counter by using "Else if" clauses with the "Reset occurrence counter" action.

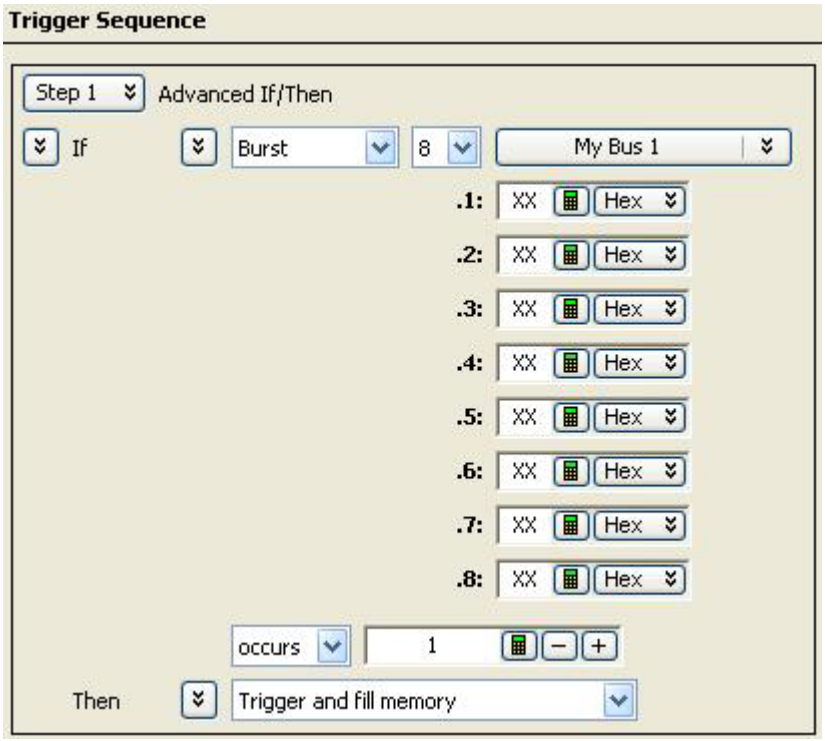
Event counter

In a U4154A/B trigger sequence step, you can add an "Event counter" as the trigger action. You can Increment or Reset this event counter in the trigger sequence.

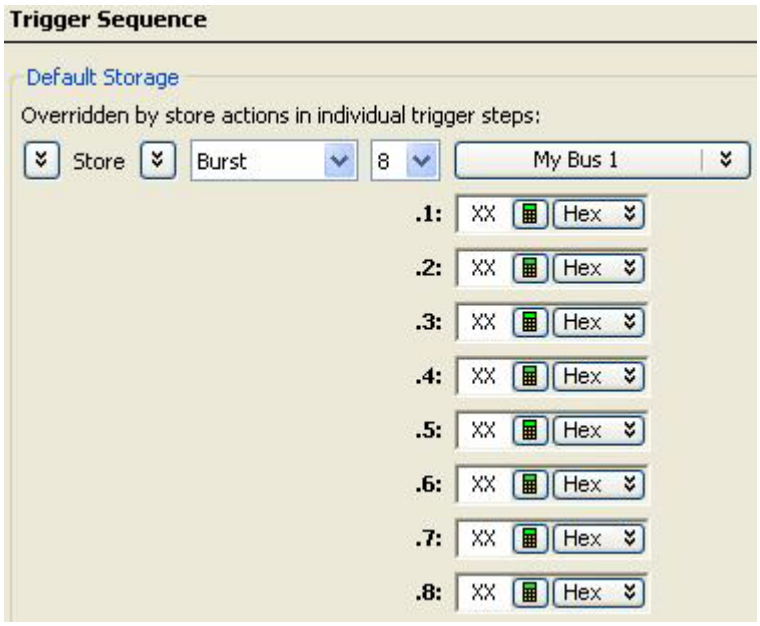


Specifying Burst Patterns

You can specify the Burst pattern event type in a trigger sequence step. The burst level has been increased to 8 in U4154A/B Logic Analyzer allowing you to specify a maximum of eight samples in the burst pattern event.



In the **Default Storage** of a trigger, you can specify a maximum of eight burst patterns to store burst samples as the default storage.



Setting up and Running Eyescans in U4154A/B Logic Analyzer

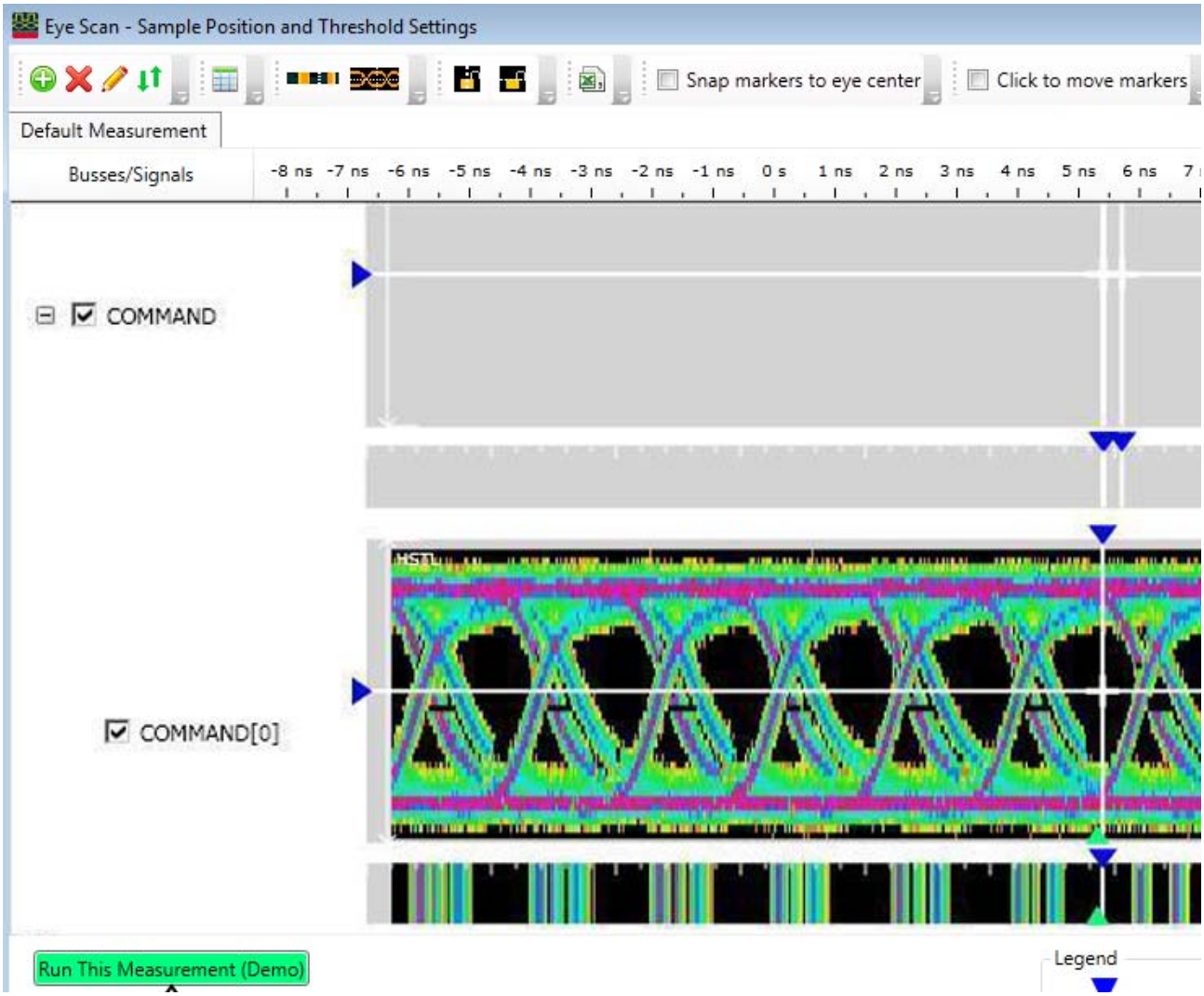
The U4154A/B logic analyzer allows you to set up and run eyescans for automatically adjusting the state mode sampling positions and threshold voltages of address and control signals so that data is sampled when it is valid. This topic describes how you can set up and run the eyescan measurements when 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 step1. 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:

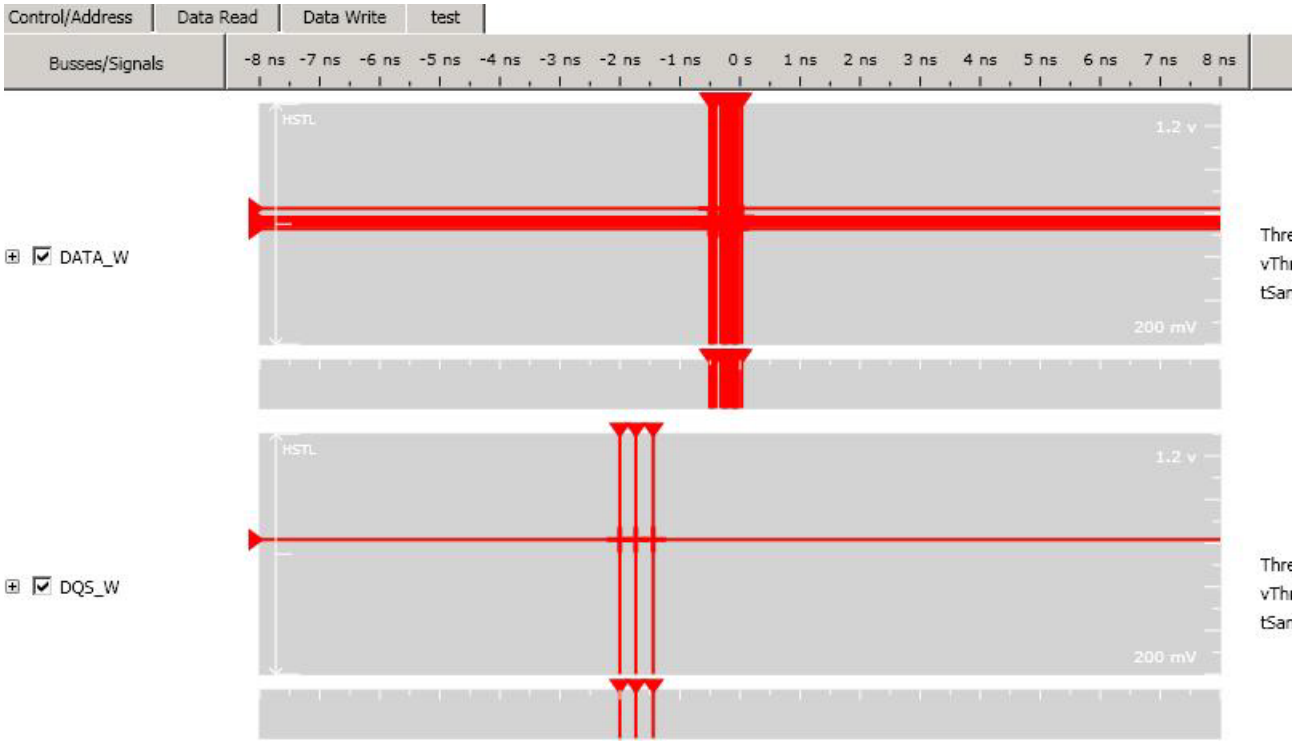
a Click **Assign Buses / Signals to Measurements** toolbar button.

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**.

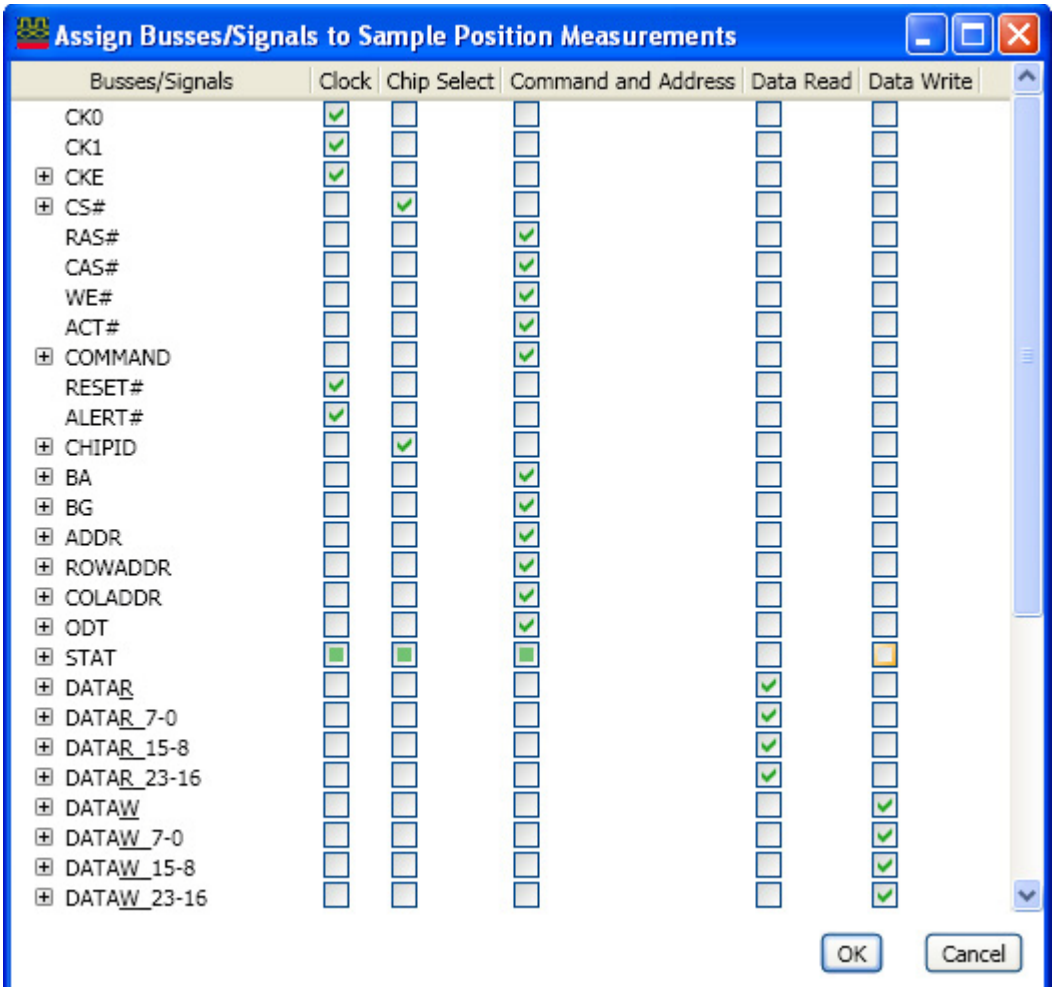
After you have created an eyescan measurement, you can edit the measurement to customize how you want the U4154A/B Logic Analyzer to run the eyescan measurement. You do this by clicking **Edit Current Measurement** button in the Eye Scan - Sample Position and Threshold Settings dialog box. Refer to the 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

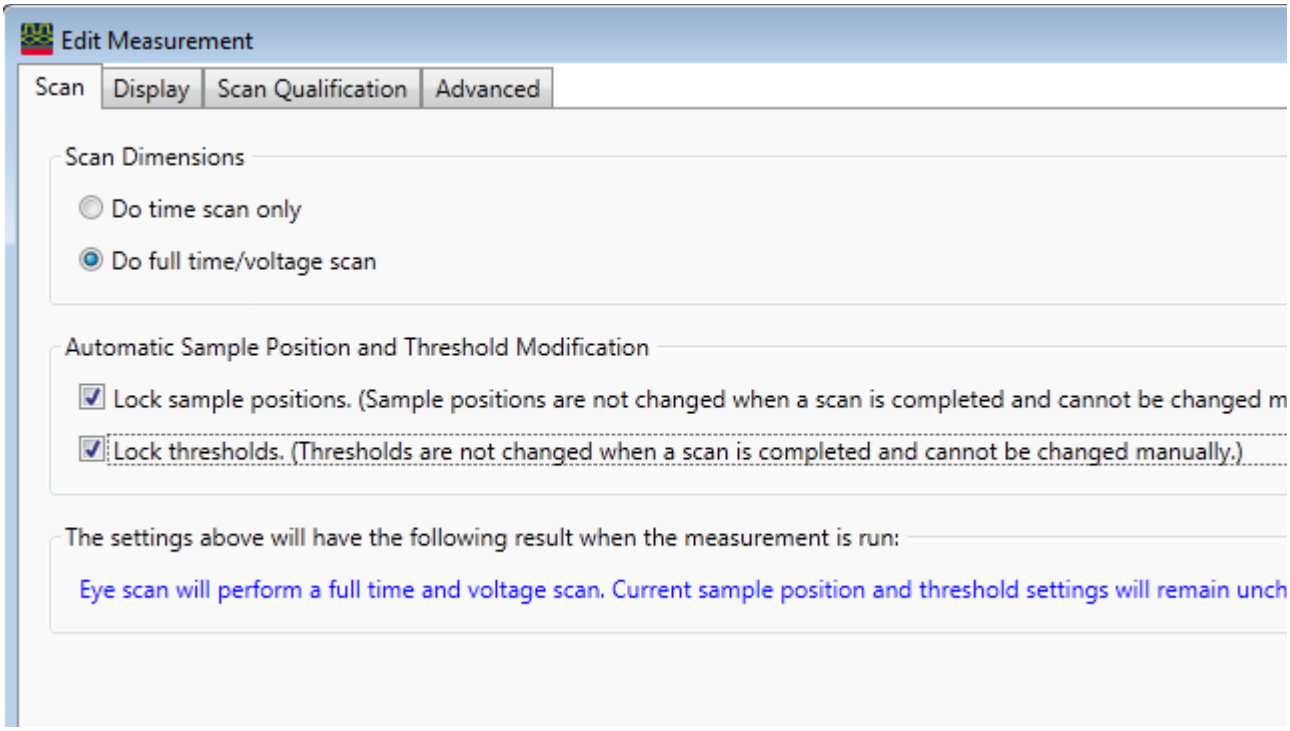
If you need to create multiple eye scan measurements, you can include a signal in only one of these measurements for setting the signal's sample position. This ensures that you get the eye scan results for a signal in only that measurement. If you assign a signal that already exists in a measurement to another measurement, then the signal is automatically removed from the earlier assigned measurement.



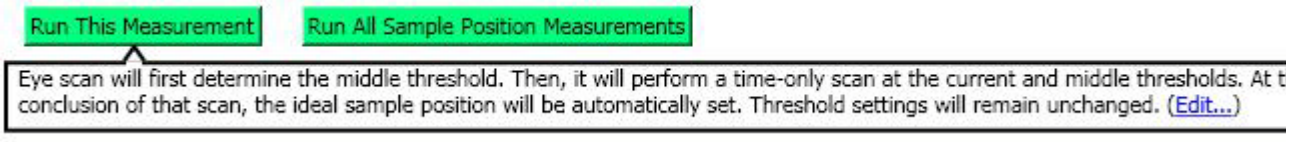
Customizing an Eyescan Measurement

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.

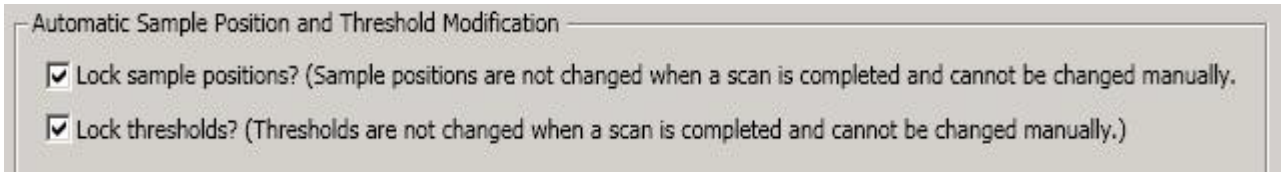
Types of eyescans:

- **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.

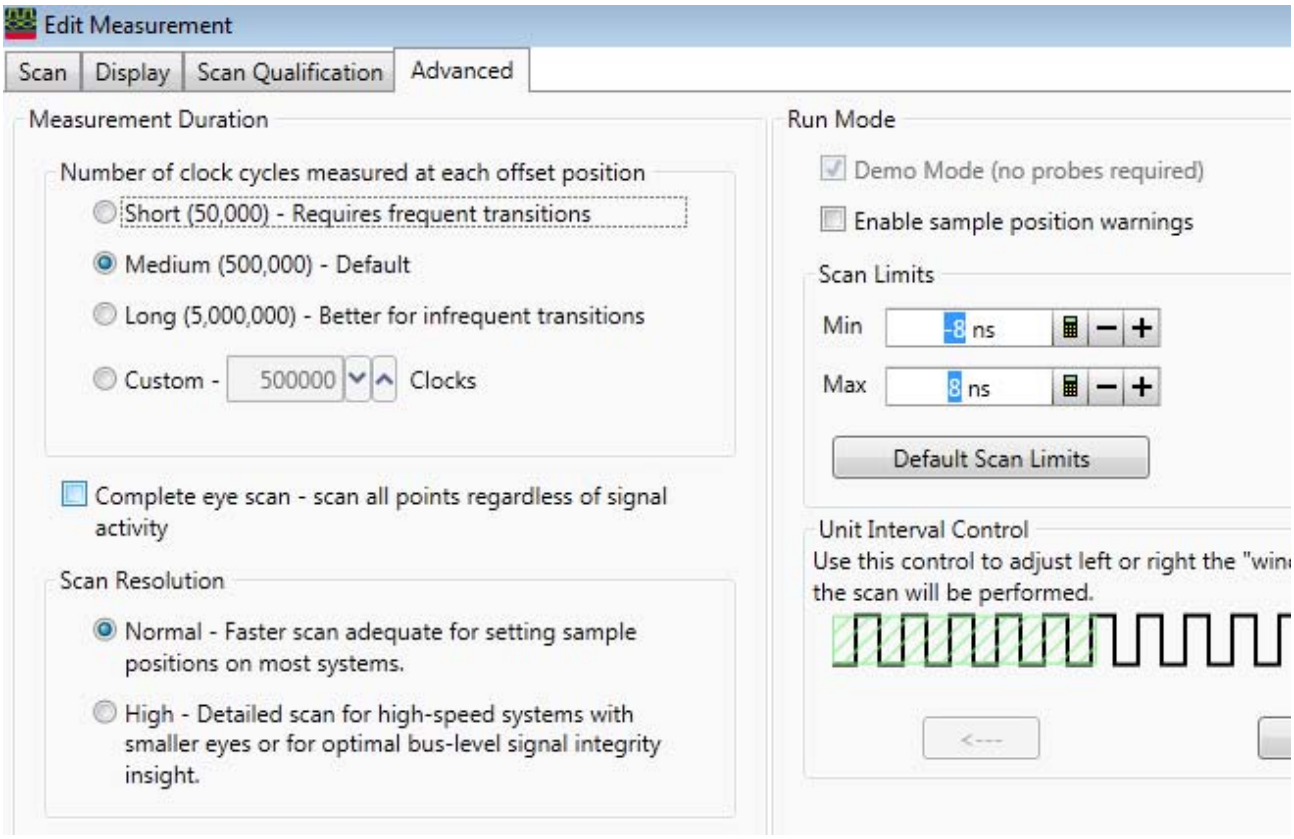


Setting eyescan resolution

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.

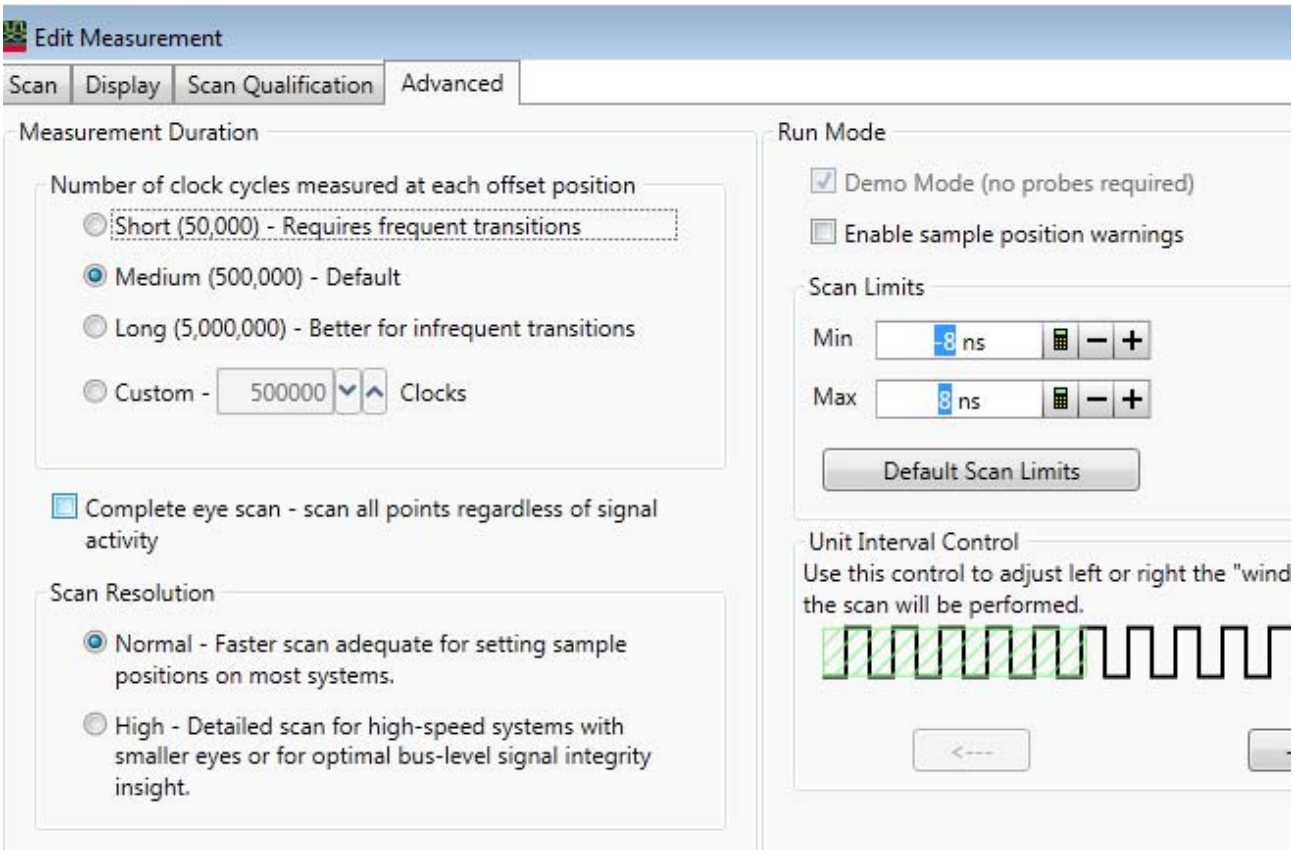
The following screen displays these two resolution options.



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

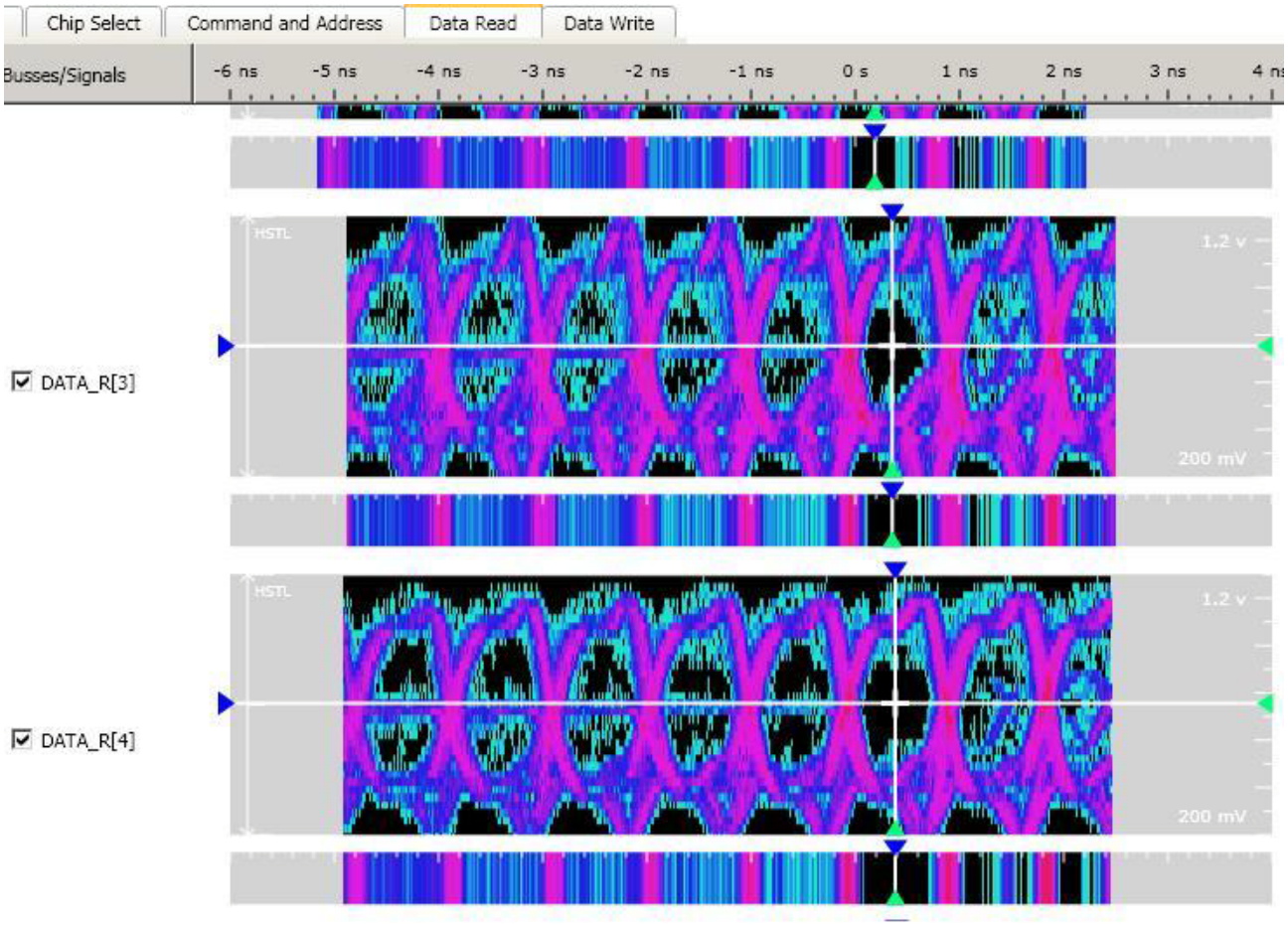
By performing full time and voltage scans, U4154A/B is able to give you a map of transitions detected in small windows of time and voltage. These scans are called eyescans.

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.

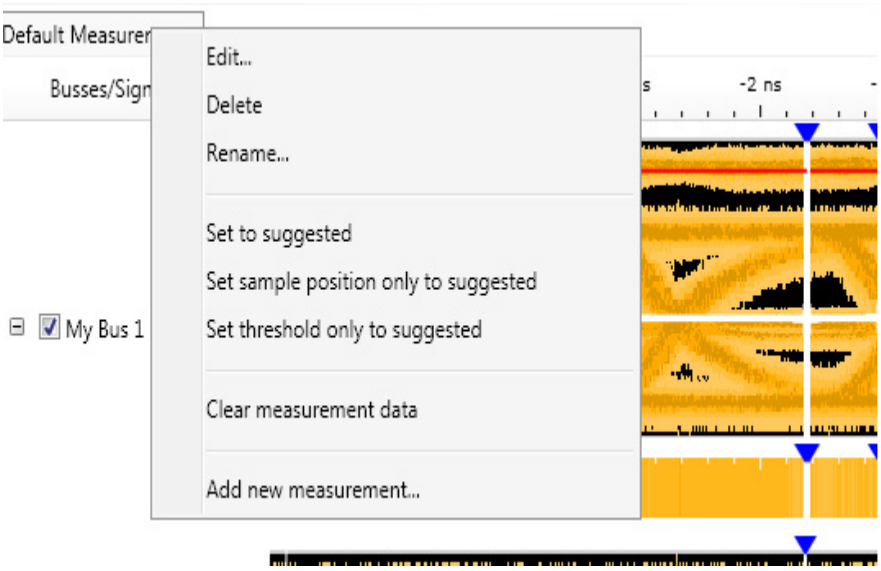
The following screen displays the results of a time and voltage eye scan run on the data read signals. The current settings have been automtaically adjusted by the eye scan based on the optimal settings determined during the eye scan.



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

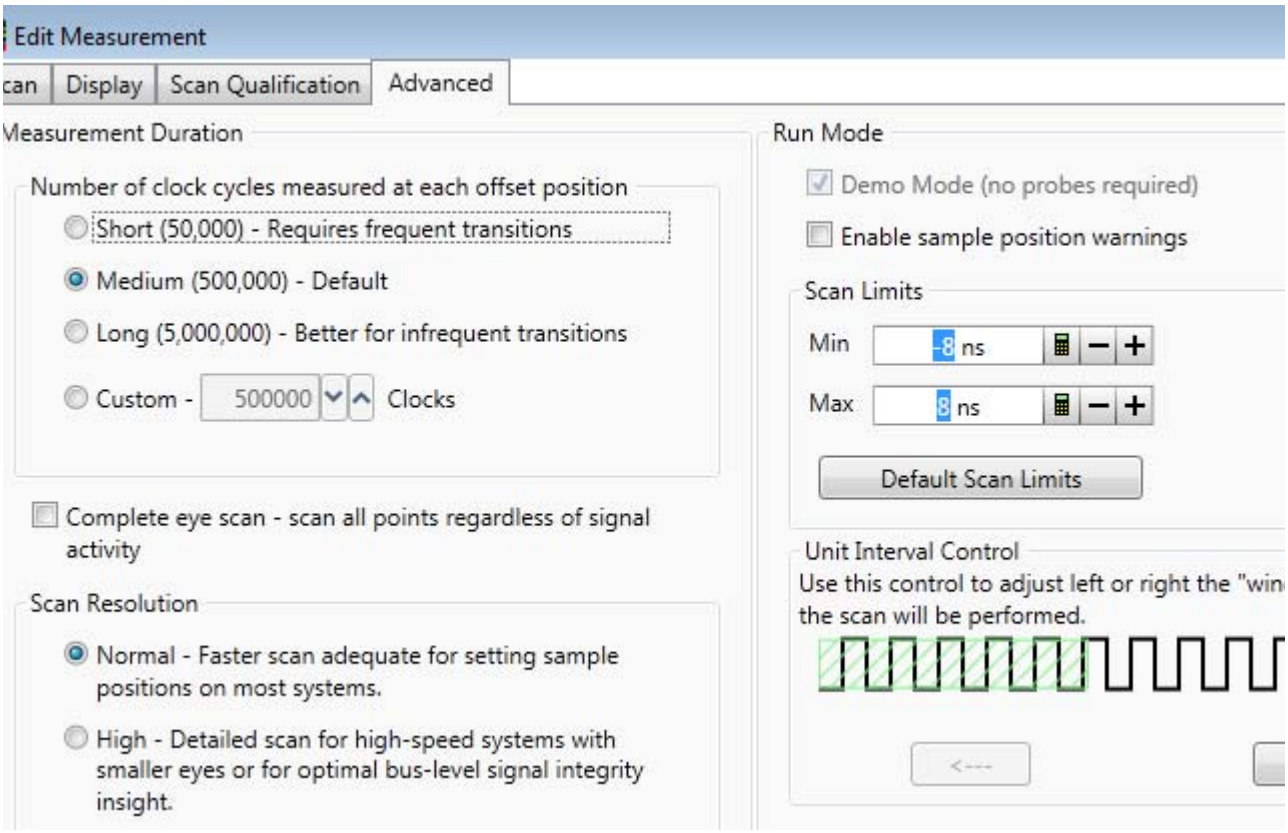
After viewing the eyescan results, if needed, you can mark the ideal sample position for each signal by manually moving the sample position markers in the eyescan. You do this:

- 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

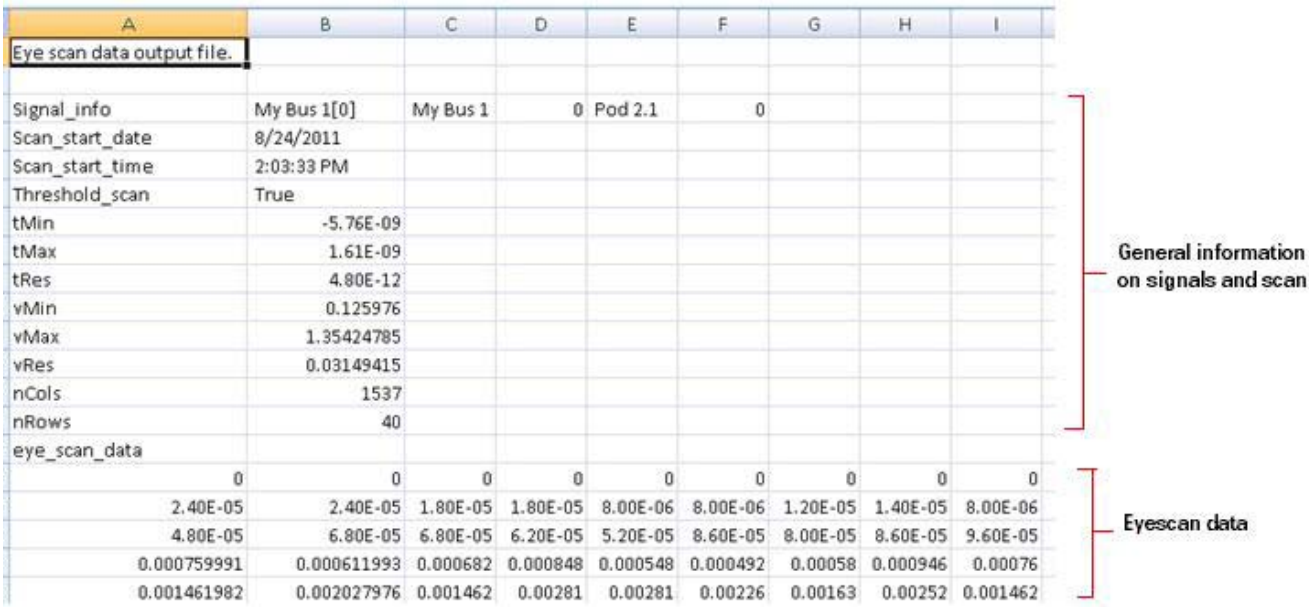
1 Click the **Export Measurement Data to CSV File** toolbar button in the Eye Scan - Sample Position and Threshold Settings dialog box.

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:

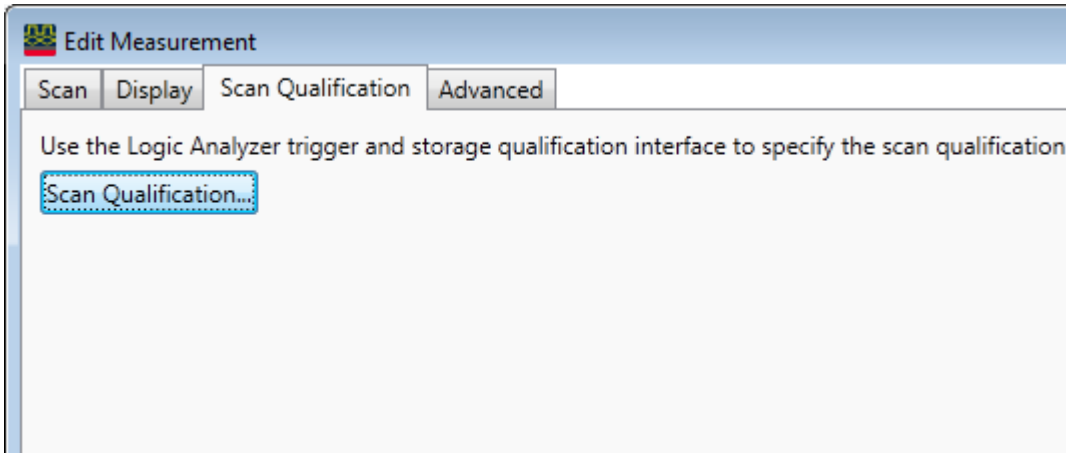
tCol = tMin + (colNum * tRes)

Modifying General or Target-specific Scan Qualification

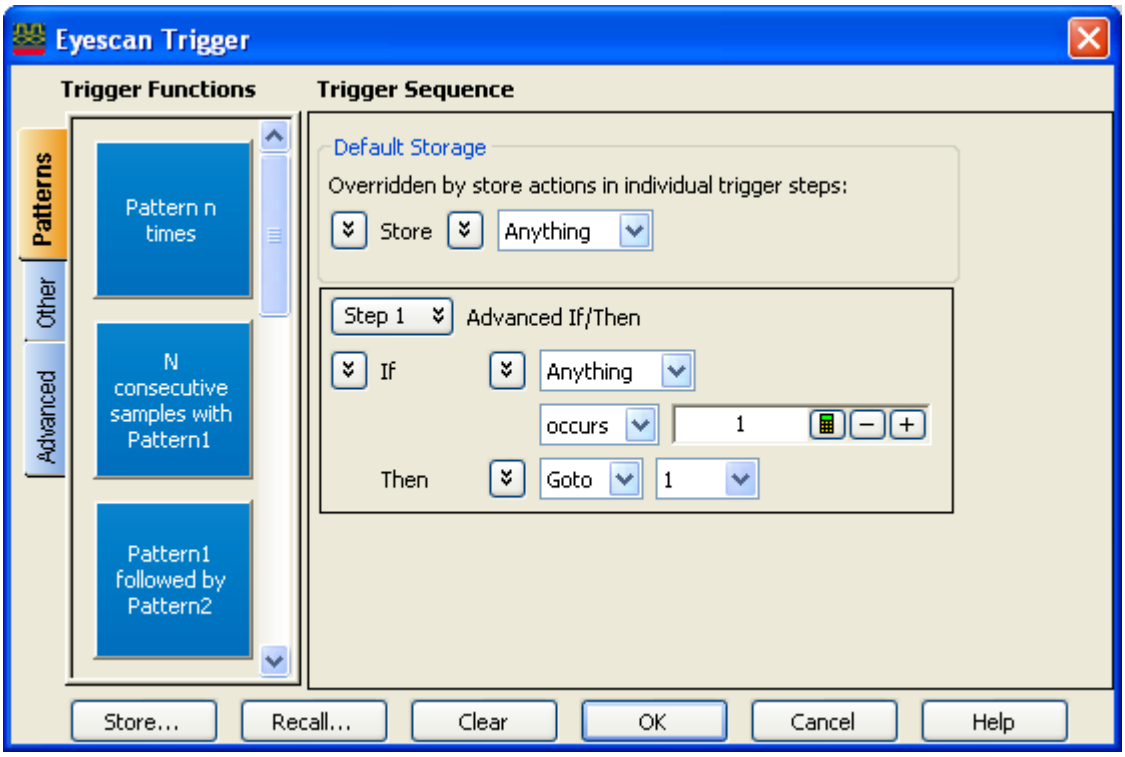
You use the **Scan Qualification** tab in the **Edit Measurement** dialog box invoked from the Eye Scan - Sample Positions and Threshold Settings dialog box to modify the scan qualification for general as well as target-specific scans such as DDR scans. This tab displays different fields based on whether 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.

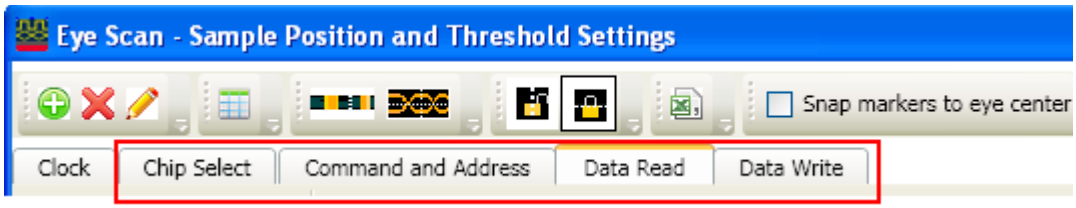
2 Click the **Sampling Setup** link of the module displayed in the DDR setup.

   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.



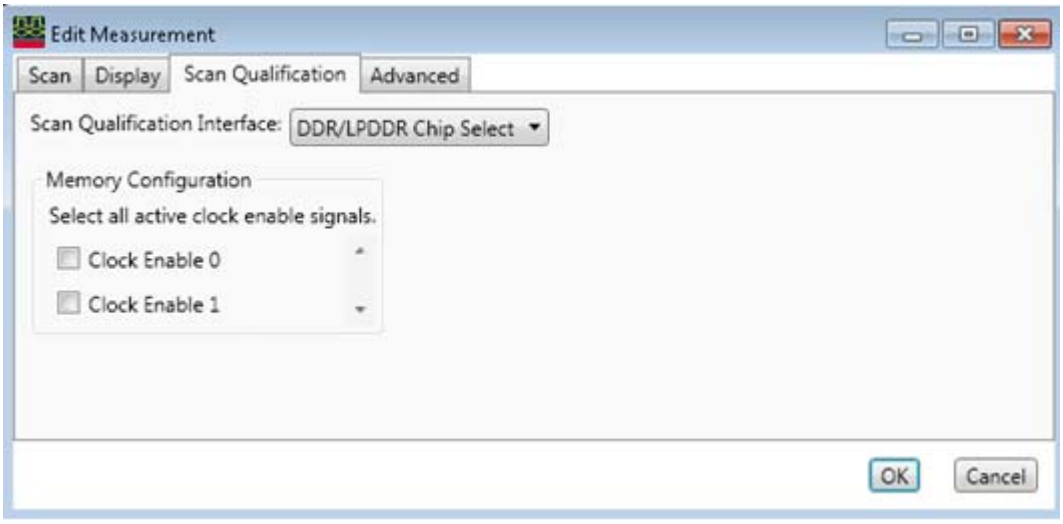
6 Click the **Edit Current Measurement** toolbar button.

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:

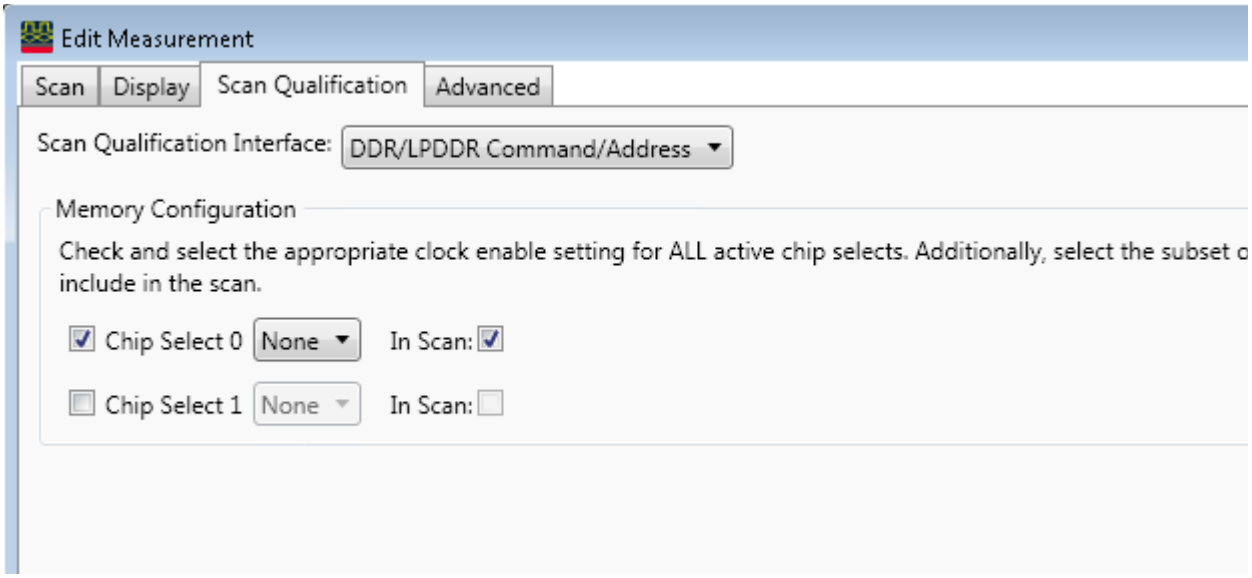
   - DDR/LPDDR Chip Select option - to modify the scan qualification for chip select signals.
   - DDR/LPDDR Command and Address option - to modify the scan qualification for command and address signals.
   - DDR Data Read option - to modify the scan qualification for data read signals.
   - DDR Data Write 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:



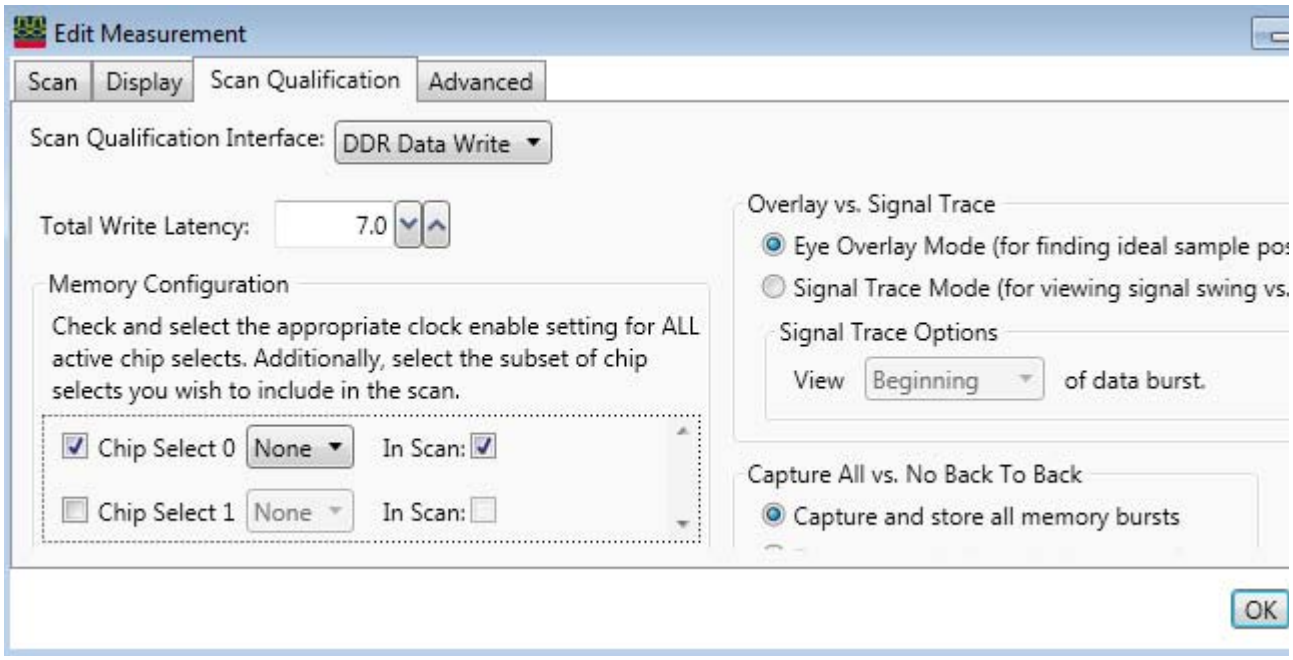
9 For the DDR/LPDDR Chip Select signals, make the appropriate selections based on the active clock enable signals.

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:

a 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 tDQSCK parameters are greater than one full clock cycle, then you must add these values (rounded to the nearest integer) in the Total Latency values.

b In the **Memory Configuration** section, choose the appropriate **Chip Select(s)** for the memory path being traced.

c 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.

d 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.

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 335).

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 124).)

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 335)).

  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 337).

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 531)

Specifications and Characteristics

Describes the specifications, characteristics, and requirements of supported logic analyzers and logic analysis systems.

- U4154A Logic Analyzer Specifications and Characteristics (see page 528)
- U4154B Logic Analyzer Specifications and Characteristics (see page 531)
- 16850-Series Logic Analyzer Specifications and Characteristics (see page 531)

See Also

- What is a Specification (see page 536)
- What is a Characteristic (see page 536)

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 528)
- State (Synchronous) Analysis Mode (see page 529)
- Timing (Asynchronous) Analysis Modes (see page 529)
- Trigger Characteristics (see page 529)
- Other (see page 530)
- Power Requirements (see page 531)
- Chassis Compatibility (see page 531)
- Environmental Characteristics (see page 531)

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
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

Menu	Description
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
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.
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 536)
- What is a Characteristic (see page 536)

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.

See Also

- What is a Specification (see page 536)
- What is a Characteristic (see page 536)

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 532)
- Probes (see page 532)
- Timing Zoom (see page 532)
- State (Synchronous) Analysis Mode (see page 532)
- Timing (Asynchronous) Analysis Mode (see page 534)
- Other (see page 535)
- General Information (see page 535)
- Environmental Characteristics (see page 535)

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

	Default	Option 700
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	
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 · 3 flags · 1 arm in	
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/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
Maximum trigger sequence speed	700 MHz
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 · 3 flags · 1 arm in
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

	Conventional and Transitional Timing
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:

100-240V ± 10 %, 50/60Hz, 400 W max

NOTE The mains supply voltage fluctuations are not to exceed ± 10 % of the nominal supply voltage.

Weight:

Max Net	Max Shipping
15.0 kg (33.0 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 (+41°F to 104°F).
Humidity:	80% to temperatures up to 31°C decreasing linearly to 50% rH at 40°C; /// max 80% rh, non-condensing.
Altitude:	2000 m (6,561 ft)

See Also
- What is a Specification (see page 536)
- What is a Characteristic (see page 536)

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 536)

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 536)

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 542) in a frame (see page 539). 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 or glitch.

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 542)s in the back for the insertion of logic analyzer card (see page 538)s. Multiple frames can be connected together to form a multiframe logic analysis system.

G

glitch

A glitch occurs when two or more transitions cross the logic threshold between consecutive timing analyzer samples.

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 move voltage detail. Consequently, a logic analyzer can capture a greater amount of overall execution time.

M

macro

- In pattern generator modules, "macros" (in the online help) are like subroutines in a programming language; they can have parameters and they can be called multiple times in the vector sequence to generate repeated or similar output vectors.
- 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 544)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 538)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, and a single card or several card module can be split into two modules. 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 538). 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 540). If a module (see page 540) 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 538), or in which slot (see page 542)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 537).

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 539) where card (see page 538)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 know 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 know as state mode.

T

target system

See device under test (see page 538).

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 104) or to read current positions, waveform transition widths, or trigger specifications (when setting up quick triggers (see page 129) with mouse).

trace

See acquisition (see page 537).

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, 133, 142
< operator, pattern trigger, 133, 142
<= operator, pattern trigger, 133, 142
= operator, pattern trigger, 133, 142
> operator, pattern trigger, 133, 142
>= operator, pattern trigger, 133, 142

Numerics

01G license, 502
02G license, 502
10 MHz CLOCK IN input, 46
16700 ASCII file format, 402
16700 ASCII format files, 46, 183
16850-series logic analyzer front panel operation, 487
16850-series logic analyzer notes, 526
16850-series logic analyzer specifications and characteristics, 531
8b, 275

A

abbreviated marker name, 223, 231
about, help, 369
absolute time, 218
accumulate waveforms, 206
acquiring data, 127
acquisition, 537
acquisition depth, 537
acquisition errors, 306
acquisition memory depth, 115, 124, 396, 493, 495, 526, 531
acquisition memory depth and channel count trade-offs, 335
acquisition mode, 396
acquisition mode, choosing, 106
acquisition mode, setting, 59
acquisition mode, state, 108
acquisition mode, timing, 106, 107
acquisition time, transitional timing, 337
action, 537
action statements, 139, 140, 141
action statements, reading, 139
action, counter, 157
action, flag, 158
action, reset occurrence counter, 140, 157
action, storage control, 159
action, timer, 157
actions and events, trigger, 138
actions in trigger sequence, 343, 392
actions, inserting or deleting, 156
activation, software licenses, 295, 424
active software licenses, 294, 423
activity (bus/signal), 93
activity indicator, 537
activity indicators, 395
activity indicators in Listing window, 214
activity indicators in Waveform window, 203
Add Application dialog, 404
add external scope, 361
add icons to toolbar, 375
add logic analysis system, 421
Add or Remove Logic Analyzer Software tool, 299
add software, 299
adding a bus or signal, 94
adding a folder, 105
adding connections, 283
adding display windows, 283, 289
adding system to use list, 77
adding tools, 283
address alignment, trigger in Source window, 427
adjustable delay on state sampling clock input, 339
advanced 2-way branch trigger function, state mode, 462
advanced 2-way branch trigger function, timing mode, 450
advanced 3-way branch trigger function, state mode, 463
advanced 3-way branch trigger function, timing mode, 450
advanced 4-way branch trigger function, state mode, 463
advanced 4-way branch trigger function, timing mode, 451
advanced clocking, 114
Advanced Clocking Setup dialog, 391
advanced if/then trigger function, state mode, 462
advanced if/then trigger function, timing mode, 449
advanced If/Then trigger functions, 440
advanced pod assignment mode, 429
advanced setup, 361
advanced trigger, 127, 138, 361, 372, 537
Advanced Trigger dialog, 392
advanced trigger example, 331
advanced trigger functions, state mode, 452
advanced trigger functions, timing mode, 440
agLogicSvc.exe service, 320
agSymbolBuild.exe, 120
agSymbolQuery.exe, 121
ALA format, 466
ALA format configuration files, 176, 188, 190
ALA format configuration files, automatic module mapping, 409
ALA format configuration files, offline analysis, 195
ALA format configuration files, won't open, 305
ALA vs. XML format, when to use, 352
ALB module binary file format, 181, 473
ALB module binary file header format, 467
alias bus/signal name, 105
align trigger with, timing zoom, 438
aligning reference data, 251
alignment, bus/signal column, 218
alignment, packet column, 281
alignment, Set Quick Trigger, 258
all reference data, 252
altitude, 531
analog properties, 210
analog signal row properties, 209
analog signal, Voltage number base in Listing, 218
analog signals, overlaying, 200
analog signals, voltage markers for, 234
analogy, conveyor belt, 341
analysis probe, 73, 537
analysis tool development API, 53
Analyzer Setup dialog, 394
Analyzer Setup toolbar, 372
analyzer triggering, understanding, 341
analyzer, splitting, 88, 429
analyzing captured data, 187
analyzing listing data, 212
analyzing waveform data, 197
AND events, 152
AND'ed find events, 246
anything event, 152
API, tool development, 53
application (external), adding to Tools menu, 403

arm from another module, 164
arm from another module, wait for, 449, 461
arm in from event, 152, 154
arming, 537
arrange icons, 368
ASCII format symbol file, 476
Ascii number base, 142, 209, 218
ASCII symbol file, creating, 121
ASCII text file, print to, 290
Assign Names dialog, 96
asynchronous sampling, 106, 537
asynchronous sampling mode, 396
auto-connect at application startup, 407
auto-connect, selecting logic analysis system for, 82
automatic FPGA dynamic probe set up, 48
automatic threshold voltage adjustment, 49
automatically mapping modules, 409
autosize columns, 218
available status, 79
AXIe chassis slots per card, 528

B

background color, Listing window, 215, 216
background color, marker, 223, 233
background color, Source window, 258
background color, timing zoom listing, 216
background color, timing zoom waveforms, 206
background color, Waveform window, 204, 205
base, number, 103, 209, 218
base, symbol, 419
beginning of acquisition, 537
beginning of data, 199
Beginning of Data marker, 288
beginning of data, go to, 226
beginning of data, go to in Listing window, 213
best-fit algorithm, module mapping, 409
big endian, 101
Binary number base, 142, 209, 218
bit order, 99
bit reordering, 52, 101
bits, 537
bits of a bus, renaming, 46
block boundary addresses, 258
BNC connectors, trigger in/out, 164
boolean expressions, 344
boolean expressions in trigger sequence, 343
boolean expressions, setting parentheses, 155
boolean expressions, triggering capability, 342
borrow floating licenses, 298, 426
borrow licenses messages, 426
branch expression too complex, 309
branches, 345
browsing source files, 254
buffer, reference, copying data to, 250
burst patterns in U4154A, 507
bus, 537
bus analysis tools, new, 362
bus clock, data valid relative to, 339
bus pattern trigger functions, timing mode, 440
bus pattern triggers, 133, 142
bus width, 93
bus, group signals into, 204
bus, renaming the bits, 96
bus, source code correlation, selecting, 256
bus, source correlation, 428
bus/signal column alignment, 218
bus/signal column properties, 217
bus/signal column width in Listing window, 218
bus/signal column, finding a, 214
bus/signal columns, re-arranging, 214
bus/signal data color, Listing window, 217
bus/signal data to export, 177
bus/signal data, exporting to CSV file, 52
bus/signal errors, 306
bus/signal event, 152
bus/signal folders, adding, 105
bus/signal name, alias, 105
bus/signal names, quick pick, 52
bus/signal names, setting up, 58
bus/signal names, sorting, 105
bus/signal naming, short or unique, 412
bus/signal number base, 209, 218
bus/signal patterns, finding, 240, 241
bus/signal patterns, finding complex, 246
bus/signal row properties, 207
bus/signal row, finding a, 201
bus/signal setup, 93, 361, 372
bus/signal values displayed as symbols, 220
bus/signal values, symbolic, 123
bus/signal, adding a new, 94
bus/signal, assigning channels in default bit order, 98
bus/signal, assigning channels in selected bit order, 99
bus/signal, deleting a, 95
bus/signal, renaming a, 95
buses, expanding/collapsing, 204
Buses/Signals tab, 395
buses/signals, define by importing netlist, 411
buses/signals, exporting data from selected, 416
buses/signals, inserting/deleting in Listing window, 215
buses/signals, inserting/deleting in Waveform window, 204
buses/signals, load warning, 311
bus-specific probes, 73
buttons, 16800-series logic analyzers, 487

C

calibration, 536
calibration failed, 309
calibration procedure, 536
cancel, 365
cancel operation, 374
cancel processing, module/tool/window, 430
captured data, 127, 538
captured data, analyzing, 187
captured data, comparing, 250
captured data, exporting to module binary (ALB) files, 181
captured data, exporting to module CSV files, 179
captured data, printing, 290
captured data, saving, 176
card, 538
card, logic analyzer, 399
cards (logic analyzer), combining, 88
cards in frame, 434
cards in module, 436
cascade windows, 368, 371
center about edges, 198
center about marker pair, 226, 364, 373
center alignment, bus/signal columns, 218
center rectangle, go to source line of, 255
center rectangle, showing/hiding, 216
center screen, placing marker at, 224
changing the display scale, 198
changing the time/division, 198
channel, 538
channel and pod connections, 436
channel assignment, split analyzer, 429
channel count and memory depth trade-offs, 335
channel counts, 528, 531
channel naming conventions, 333
channel numbers, 395
channel numbers (bus/signal), 93
channel order selection, enable, 99
channel order, reverse, 101
channel reordering, 52, 101
channel width, 107
channels, 493, 495, 526, 528, 531
channels (directions), packet column, 281

channels assigned (bus/signal), 93, 99
channels assigned (bus/signal), reordering, 101
channels on a single time base and trigger, 528, 531
channels per module, 528
channels, assigning in default bit order, 98
channels, assigning in selected bit order, 99
channels, data on unassigned, transitional timing, 337
character names, 275
characteristics, 528, 536
characteristics, 16800-series logic analyzer, 531
chart, viewing Waveform bus data as, 201
chassis, 538
Chat dialog, 397
chat messages, 83
Chat Select Destination dialog, 398
Choose a Protocol Family and Bus dialog, 398
clear favorite/recently used triggers, 417
clear flag, 141
clear flag action, 158
clear trigger sequence, 163
clearing a remote access password, 84
clearing the trigger sequence, 392
clip board, copying module/tool/window status to, 430
clip board, copying screens to, 291
clip board, copying text to, 290
clipped (show) analog property, 210
clock channel, 98, 99, 538
CLOCK IN BNC, 538
clock mode (state sampling), Demultiplex, 110, 111
clock mode (state sampling), Dual Sample, 110
clock mode (state sampling), Master Only, 109, 111
clock mode (state sampling), Master/Slave, 109, 111
clock mode (state sampling), selecting, 109
clock pod/channel naming, 333
clock qualifier hold time, 531
clock qualifier setup time, 531
clock qualifiers, 531
clock setup, state mode, 306
clock threshold voltage, 91
clock, slow or missing, 311
clock, state mode sampling, 330, 331
clock, state mode sampling, setting up, 113
clock, timing mode sampling, 328
clocks, 531
clocks pod, 99
Coff/Stabs symbol files, 121
collapse a bus, 204
color, difference foreground/background, 252
color, filtered data in Waveform window, 206
color, Listing window background, 216
color, overlaid waveform, 205
color, reference data background, 252
color, selected packet box, 279
color, Source window background, 258
color, Source window text, 258
color, waveform, 208
color, Waveform window background, 204
colorize/filter tool, 362
column alignment in Listing window, 218
column autosize, 218
column base in Listing window, 214
column channels (directions), protocol, 264
column headings in Waveform window, 203
column properties, bus/signal, 217
column properties, packet summary, 280
COM Automation Server, 52
COM automation, use of XML format, 352
COM/DCOM, extending capture capabilities, 186
combining logic analyzer cards, 88
commands, keyboard, 489
comma-separated value files, 177, 466
comments, 104
comments, bus/signal, 93, 395
comments, General-Purpose ASCII symbol file format, 481
comments, If clause, displaying/hiding, 161
comments, marker, 234
comments, system in use, 79, 81, 421
communication timeout, floating license server, 314
compare, 53
compare differences, run until a number, 251
Compare menu, 367
compare operation keyboard commands, 489
Compare Overview bar, 250
compare packets, 270
Compare window, 384
Compare window properties, 252
Compare window, new, 368
compared data, finding differences, 250
comparing a range of samples, 251
comparing captured data, 250
compatible modules, 352
complex bus/signal patterns, finding, 246
complex triggers, setting up, 350
concepts, 325
conclusions, understanding logic analyzer triggering, 351
conditions in trigger sequence, 343
configuration file header information, 188
configuration file, ALA format, 466
configuration file, creating new after offline startup, 411
configuration file, mapping modules from, 429
configuration file, new/open/save, 371
configuration file, offline creation of, 399
configuration file, opening, 188
configuration file, opening after offline startup, 411
configuration file, recently-used, 190
configuration file, saving, 176
configuration files, default folders, 412
configuration files, loading/saving, 62
configuration files, new/open/save, 358
configuration files, trigger setup, 172
configuration files, won't open, 305
configuring counters, 140
configuring timers, 139
connect samples analog property, 210
connect samples in chart, 201
connect to logic analysis system, 421
connecting multiple frames, 353
connecting to a LAN, 291
connection attributes, external oscilloscope, 373
connection rules, 389
connections for disabled modules, 48
connections to device under test, 436
connections, deleting/adding, 283
connectivity, 291
connectorless probing, Soft Touch, 73
consecutive files, storing repetitive run data, 186
contact information, 323
content information, configuration file, 188
control (of logic analyzer), remote, 52
convert trigger step to if/then, 160
conveyor belt analogy, 341
copy data to Compare reference buffer, 367
copy to clip board, 359, 371
copying data to reference buffer, 250
copying module/tool/window status to clip board, 430
copying screens to clip board, 291
copying text to clip board, 290
copyright information, 369
correlated source line, showing, 255
Correlation Bus tab, 428
correlation bus, selecting, 256
counter action, 156, 157

counter event, 152, 153, 309
counter event check without increment action, 311
counter in trigger sequence, 309
counters, 138
counters, triggering capability, 342
counters, using, 140
counters, using in triggers, 345
creating new markers, 223
CSV file (module) header format, 467
CSV format, 177, 466
CSV format files, offline analysis in Excel, 195
CSV module file format, 179, 466, 472
custom default storage, 151
customizing toolbars, 375
cut to clip board, 359, 371
cutting trigger sequence steps, 159

D

data and setup, loading, 188
data channel, 538
data formats, 466
data on unassigned channels, transitional timing, 337
data storage, transitional timing, 337
data valid window, 339
data values in waveform, showing/hiding, 209
data, buses/signals to export, 416
data, comparing captured to reference, 250
data, copying to reference buffer, 250
data, exporting to module binary (ALB) files, 181
data, exporting to module CSV files, 179
data, finding bus/signal patterns in, 240, 241
data, finding compare differences, 250
data, loading saved, 188
data, print to file, 290
data, printing, 290, 415
data, range to export, 416
data, reference, offset, 251
data, saving, 176
data, viewing saved, 352
date information, configuration file, 188
Decimal number base, 142, 209, 218
default base, 103
default base (bus/signal), 93
default bit order, assigning channels, 98
default bit order, resetting to, 99, 101
default folders, 412
default number base, 395
default storage, 138, 151, 347, 392, 538
default storage, action to turn on/off, 159
default trigger sequence, 392
defining probes, 436
delay, 538
delay from trigger in waveform display, 199
delay, adjustable, on state sampling clock input, 339
delay, repetitive run, 52, 412
delete Compare window, 367
delete event, 152
delete external scope, 361
Delete Find Event, 246
delete Listing window, 366
delete logic analysis system, 421
delete marker, 364
delete Overview window, 366
delete Source window, 368
delete Waveform window, 366
delete, Overview window, 287
deleting a bus or signal, 95
deleting actions, 156
deleting buses/signals from Listing window, 215
deleting buses/signals from Waveform window, 204
deleting connections, 283
deleting display windows, 283, 289
deleting favorite find patterns, 247
deleting markers, 227
deleting system from use list, 77
deleting tools, 283
deleting trigger sequence steps, 160
demangled symbols, 475
Demo Center, 46, 71
demo data, creating when offline, 412
Demultiplex state sampling clock mode, 110, 111
depth, acquisition memory, 115, 124, 396, 493, 495, 526, 531
description information, configuration file, 188
description, configuration file, 176
details, module/tool/window, 430
details, trigger, 417
details, viewing system, 80
development (tool) API, 53
device under test, 538
device under test connection, 436
device under test, connecting, 57
device under test, controlling signals in, 74
device under test, loading in the, 306
device under test, probing, 73
device under test, remote control for switches, 437
dialogs, 391
difference foreground/background color, 252
difference pair reference data, 252
differences in compared data, finding, 250
differential probes and single-ended signals, 436
differential threshold voltage level, 91
directories, source file, 253, 428
directory list, source file, 255
disabling module, 88, 361
disabling modules, 48
disk space, freeing, 300
display center rectangle Listing window, 216
display line numbers, 258
display scale, changing the, 198
display window properties, editing, 286
display windows, 377
display windows, adding/deleting, 289
display windows, adding/deleting/duplicating, 283
display windows, lock scrolling, 206, 216
display windows, renaming, 287
display windows, using, 289
displaying symbols, 220
displaying the Overview window, 283
documenting trigger sequences, 350
don't care, 538
don't store sample action, 159
Dont Care signal pattern value, 142
double-click, 538
drag and drop, 539
drag and drop markers, 224
draw box triggers, 129
draw rectangle in data, 198
Dual Sample state sampling clock mode, 110
dual sample state sampling clock mode, 51, 110, 112
duplicate Compare window, 367
duplicate Listing window, 366
duplicate Source window, 368
duplicate Waveform window, 366
duplicating display windows, 283
duration counts, 156
duration of storage, transitional timing, 337
dynamic triggers, 186

E

E5850A time-correlation fixture, 52
early return of borrowed floating licenses, 298, 426
edge, 539
edge (multiple) trigger, 53
edge and pattern trigger function, 442
edge followed by edge trigger function, 442
edge followed by pattern trigger function, 443
edge operator, pattern trigger, 142
edge resources, 309
edge trigger, 132, 329
edge trigger function, 441

edge trigger functions, timing mode, 440
edge, triggering on signal, 134
edges on a bus, triggering on, 133
edges too far apart trigger function, 443
edges, center waveform display about, 198
edges, multiple, 68
edges, state clock, too close, 306
edges, triggering capability, 342
edges, triggering on, 345
Edit Application dialog, 404
Edit menu, 359
edit operation keyboard commands, 489
editing window/tool properties, 286
Either Edge signal edge value, 142
either edge, triggering on, 345
elastomeric probe adapter, 73
ELF/DWARF object file format, 475
ELF/stabs object file format, 475
ELF/Stabs symbol files, 121
else if branches, 345, 450, 451, 462, 463, 464
E-mail dialog, 399
e-mail on trigger, 150
e-mail, send on compare differences, 251
e-mail, send on interval marker statistic, 52, 228, 229
e-mail, send when marker value found, 224
enable channel order selection, 99
enable module, 361
enabling modules, 88
end of data, 199
End of Data marker, 288
end of data, go to, 226
end of data, go to in Listing window, 213
end value of symbol, 419
endian, 101
English online help, 369
Entering, find qualifier, 241
equal to operator, pattern trigger, 133, 142
error dialogs, showing/hiding, 412
error messages, 306
error messages in the status log, 430
evaluation order of events, 155
evaluation order of find events, 246
event, 539
event checks without action, 311
Event Editor dialog, 401
event logging, 412
event statements, 139, 140, 141
event statements, reading, 139
event, arm in from, 154
event, counter, 153
event, flag, 154
event, timer, 153
events (advanced trigger), grouping, 52, 155
events and actions, trigger, 138
events in trigger sequence, 392
events, duration or occurrence counts in timing mode, 156
events, inserting or deleting, 152
events, negating, 154
excluding bus/signal transitions, 151
Exiting, find qualifier, 241
expand a bus, 204
expand bus into signal waveforms, 201
experienced users, tips for, 55
export bus/signal data to CSV file, 52, 177
export CSV files, 358
Export dialog, 401
Export file selection dialog, 402
export files, default folders, 412
exporting a range of data, 416
exporting data, 177
exporting data from selected buses/signals, 416
exporting data to module binary (ALB) files, 181
exporting data to module CSV files, 179
expressions, boolean, 344
External Application Setup dialog, 403
external arm, wait for, 448, 460
external instrument, trigger in from, 170
external instrument, triggering, 404
external instruments, trigger out to, 169
External Oscilloscope Setup toolbar, 373
external oscilloscope time correlation, 47, 52
external scope, add/delete, 361
external trigger, 154, 164, 361, 539
External Trigger dialog, 404
extra data channels, 98, 99

F

Falling Edge signal edge value, 142
falling edge, triggering on, 345
fan out/in to tools in Overview window, 389
fast binary data format, 194
fast binary data format files, offline analysis, 195
Fast Zoom In option for Waveform windows, 207
favorite find pattern, 247
favorite packet events, organizing, 149
favorite packet events, saving, 149
favorite trigger, 172, 173, 417
feedthrough mode, external trigger, 404
FFT VbaView window, 47
field decode, 275
field patterns (packet), finding, 243
file errors, 308
file format, 195, 466
file format, module binary (ALB), 181, 473
file format, module CSV, 179, 466, 472
file format, netlist, 100
File menu, 358
file operation keyboard commands, 489
file, print to, 290
file, symbol, 419
files (configuration), opening, 188
files (configuration), recently-used, 190
files (configuration), saving, 176
files (configuration), translating 167xG and 16700-series, 176
files (configuration), won't open, 305
files, source, browsing, 254
files, symbol, 433
fill memory action, 150
fill memory storage qualifiers, 151
filling memory after trigger, 312
filter/colorize tool, 362
filtered data color, Waveform window, 206
find, 359, 362
find a packet trigger function, 458
find anything N times trigger function, state mode, 459
find anything N times trigger function, timing mode, 446
Find Bus/Signal dialog, 201, 214
find data, 371
Find dialog, 406
find events, 246
Find Options dialog, 248
find patterns, favorites, 247
find qualifiers, 241
find symbol name, 419
find, quick, 240
FindEdges macro, 364
finding bus/signal patterns, 240, 241
finding complex bus/signal patterns, 246
finding differences in compared data, 250
fixed unit for time column, Listing window, 219
fixed unit for time column, Protocol Viewer window, 282
flag action, 156, 158
flag action delay, 154
flag event, 152, 154
flag line used for arming in multiframe systems, 353
flags, 138, 345
flags status, 430
flags, using, 141
FLEXlm License Finder dialog, 423

floating license server communication timeout, 314
floating license servers, 296, 425
floating licenses, 47, 293
flying lead probe type, 436
folder (bus/signal), adding, 105
folders, default, 412
font size, Listing window, 215, 216
font size, Protocol Viewer, 279
font size, Source window, 258
font size, waveform, 206
font size, Waveform window, 204
footers when printing, 290
force prestore, 46, 115
foreground color, marker, 223, 233
format, netlist file, 100
formats, data, 466
Found marker placement, 248
FPGA dynamic probe, 50
FPGA dynamic probe, automatic set up, 48
frame, 539
Frame Module Information dialog, 407
frame that module is in, 436
frame, connecting to, 421
frame, slots/cards/modules in, 434
frequency of changes in time interval, 228
front panel operation, 16880-series logic analyzers, 487
full channel timing mode, 107, 493, 495, 526
full screen, 360
full screen display, 371
function names, 118
function tests, 536
FUNCTIONS, General-Purpose ASCII symbol file format, 478

G

general purpose knob, 16800-series logic analyzer, 487
general purpose probe, 50
general purpose probing, 73
general state mode, 108, 526
General-Purpose ASCII (GPA) symbol file, 121, 433, 476
General-Purpose ASCII (GPA) symbol file format, 476
generic text only printer, 290
getting started, 55
glitch, 539
glitch (multiple) trigger, 53, 68
glitch drawing, 133, 134
Glitch signal edge value, 142
glitch, triggering on signal, 134
glitches on a bus, triggering on, 133
global counter status, 430
global counters, 345
global counters, transitional timing, 337
glossary, 537
go offline, 358
go online, 358
go to action, 343
go to beginning/end/trigger, 359, 371
go to here (in markers overview bar), 222
go to locations in displayed packet data, 262
go to locations in Listing window, 213
go to locations in Waveform window, 199
go to marker, 364, 373
Go To Marker dialog, 226
go to trigger on run, 174, 412
goto actions, 150
GPA (General-Purpose ASCII) symbol file, 121, 433
GPA (General-Purpose ASCII) symbol file format, 476
gray bus and signal icons, 93, 95
greater than duration, 531
greater than operator, pattern trigger, 133, 142
greater than or equal to operator, pattern trigger, 133, 142
grid style/color analog property, 210
group events (advanced trigger), 52, 155
group signals into bus, 204
grouping, numeric, 412

H

half channel timing mode, 107, 493, 495, 526
hardware errors, 309
Hardware Update Utility program, 393
hardware upgrades, installing, 124
hardware_log.txt file, 308
hardware, selecting offline, 399
header (packet), viewing, 273
header format in module CSV and module binary files, 467
headers when printing, 290
height, waveform row, 209
help, 56
help file errors, 309
Help menu, 369
help operation keyboard commands, 489
help, Japanese, accessing on Windows XP, 317
Hex number base, 142, 209, 218
hide marker measurement display bar, 373
hide measurements list, 364
hiding markers, 233
hiding source file line numbers, 258
hiding/showing data values in waveforms, 209
High signal pattern value, 142
high-level source code, viewing, 253
history depth, trigger, 172, 173
host offline status, 79
hostname, logic analysis system frame, 77, 421
humidity, 531

I

I/O channel error, 309
IEEE 1394 interface card/cable, 304
IEEE-695 object file format, 475
If clause comments, displaying/hiding, 161
if/then, show trigger step as, 348
import netlist, 100
import netlist to define buses/signals, 411
in range operator, pattern trigger, 133, 142
in use status, 79
incompatible remote service, 304
incompatible remote service version status, 79
increment counter action, 157
independent analyzers, 531
Infiniium oscilloscope time correlation, 52
information messages in the status log, 430
informational dialogs, showing/hiding, 412
informational messages, 312
input only reference data, 252
insert action after, 156
insert action before, 156
insert bus/signal into window, 359
insert event after (AND/OR), 152
insert event before (AND/OR), 152
Insert Find Event After (AND/OR), 246
Insert Find Event Before (AND/OR), 246
inserting buses/signals into Listing window, 215
inserting buses/signals into Waveform window, 204
inserting trigger functions into trigger sequence steps, 142
installation guides, 323
installing a printer, 291
installing on to LAN, 291
Instrument COM Automation Server, 52
interactive self tests, 315
interpreting error messages, 306
intersecting voltage and time markers, 236
interval marker statistics, 52, 228, 229
interval measurement, 376
inverse assembler, 539
inverse assembler development API, 53

inverse assembler tool, 253
inverse assembler tool, Software Address bus, 256
inverse assembler, locked bus/signals, 93
inverse assembly tools, new, 362
IP address, logic analysis system frame, 77, 421

J

Japanese help, 54
Japanese help, accessing on Windows XP, 317
Japanese online help, 369

K

keyboard commands, 489
keyboard, on-screen, 176, 188
knob, 16800-series logic analyzer, 487

L

LAN, connecting to, 291
lanes (packet), viewing, 275
lanes associated with packet, viewing, 275
language options, Windows XP, 317
language, help, 369
Large System Setup utility program, 191
latency, timer reset, 139
left alignment, bus/signal columns, 218
less than duration, 531
less than operator, pattern trigger, 133, 142
less than or equal to operator, pattern trigger, 133, 142
level, triggering on signal, 134
license activation wizard, 295, 424
license file, 124
license manager, 47
license not available, 314
license problems, 314
license server communication timeout, 314
license servers, 293
licensed hardware upgrades, installing, 124
licenses (software), activation, 295, 424
licenses (software), borrow floating, 298, 426
licenses (software), floating license servers, 296, 425
licenses (software), summary, 294, 423
Licensing dialog, 423
licensing host ID, 295, 424
licensing, software, 369
limits in chart, 201
line number symbols, 253, 256
line number symbols, source correlation bus, 428
line numbers in Source window, 427
line numbers, source file, showing/hiding, 258
listing center rectangle, go to source line, 255
Listing menu, 366
listing operation keyboard commands, 489
Listing window, 382
Listing window, analyzing data, 212
Listing window, background color, 216
Listing window, bus/signal column properties, 217
Listing window, bus/signal data color, 217
Listing window, center rectangle, 216
Listing window, column width, 218
Listing window, font size, 216
Listing window, go to locations in, 213
Listing window, inserting/deleting buses/signals, 215
Listing window, lock scrolling, 216
Listing window, marker-relative times, 219
Listing window, new, 368
Listing window, properties, 215
Listing window, Quick Trigger in, 130
Listing window, re-arranging bus/signal columns, 214
Listing window, showing/hiding parts of, 214
Listing window, timing zoom background color, 216
little endian, 101
loading in the device under test, 306
loading saved data and setups, 188
loading/saving user-defined symbols, 53
local area network, 291
Local online mode, 75
local printer, 291
local session, 304
locate bus/signal patterns, 240
lock chart limits to setup, 201
lock display window scrolling, 52, 204, 215
lock in viewer, marker property, 233
lock scrolling, 206, 216
lock to relative marker, 234
locked bus/signal, 93, 95, 306, 395
lockstep, 269
lockstep window, Protocol Viewer, 279
log, status, 430
logic analysis system details, 407
logic analysis system self tests, 315
logic analysis system, adding, 77
logic analysis system, deleting, 78
logic analysis system, going online, 421
logic analysis systems, refresh list, 79
logic analyzer, 539
logic analyzer basics, 56
logic analyzer cards, combining, 88
logic analyzer memory, 341
logic analyzer notes, 492
logic analyzer pod and channel connections, 436
logic analyzer triggering, understanding, 341
Logic Analyzer Upgrade utility, 393
logic analyzer, setting up, 87
logic analyzer, turning on, 56
logic analyzer, what is, 328
logic analyzer, when should I use, 327
logic analyzers, offline analysis on, 194
long marker name, 223, 231
Low signal pattern value, 142
LTSSM States and Transitions, 276

M

macro, 539
macro (Visual Basic), 374
macro, COM/DCOM, 362
macro, pattern generator, 372
mail applications, 399
manage releases, 300
manually mapping modules, 409
mapping modules, 190, 409
mapping modules from configuration file, 429
margins when printing, 290
marker, 540
marker background color, 223, 233
marker comments, 234
marker display bar, 223, 231
marker for system trigger, 288
marker foreground color, 223, 233
marker lock in viewer setting, 233
marker measurement display bar, 376
marker measurements, sample interval, 229
marker measurements, time interval, 228
marker pair, center about, 226
marker position, 223
marker properties, comments, 234
marker properties, lock to relative, 234
marker properties, position, 224
marker properties, snap to edge, 224, 227
marker quick measurements, 63
marker tool tip, 223, 231, 234
marker, Found, 248
marker, go to in Waveform window, 199
marker-relative times in Listing window, 218, 219
marker-relative times, Protocol Viewer, 282
markers, 222

markers display bar, 380
markers display bar in Waveform window, 203
markers display bar, reading, 222
markers display in Listing window, 214
Markers menu, 364
markers overview bar, 381
markers overview bar, reading, 222
Markers toolbar, 373
markers, abbreviated name, 223
markers, creating new, 223
markers, deleting, 227
markers, go to, 226
markers, go to in Listing window, 213
markers, hiding/showing, 233
markers, long name, 223
markers, placing in data, 224
markers, placing where data is found, 248
markers, properties, 232
markers, renaming, 231
markers, sending to back, 231
markers, using, 60
markers, value at measurements, 230
master clock, 109, 110, 111, 112
master frame, 353
Master Only state sampling clock mode, 109, 111
master pods, 414, 429
Master/Slave state sampling clock mode, 109, 111, 114
MatLab Analysis VbaView window, 47
MatLab Connectivity and Analysis package, 47
maximized, starting, 412
maximum 128 channels per bus, 306
maximum channels on a single time base and trigger, 528, 531
maximum global counter, 531
maximum occurrence counter, 531
maximum pattern width, 531
maximum range width, 531
maximum state clock speed, 306
maximum state data rate on each channel, 531
maximum state sampling speed, 526
maximum time count between stored states, 531
maximum transitions stored, 337
maximum trigger sequence steps, 531
measure time, 222
measurement data, printing, 415
measurement examples, 66, 67
memory depth, 115, 124, 396, 526, 531
memory depth and channel count trade-offs, 335
memory depth in full channel mode, 531
memory depth in half channel mode, 531
memory, logic analyzer, 341
menu (Tools), adding external applications, 403
menu access keyboard commands, 489
menu bar, 540
menu, File, 358
menus, 358
message dialog options, 412
messages, borrow licenses, 426
messages, error, 306
messages, return borrowed licenses, 427
MicroBlaze trace core, 46
MicroBlaze version 5, 45
minimum data pulse width, 531
minimum detectable glitch, 134
minimum master-to-slave clock time, 531
minimum PC requirements for offline analysis, 195
minimum slave-to-master clock time, 531
minimum slave-to-slave clock time, 531
minimum state clock pulse width, 531
minimum time between active clock edges, 531
minimum transitions stored, 337
miscellaneous keyboard commands, 489
missing or slow clock, 311
missing pods, 334
module, 540
module binary (ALB) file format, 181, 473
module binary and module CSV file header format, 467
module binary file, 402
module CSV file format, 179, 466, 472
module CSV text file, 402
module details, 430
module mapping, 190
Module Mapping dialog, 409
module setup information, transferring, 352
module setup, mapping from configuration file, 429
module setups, transferring, 190
Module Skew and System Trigger dialog, 410
Module Summary tab, 436
module upgrade, 369
module, cancel processing, 430
module, disable/enable/rename, 361
module, number of cards in, 399
module, resume canceled processing, 430
module, splitting, 88, 429
modules in frame, 434
modules only, loading, 188
modules, configuring, 88
modules, disabling, 48
modules, disabling/enabling, 88
modules, renaming, 287
mouse cursor, placing marker at, 224
mouse shortcuts, 65
moving markers, 224
MSB ordering in bus waveform, 201
multiframe, 353
multiframe cable, 353
multiframe logic analysis system, flags, 141
multiframe remote access passwords, 84
multi-module systems, transferring setups, 190
multiple conditions, testing in trigger sequence, 345
multiple edge/glitch trigger, 53, 68
multiple frames, 353
multiple-modules, setting up, 191

N

N consecutive samples with pattern1 trigger function, 453
N number of edges trigger function, 441
name, symbol, 419
negating events, 154
negative logic, 104
netlist file format, 100
netlist import, 51
Netlist Import dialog, 411
Netlist Import folder, 100
netlist, importing, 52, 100
network, 291
network printer, 291
network troubleshooting, 320
new Compare window, 368
new configuration file, 358, 371
new Listing window, 368
new marker, 364, 373
new sample interval measurement, 364, 373
new Source window, 368
new time interval measurement, 364, 373
new toolbar, creating, 375
new value at measurement, 364, 373
new VbaView window, 368
new Waveform window, 368
next data for source line, 254
next difference, 250
next difference in Compare window, 367
next source line, 253
next window, 368, 371
no hardware found, 124
no matching symbol found, 255
node-locked licenses, 293
not equal to operator, pattern trigger, 133, 142

not in range operator, pattern trigger, 133, 142
not in range, triggering on, 345
Not Present for Range, find qualifier, 241
Not Present, find qualifier, 241
notes, 16800-series logic analyzer, 526
notes, 16962 logic analyzer, 492, 494
notes, logic analyzer, 492
nothing event, 152
number base, 103, 142, 209, 218
number base, default (bus/signal), 93
number base, packet column, 282
numeric data values, showing/hiding, 209
numeric grouping, 412

O

object file formats supported by symbol reader, 475
object file symbols, 118
object file symbols, converting to ASCII, 121
observed value on target control port signals, 437
occurrence, 540
occurrence counter status, 430
occurrence counter, reset action, 156, 157
occurrence counters, 140, 156, 345
Octal number base, 142, 209, 218
offline analysis, 188, 194, 195, 540
offline analysis for 16700-series logic analyzers, 53
offline analysis warnings, 311
offline file formats, 195
offline hardware, selecting, 399
offline mode, 409
offline mode unexpected, 304
offline module setup, 191
Offline Startup Options dialog, 411
offline, creating demo data when, 412
offline, go, 358
offset in Compare window, 367
offset, adding to symbol value, 419
offset, reference data, 251
OMF96 object file format, 475
OMFx86 object file format, 475
online analysis, 75
online help, 369, 371
online, go, 358
online, going after offline startup, 411
on-screen keyboard, 176, 188
open configuration file, 358, 371
opening configuration files, 188, 305
opening the Overview window, 283
option attributes, external oscilloscope, 373
options, 359
options, symbol reader, 121
OR events, 152
or trigger, 164
OR'ed find events, 246
order, channel selection, 99
oscilloscope time correlation, 47, 52
oscilloscope, when should I use, 326
other trigger functions, state mode, 452
other trigger functions, timing mode, 440
Outlook Express, setting up, 400
overlaid waveform color, 205
overlaying waveforms, 200
overview, 56
Overview menu, 366
Overview window, 52, 283, 362, 389
Overview window and multiframe systems, 353
Overview window probes column, 50
Overview window, delete, 287
Overview window, open/display, 283, 368, 371
Overview window, redraw, 287
owner information, configuration file, 188
owner, configuration file, 176

P

packet column, width, 281
packet columns, inserting or deleting, 264
packet columns, re-arranging, 263
packet data, analyzing, 259
packet decode column, alignment, 281
packet decode column, channels (directions), 281
packet decode column, number base, 282
packet details, viewing and comparing, 270
packet event editor, 148
packet event, save as favorite, 149
packet event, view as bits, 149
packet events, organizing favorites, 149
packet events, specifying, 148
packet header, viewing, 273
packet information, viewing, 269
packet lanes, 275
packet lanes, viewing, 275
packet patterns, finding, 243
packet payload, viewing, 274
packet summary, 261
packet summary event colors, 277
packet summary, column properties, 280
packet summary, showing/hiding parts of the display, 268
Packet Viewer window, 46
packet, viewing selected, 269
page headers/footers/margins when printing, 290
parentheses in boolean expressions, 155
partial logic analyzer setup, transferring, 352
passwords for remote access, 84
paste from clip board, 359, 371
pattern, 540
pattern absent for greater than time trigger function, 446
pattern absent for less than time trigger function, 446
pattern and pattern trigger function, state mode, 464
pattern and pattern trigger function, timing mode, 451
pattern generator, 49
pattern generator CSV text file, 402
pattern generator vectors in XML configuration files, 48
pattern N times trigger function, 453
pattern or pattern trigger function, state mode, 465
pattern or pattern trigger function, timing mode, 452
pattern present for greater than time trigger function, 445
pattern present for less than time trigger function, 445
pattern resources, 309
pattern too late after edge trigger function, 444
pattern trigger, 133, 142, 329
pattern trigger function, 444
pattern1 followed by pattern2 before pattern3 trigger function, 455
pattern1 followed by pattern2 trigger function, 454
pattern1 immediately followed by pattern2 trigger function, 454
pattern2 occurring too late after pattern1 trigger function, 458
pattern2 occurring too soon after pattern1 trigger function, 457
patterns trigger functions, state mode, 452
patterns, finding bus/signal, 240, 241
patterns, finding complex bus/signal, 246
pause timer action, 157
payload (packet), viewing, 274
PC requirements, minimum for offline analysis, 195
PCI Express analysis probe, 50
performance improvements, 51
personal computers, offline analysis on, 195
physical view of cards in system, 434
pin mapping details, 100
place marker on screen, 364, 373
place on edge markers, 227

placing markers in data, 224
placing markers with find, 248
pod, 540
pod and channel connections, 395, 436
pod assignment, 361
Pod Assignment dialog, 414
pod index, 540
pod naming conventions, 333
pod pair, 540
pod pair, reserving for time tag storage, 335
pod truncation, 188, 540
pods, missing, 334
point, 541
polarity, 93, 104, 395, 541
polarity, external trigger, 404
position marker property, 224
position, marker, 223
position, trigger, 341
poststore, 115
preprocessor, 73, 541
Present for Range, find qualifier, 241
Present greater than or equal to, find qualifier, 241
Present greater than, find qualifier, 241
Present less than or equal to, find qualifier, 241
Present less than, find qualifier, 241
Present, find qualifier, 241
prestore, 115
previous data for source line, 254
previous difference, 250
previous difference in Compare window, 367
previous source line, 253
previous window, 368, 371
print data, 358, 371
print to file, 290
printer, installing, 291
printing captured data, 290
Printing Data dialog, 415
probe, 541
probe compatibility, 530
probe grounding, 306
Probe Summary tab, 436
probes, 531
Probes toolbar, 373
probing, 73
probing documentation, 323
probing options, 73
problems, solving, 303
procedure names, 118
processing, module/tool/window, canceling or resuming, 430
processor-specific probes, 73
product information web sites, 323
product overviews, 482
programming the logic analyzer, 52
project information, configuration file, 188
project, configuration file, 176
Properties dialog, 415
properties, Compare window, 367
properties, editing window/tool, 286
properties, Listing window, 215, 366
properties, marker, 232, 364, 373
properties, Source window, 257, 368, 427
properties, voltage marker, 239
properties, Waveform window, 204, 366
properties, window, 359
protocol, 541
protocol column channels (directions), 264
Protocol Viewer, 386
Protocol Viewer menu, 368
Protocol Viewer panes, showing/hiding, 277
Protocol Viewer, window properties, 278
pulse clear flag action, 141, 158
pulse set flag action, 141, 158
pulse target control port signals, 437
pulse width in waveform tool tip, 207
pulse width, flag pulse set/clear, 158
pulsed mode, external trigger, 404

Q

QFP package probing, 73
qualifier, state sampling clock, 113
quick find, 240
quick pick bus/signal names, 52
quick start for 16700-series users, 55
quick start guides, 323
quick trigger, 541
Quick Trigger alignment, setting, 258
Quick Trigger in Listing window, 130
Quick Trigger in Source window, 131
Quick Trigger in Waveform window, 129
quick triggers, 127, 129

R

range and offset in Compare window, 367
range of data to export, 177
range of samples, comparing, 251
range operator, pattern trigger, 68, 133, 142
range pattern, 541
Range Properties dialog, 416
ranges, triggering capability, 342
ranges, triggering on, 345
RDP (Remote Desktop Protocol), 322
reader, symbol, 121
readers.ini file, 121
reading markers display and overview bars, 222
readme information, 303
re-arranging bus/signal columns, 214
re-arranging waveforms, 200
re-assign pods to split analyzers, 414
recall recently-used configuration file, 190
recall trigger, 62, 172, 361
Recall Trigger dialog, 417
recalling favorite find patterns, 247
recalling triggers, 392
recent configuration files, 358
recent file list entries, 412
recently used bus/signal names, 142
recently-used configuration file, recalling, 190
recently-used trigger, 172, 173, 417
redraw, Overview window, 287, 366
reference, 357
reference buffer, copy data to, 250, 367
reference buffer, range compare, 251
reference data background color, 252
reference data offset, 251
reference data, comparing to, 250
reference data, display options, 252
refresh logic analysis system, 421
refreshing system to use list, 79
regional and language options, Windows XP, 317
re-group events (advanced trigger), 52, 155
relative marker, lock to, 234
relative time, 218
remote (logic analyzer) programming, 52
remote access, 51
remote access passwords, 84
remote control for switches in DUT, 437
Remote Desktop Protocol (RDP), 322
Remote Desktop, setting up, 322
Remote online mode, 75
remote programming (RPI) commands, 47
remote session, 304
remove bus/signal from window, 359
remove icons to toolbar, 375
remove software, 300
rename bits of a bus, 96
rename Compare window, 367
rename Listing window, 366
rename module, 361
rename Source window, 368
rename trigger, 417
rename Waveform window, 366
renaming a bus or signal, 95
renaming markers, 231
renaming windows/tools/modules, 287
reorder bits, 52, 101
reordering symbols, 118
repetitive measurement, 541
repetitive run, 174, 365, 374
repetitive run delay, 52, 412

repetitive run, go to trigger behavior, 174
repetitive run, stop on compare differences, 251
repetitive run, stop on interval marker statistic, 52, 228, 229
repetitive run, stop when marker value found, 224
repetitive run, storing data to consecutive files, 186
RepetitiveSaveToFile macro, 364
replacement failed, 309
replacing trigger functions in trigger sequence steps, 142
reporting level, self tests, 315
requirements, minimum PC for offline analysis, 195
reset and start timer trigger function, 459
reset counter action, 157
reset occurrence counter action, 140, 156, 157, 309
reset toolbar, 375
resume, 365
resume canceled operation, 374
resume canceled processing, module/tool/window, 430
resume timer action, 157
return borrowed floating licenses early, 298, 426
return borrowed licenses messages, 427
reverse channel order, 101
right alignment, bus/signal columns, 218
right-click, 541
Rising Edge signal edge value, 142
rising edge, triggering on, 345
row height, 209
row properties, analog signal, 209
row properties, bus/signal waveform, 207
RPI (Remote Programming Interface) compatibility package, 47
rules, Overview window connection, 389
run, 541
Run Properties dialog, 418
run repetitive, 541
run status, 174, 430
Run toolbar, 374
run until a number of compare differences, 251
run until user stop trigger function, state mode, 460
run until user stop trigger function, timing mode, 448
run, go to trigger on, 174, 412
run, repetitive, storing data to consecutive files, 186
Run/Stop menu, 365
run/stop operation keyboard commands, 489
running self tests repetitively, 315
running the analyzer, 174

S

sample, 542
sample interval measurement, 229, 376
sample interval measurement, new, 364, 373
sample period, 531, 542
sample period, timing zoom, 117
sample rate in half channel mode, 531
sample rate on all channels, 531
sample, go to in Listing window, 213
sample, go to in Waveform window, 199
sample, position marker by, 223, 224
sampled data, 542
samples, comparing range of, 251
samples, how they are stored in memory, 341
samples, storage qualification, 347
sampling, 542
sampling mode, state, 108
sampling options, 107
sampling options, memory depth and channel count trade-offs, 335
sampling options, state, 108
sampling period, 106, 108, 328
sampling period, timing zoom, 438
sampling period, transitional timing mode, 335
sampling positions, automatic adjustment for 1680/1690-series logic analyzers, 50
sampling positions, state mode, understanding, 339
sampling setup, 361, 372, 396
Sampling tab, 396
save configuration file, 358, 371
saved data and setups, loading, 188
saving captured data, 176
saving configuration files, 176
saving trigger sequences, 350
saving trigger setups, 62
saving/loading user-defined symbols, 53
scale of waveform display, 226
scaling analog property, 210
screen, copy to clip board, 359
screens, copying to clip board, 291
scroll lock on display windows, 52, 204, 215
search, 542
search for symbol name, 419
search operation keyboard commands, 489
search order of source file directories, 428
search, quick, 240
searching data, 64
searching for bus/signal patterns, 240
searching for text in Source window, 255
SECTIONS, General-Purpose ASCII symbol file format, 477
Select Offline Hardware dialog, 399
Select Symbol dialog, 419
Select System to Use dialog, 421
selected bit order, assigning channels, 99
selected packet box color, 279
selected source line, go to data for, 254
self test, 315, 369
send e-mail on trigger, 150
sending markers to the back, 231
SendToExcel macro, 364
SendToExcel VBA macro, 47
SendToMatLab VBA macro, 47
SendToPatternGeneratorModule macro, 364
SendToPatternGeneratorModule VBA macro, 47
separator rows in Waveform window, 46, 204
sequence step branching, transitional timing, 337
sequence step storage, 151, 347
sequence steps, trigger, 309, 343, 344, 345
Serial To Parallel tool, 48
sessions, saving and loading, 352
set flag action, 141, 158
Set Quick Trigger alignment, 258
Setup menu, 361
setup only, loading, 188
setup/hold time, logic analyzer, 306
setup/hold window, logic analyzer, 339
setups, loading saved, 188
short bus/signal names, 412
show axis in chart, 201
show clipped analog property, 210
show correlated source line, 255
show marker measurement display bar, 373
show measurements list, 364
show trigger step as if/then, 160, 348
show values in waveform tool tip, 207
showing markers, 233
showing source file line numbers, 258
showing/hiding data values in waveforms, 209
signal activity envelope, 339
Signal Extractor tool, 46
signal names, 93
signal, grouping into a bus, 204
signal/bus column alignment, 218
signal/bus column properties, 217
signal/bus column width in Listing window, 218

signal/bus data color, Listing window, 217
signal/bus errors, 306
signal/bus folders, adding, 105
signal/bus name, alias, 105
signal/bus names, sorting, 105
signal/bus naming, short or unique, 412
signal/bus patterns, finding, 240, 241
signal/bus patterns, finding complex, 246
signal/bus row properties, 207
signal/bus values displayed as symbols, 220
signal/bus, adding a new, 94
signal/bus, assigning channels in default bit order, 98
signal/bus, assigning channels in selected bit order, 99
signal/bus, deleting a, 95
signal/bus, renaming a, 95
signals/buses, define by importing netlist, 411
signals/buses, exporting data from selected, 416
signals/buses, inserting/deleting in Listing window, 215
signals/buses, inserting/deleting in Waveform window, 204
signals/buses, load warning, 311
Signed Decimal number base, 142, 209, 218
simple compare, 53
simple trigger, 127, 132, 361, 542
simple trigger example, 331
simple trigger in Listing window, 214
simple trigger in Waveform window, 203
simple trigger, setting up, 59
single measurement, 542
single run, 174
single-ended signals and differential probes, 436
skew, 542
skew and system trigger, 361
skew and system trigger, setting, 288
skew between modules, 51
skew, module, 410
slave clock, 109, 110, 111, 114
slave frame, 353
slot, 542
slot naming, 333
Slot Summary tab, 434
slot, starting, 399
slots of cards in module, 436
slow or missing clock, 311
snap to edge markers, 224, 227, 542
Soft Touch connectorless probing, 73
Software Address bus, 256, 428
software licenses, 47
software licenses, activation, 295, 424
software licenses, borrow floating, 298, 426
software licenses, floating license servers, 296, 425
software licenses, summary, 294, 423
software licensing, 369
software, updating, 299
solving problems, 303
sorting bus/signal names, 105
source code correlation bus, selecting, 256
Source Code Directories tab, 428
source code directory list, 255
source code, viewing, 253
source correlation, 428
Source display window, 385
source file line numbers, 118
source files, browsing, 254
source line numbers, showing/hiding, 258
source line symbol values, adjusting, 258
source line, go to data for, 254
source line, showing correlated, 255
SOURCE LINES, General-Purpose ASCII symbol file format, 480
source lines, step by, 253
Source menu, 368
source of data export, 177
Source Properties tab, 427
Source Viewer Properties dialog, 427
Source window, 51
Source window background color, 258
Source window font size, 258
Source window properties, 257
Source window tab width, 258
Source window text color, 258
Source window, new, 368
Source window, Quick Trigger in, 131
Source window, text search, 255
specifications, 528, 536
specifications, 16800-series logic analyzer, 531
Specify Mapping dialog, 429
split analyzer, 88, 361, 429
Split Analyzer Setup dialog, 429
split logic analyzer modules, 51
spreadsheet, importing CSV files, 176, 177, 358
spreadsheet, pasting into from clip board, 290
standard CSV text file, 402
standard threshold voltage level, 91
Standard toolbar, 371
START ADDRESS, General-Purpose ASCII symbol file format, 481
start from reset timer action, 157
start maximized, 412
start value of symbol, 419
startup options, offline, 411
state acquisition mode, 396
state analyzer, 328, 542
state analyzer, triggering, 331
state clock edges too close, 306
state measurement, 542
state mode, 67, 106, 127, 331, 452, 543
state mode sampling positions, understanding, 339
state mode, default storage, 151
state mode, selecting, 108
state sampling clock, 113
state sampling clock mode, Demultiplex, 110, 111
state sampling clock mode, Dual Sample, 110
state sampling clock mode, Master Only, 109, 111
state sampling clock mode, Master/Slave, 109, 111
state sampling clock mode, selecting, 109
state sampling mode, 108
state sampling mode, memory depth and channel count trade-offs, 335
state sampling options, 108
state sampling setup, 372
state speed, 124
statistics, interval marker, 52, 228, 229
status bar, 360
Status dialog, 51, 430
status, system, 365
step to next/previous source lines, 253
steps in trigger sequence, cutting, 159
steps in trigger sequence, deleting, 160
steps, trigger sequence, 138, 309, 343
stop, 543
stop and reset timer action, 157
stop button, 16800-series logic analyzers, 487
stop measurement, 365, 374
stopping self tests on fail, 315
stopping the analyzer, 174
storage control action, 138, 151, 156, 159, 392
storage qualification, 151, 347, 543
storage qualification, triggering capability, 342
storage qualifier when filling memory, 150
store action, 309, 347
store qualification, 531
store qualified (transitional) timing mode, 107, 494, 496, 526
store qualified (transitional) timing mode, default storage, 151
store sample action, 159
store trigger, 172, 361
storing favorite find patterns, 247
storing triggers, 392
strategies for setting up triggers, 348
subdirectories, searching for source files, 428
summary of triggering capabilities, 342

summary, module, 436
summary, system, 434
supported signal types, 530
switch endian, 101
symbol decode, 275
symbol file, ASCII format, 476
symbol file, creating ASCII, 121
symbol information, XML format, 352
symbol lookup, 118
Symbol number base, 142, 209, 218
symbol reader options, changing, 121
symbol reader, object file formats supported, 475
symbol reader, running outside the application, 120
symbol search, 419
symbol selection, 419
symbolic bus/signal values, 123
symbols, 118, 359, 361, 419, 543
symbols (user-defined), saving/loading, 53
Symbols dialog, 433
symbols from object file, 51
symbols with same name, 118
symbols, displaying for bus/signal values, 220
symbols, line number, 253, 256
symbols, loading from file, 119
symbols, user-defined, 118
synchronous sampling, 108, 396, 543
system clock failure, 309
system details, 407
system details, viewing, 80
system in use comments, 79, 81, 421
system options, 412
system summary, 366
System Summary dialog, 51, 434
system to use list, adding frame, 77
system to use list, deleting frame, 78
system trigger, 51, 410
system trigger and skew, 361
system trigger and skew, setting, 288

T

tab width in Source window, 258, 427
tabbed windows, 197, 212, 289, 360
target connector probing, 73
target control port, 74, 361
Target Control Port dialog, 437
target system, 543
temperature, 531
terminating slave frame, 353
testing the logic analyzer, 315
text color, Source window, 258
text data, copying to clip board, 290
text file, print to, 290
text search in Source window, 255
third party tools, 362
tHold, 531
threshold (bus/signal), 93
threshold accuracy, 530
threshold level, trigger in/out, 404
Threshold Settings dialog, 91
threshold value, 306
threshold voltage, 91, 395, 543
TI COFF object file format, 475
Ticoff symbol files, 121
tile windows horizontally, 368, 371
tile windows vertically, 368, 371
time axis in Waveform window, 203
time correlation fixture, 47, 52
time duration in find qualifiers, 241
time interval accuracy, 531
time interval measurement, 228, 364, 373, 376
time measured, transitional timing, 337
time tag resolution, 531
time tag storage, memory requirements, 335
time tag storage, reserve pod pair for, 414
time tag storage, transitional timing, 337
time trigger occurred, 436
time, absolute, 218
time, go to in Listing window, 213
time, go to in Waveform window, 199
time, marker-relative, 219
time, position marker by, 223, 224
time, relative marker, 218
time, relative previous, 218
time/division, 198, 543
timeout, floating license server communication, 314
timer, 543
timer accuracy, 531
timer action, 156, 157
timer event, 152, 153
timer event check without start action, 311
timer in trigger sequence, 309
timer reset latency, 139, 531
timer resolution, 531
timer status, 430
timer value range, 531
timers, 138, 346
timers, number available, 335
timers, triggering capability, 342
timers, using, 139
timing acquisition mode, 396
timing analysis sample rate, 531
timing analyzer, 328, 543
timing analyzer, triggering, 329
Timing Compare VbaView window, 47
timing interval accuracy, 531
timing measurement, 543
timing mode, 66, 106, 127, 328, 544
timing mode sampling period, 108
timing mode trigger functions, 440
timing mode, duration or occurrence counts for events, 156
timing mode, selecting, 106
timing sampling mode, 107
timing sampling options, 107
timing zoom, 116, 526
timing zoom background color, Listing window, 216
timing zoom background color, Waveform window, 206
timing zoom sample period, 117
timing zoom trigger position, 117
timing zoom, aligning with trigger in split analyzer, 117
timing zoom, turning on or off, 117
timing/state sampling setup, 372
timing/state setup, 361
TimingZoom Setup dialog, 438
tips for experienced users, 55
toggle target control port signals, 437
too few states between pattern1 and pattern2 trigger function, 456
too many states between pattern1 and pattern2 trigger function, 456
tool details, 430
tool development API, 53
tool properties, editing, 286
tool tip, 544
tool tip display, Waveform window, 207
tool tip, marker, 223, 231, 234
tool tips, 65, 104
tool tips, Waveform window, 204
tool, cancel processing, 430
tool, resume canceled processing, 430
toolbars, 360, 371
toolbars and mouse shortcuts, 65
toolbars, customizing, 375
Tools menu, 362
Tools menu, adding external applications, 403
tools, adding/deleting, 283
tools, help on, 369
tools, renaming, 287
tools/viewers, loading, 188
topics, help, 369
trace, 544
trace memory, 174
transferring setups to/from multi-module systems, 190
transition width in waveform tool tip, 207
transitional default storage, 151
transitional timing, 107, 337, 544
transitional timing mode, counters in, 140
transitional timing sampling mode, memory depth and channel count trade-offs, 335
Transitioning, find qualifier, 241
trigger, 132, 138, 172, 544
trigger actions, 150, 343, 531
trigger analyzer from another instrument, 170
trigger and fill memory action, 150

trigger and goto action, 150
trigger attributes, external oscilloscope, 373
trigger errors, 309
trigger function, 392, 544
trigger functions, 138, 440
trigger functions, converting to if/then, 160
trigger functions, replacing or inserting, 142
trigger functions, state mode, 452
trigger functions, timing mode, 440
trigger functions, understanding triggering, 348
trigger history, 544
trigger history depth, 172, 173, 412
trigger in, 164, 404
trigger in from external instrument, 170
Trigger In/Out and multiframe systems, 353
trigger inhibited during prestore, 312
trigger other instruments, 169
trigger out, 164, 404
trigger out to external instruments, 169
trigger overview, 46
trigger point, 199, 341
trigger position, 115, 341, 396, 531, 544
trigger position, timing zoom, 117, 438
trigger position, transitional timing, 337
trigger reference point skew, 410
trigger resource conditions, 531
trigger resources, 530, 531
trigger sequence, 392, 544
trigger sequence in U4154A, 506
trigger sequence step branching, 531
trigger sequence step, show as if/then, 160
trigger sequence steps, 138, 309, 343
trigger sequence steps, cutting, 159
trigger sequence steps, deleting, 160
trigger sequence steps, replacing or inserting trigger functions, 142
trigger sequence, clearing, 163
trigger sequences, save and document, 350
trigger setup file, 172
trigger setups, saving/recalling, 62
trigger specification file, 417
trigger specification too complex, 309
trigger step status, 430
trigger, go to, 226
trigger, go to in Listing window, 213
trigger, go to on run, 174, 412
trigger, multiple edge/glitch, 53
trigger, recalling, 172, 417
trigger, send e-mail, and fill memory action, 150
trigger, system, 410
trigger, system, setting, 288
trigger, time trigger occurred, 436
triggering, 132, 138, 172
triggering after external data analysis, 186
triggering capabilities, summary of, 342
triggering the state analyzer, 331
triggering the timing analyzer, 329
triggering, understanding logic analyzer, 341
triggers, dynamic, 186
triggers, quick, 129
triggers, recalling, 172
triggers, storing, 172
triggers, strategies for setting up, 348
troubleshooting, 303
tSample accuracy, manual adjustment, 531
tSample adjustment resolution, 531
tSample range, 531
tSetup, 531
turbo state mode, 108, 526
turbo state mode, flag actions, 141
turn off/on default storage action, 159
tutorial, 56
tutorial configuration file, 60
tWidth, 531
two's complement, 209, 218
two's complement, 142
type, symbol, 419
types, probe, 436
TZ (timing zoom), 116

U

U4154A, 482, 484
U4154A Characteristics, 528, 531
U4154A clock channels, 497
U4154A data channels, 497
U4154A eyescans, 508
U4154A logic analyzer notes, 492, 494
U4154A pod, 497
U4154A Specifications, 528, 531
U4154A state clock qualifiers, 503, 505
U4154A state sampling clock, 503, 505
unassigned channels, data on, transitional timing, 337
understanding logic analyzer triggering, 341
undo, 359, 371
undo trigger sequence step shown as if/then, 160
unique bus/signal names, 412
unsplit analyzer, 361
update software, 299
upgrade, module, 369, 393
upgrades, E5876A, for 16801A/16821A logic analyzer, 526
upgrades, E5877A, for 16802A/16822A logic analyzer, 526
upgrades, E5878A, for 16803A/16823A logic analyzer, 526
upgrades, E5879A, for 16804A logic analyzer, 526
upgrades, E5880A, for 16806A logic analyzer, 526
upgrades, licensed hardware, 124
user interface, 487
user threshold value, 306
USER, General-Purpose ASCII symbol file format, 478
user-defined symbols, 118, 433
user-defined symbols, saving/loading, 53
user-defined threshold voltage level, 91

V

value at marker measurement, 230, 364, 373, 376
value at measurement, 544
value, position marker by, 223, 224
value, symbol, 419
variable names, 118
variable size, 475
VARIABLES, General-Purpose ASCII symbol file format, 480
VBA (Visual Basic for Applications), 49
VBA macros in XML configuration files, 48
VbaView windows in XML configuration files, 48
version information, 369
version information, configuration file, 188
vibration, 531
View As dialog, 201
View menu, 360
view operation keyboard commands, 489
view packet event as bits, 149
viewers/tools, loading, 188
viewing bus data as chart, 201
viewing captured data, 187
viewing tabbed windows, 289
virtual analyzer, 399
Visual Basic Editor, 374
Visual Basic for Applications (VBA), 49
Visual Basic toolbar, 374
voltage interval measurements, 237
voltage marker properties, changing, 239
voltage markers for analog signals, 234
voltage markers, creating new, 235
voltage markers, deleting, 237
voltage markers, placing, 236
voltage markers, renaming, 238
voltage markers, sending to back, 238

Voltage number base for analog signal in Listing, 218
voltage threshold, 530

W

wait for arm from another module, 164
wait for arm from another module trigger function, state mode, 461
wait for arm from another module trigger function, timing mode, 449
wait for external arm, 170
wait for external arm trigger function, state mode, 460
wait for external arm trigger function, timing mode, 448
wait N external clock states trigger function, 461
wait T seconds trigger function, 447
waiting in trigger step, 312
warning dialogs, showing/hiding, 412
warning messages, 311
warning messages in the status log, 430
waveform color, 208
waveform display scale, changing, 198
waveform display, changing the delay, 199
waveform font size, 206
waveform height, 209
Waveform menu, 366
waveform numeric data values, showing/hiding, 209
waveform operation keyboard commands, 489
waveform tool tip display, 207
Waveform window, 377
Waveform window background color, 205
Waveform window, analyzing data, 197
Waveform window, bus/signal row properties, 207
Waveform window, filtered data color, 206
Waveform window, finding a bus/signal row, 201
Waveform window, go to locations in, 199
Waveform window, group signals into bus, 204
Waveform window, inserting/deleting buses/signals, 204
Waveform window, lock scrolling, 206
Waveform window, new, 368
Waveform window, overlaid waveform color, 205
Waveform window, properties, 204
Waveform window, Quick Trigger in, 129
Waveform window, showing/hiding parts of, 203
Waveform window, timing zoom background color, 206
Waveform window, viewing bus data as chart, 201
Waveform windows, Fast Zoom In option, 207
waveforms, accumulate, 206
waveforms, overlaying, 200
waveforms, re-arranging, 200
web control, external oscilloscope, 48
web sites, 323
what's new, 21
what's new, version 01.10, 54
what's new, version 01.20, 53
what's new, version 01.40, 52
what's new, version 02.00, 51
what's new, version 02.50, 50
what's new, version 03.00, 49
what's new, version 03.20, 48
what's new, version 03.30, 47
what's new, version 03.50, 46
what's new, version 03.55, 45
what's new, version 03.60, 44
what's new, version 03.65, 43
what's new, version 03.67, 42
what's new, version 03.70, 41
what's new, version 03.80, 40
what's new, version 03.82, 39
what's new, version 03.83, 38
why are pods missing?, 334
width (bus/signal), 93
width of buses/signals, 395
width violation on pattern or pulse trigger function, 447
width, packet decode column, 281
width, tabs in Source window, 258
window details, 430
window menu, 368
window operation keyboard commands, 489
window properties, 359
window properties, editing, 286
window tabs, turning on/off, 289
window, cancel processing, 430
window, Overview, 52, 283, 389
window, resume canceled processing, 430
windows, 377
windows, adding/deleting, 289
windows, adding/deleting/duplicating, 283
windows, display, using, 289
windows, help on, 369
windows, renaming, 287
writing to target control port signals, 437
www.keysight.com, 31
www.keysight.com/environment/product/, 6

X

XML format configuration file, 176, 188, 190
XML format configuration files, automatic module mapping, 409
XML format configuration files, pattern generator vectors, 48
XML format configuration files, VBA macros, 48
XML format configuration files, VbaView windows, 48
XML format configurations, setting up multiple-modules with, 191
XML format files, offline analysis, 195
XML vs. ALA format, when to use, 352

Z

zoom in on data, 61, 198
zoom out/in, 360, 371
zooming, 545
zooming fast in Waveform windows, 207