

**Tektronix Logic Analyzer
Online Help
Printed Help Document**

Copyright © Tektronix. All rights reserved. Licensed software products are owned by Tektronix or its subsidiaries or suppliers, and are protected by national copyright laws and international treaty provisions.

Tektronix products are covered by U.S. and foreign patents, issued and pending. Information in this publication supersedes that in all previously published material. Specifications and price change privileges reserved.

TEKTRONIX and TEK are registered trademarks of Tektronix, Inc.

D-Max, MagniVu, iView, iVerify, iCapture, iLink, PowerFlex, and TekLink are trademarks of Tektronix, Inc.

076-0113-01

Version 5.6

July 17, 2008

Contacting Tektronix

Tektronix, Inc.
14200 SW Karl Braun Drive
P.O. Box 500
Beaverton, OR 97077
USA

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

- In North America, call 1-800-833-9200.
- Worldwide, visit www.tektronix.com to find contacts in your area.

Warranty 2

Tektronix warrants that this product will be free from defects in materials and workmanship for a period of one (1) year from the date of shipment. If any such product proves defective during this warranty period, Tektronix, at its option, either will repair the defective product without charge for parts and labor, or will provide a replacement in exchange for the defective product. Parts, modules and replacement products used by Tektronix for warranty work may be new or reconditioned to like new performance. All replaced parts, modules and products become the property of Tektronix.

In order to obtain service under this warranty, Customer must notify Tektronix of the defect before the expiration of the warranty period and make suitable arrangements for the performance of service. Customer shall be responsible for packaging and shipping the defective product to the service center designated by Tektronix, with shipping charges prepaid. Tektronix shall pay for the return of the product to Customer if the shipment is to a location within the country in which the Tektronix service center is located. Customer shall be responsible for paying all shipping charges, duties, taxes, and any other charges for products returned to any other locations.

This warranty shall not apply to any defect, failure or damage caused by improper use or improper or inadequate maintenance and care. Tektronix shall not be obligated to furnish service under this warranty a) to repair damage resulting from attempts by personnel other than Tektronix representatives to install, repair or service the product; b) to repair damage resulting from improper use or connection to incompatible equipment; c) to repair any damage or malfunction caused by the use of non-Tektronix supplies; or d) to service a product that has been modified or integrated with other products when the effect of such modification or integration increases the time or difficulty of servicing the product.

THIS WARRANTY IS GIVEN BY TEKTRONIX WITH RESPECT TO THE PRODUCT IN LIEU OF ANY OTHER WARRANTIES, EXPRESS OR IMPLIED. TEKTRONIX AND ITS VENDORS DISCLAIM ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. TEKTRONIX' RESPONSIBILITY TO REPAIR OR REPLACE DEFECTIVE PRODUCTS IS THE SOLE AND EXCLUSIVE REMEDY PROVIDED TO THE CUSTOMER FOR BREACH OF THIS WARRANTY. TEKTRONIX AND ITS VENDORS WILL NOT BE LIABLE FOR ANY INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES IRRESPECTIVE OF WHETHER TEKTRONIX OR THE VENDOR HAS ADVANCE NOTICE OF THE POSSIBILITY OF SUCH DAMAGES.

Warranty 9(b)

Tektronix warrants that the media on which this software product is furnished and the encoding of the programs on the media will be free from defects in materials and workmanship for a period of three (3) months from the date of shipment. If any such medium or encoding proves defective during the warranty period, Tektronix will provide a replacement in exchange for the defective medium. Except as to the media on which this software product is furnished, this software product is provided "as is" without warranty of any kind, either express or implied. Tektronix does not warrant that the functions contained in this software product will meet Customer's requirements or that the operation of the programs will be uninterrupted or error-free.

In order to obtain service under this warranty, Customer must notify Tektronix of the defect before the expiration of the warranty period. If Tektronix is unable to provide a replacement that is free from defects in materials and workmanship within a reasonable time thereafter, Customer may terminate the license for this software product and return this software product and any associated materials for credit or refund.

THIS WARRANTY IS GIVEN BY TEKTRONIX WITH RESPECT TO THE PRODUCT IN LIEU OF ANY OTHER WARRANTIES, EXPRESS OR IMPLIED. TEKTRONIX AND ITS VENDORS DISCLAIM ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. TEKTRONIX' RESPONSIBILITY TO REPLACE DEFECTIVE MEDIA OR REFUND CUSTOMER'S PAYMENT IS THE SOLE AND EXCLUSIVE REMEDY PROVIDED TO THE CUSTOMER FOR BREACH OF THIS WARRANTY. TEKTRONIX AND ITS VENDORS WILL NOT BE LIABLE FOR ANY INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES IRRESPECTIVE OF WHETHER TEKTRONIX OR THE VENDOR HAS ADVANCE NOTICE OF THE POSSIBILITY OF SUCH DAMAGES.

Table of Contents

Welcome

Technical Support	1
About Tektronix Logic Analyzers	1
Customer Feedback	2

Getting Started

Overview of Connection Choices	3
Configuring your TLA7000 without a TL708EX	4
Connect to a Networked Instrument	5
Network Search	6
Network Properties	8
Options of Powering On and Off	9
Examine the Waveform	9
Viewing Other Types of Waveforms	10
Viewing a Magnitude Waveform Sample	11
Examine the Listing Data	11
Searching Selected Data	12

System Setup

LA Setup Window	
LA Setup Window	13
Sampling	14
Synchronous Sampling: Define Single Clock	15
Synchronous Sampling: Define Compound Clock	15
Deskew Panel	16
Group Deskew	17
Signal Mapping Area	18
Group Management Area	22
Signal Sampling Screen: Setting Thresholds	23
Signal Sampling Screen: Adjusting Setup and Hold Values	24
Acquisition Options	25
Sample Storage	27
Merged Module Synchronous Sampling	27
Highlight Signal Names Dialog Box	28
iCapture Analog Feeds	29
Filters	
Filter Definitions Dialog Box	32
Define Filter Dialog Box	33

Load Filter Dialog Box	34
Load Filter Browser Dialog Box	35
Applying Filters	35
Color Options Dialog Box	35
Signal Import and Export	
Signal Import and Export Formats	36
Signal Setup Import Behavior	36
Signal Setup Export Dialog Box	37
Signal Setup Import Dialog Box	38
Sample Signal Setup File	38
Comment Lines	40
Control Lines	41
Information Lines	43
Serial Analyzer Setup Window	
Serial Analyzer Setup Window	43
Status Area	44
Autoset Status	45
Left Side Control Area	45
Channel-Lane Assignment Area	46
External Oscilloscope Setup Window	
Setting Up External Oscilloscope Communication	47
iView Setup Window	47
Troubleshooting External Oscilloscope Communication	48
Signal Routing from the Logic Analyzer	48
Signal Routing to the Oscilloscope	49
System Inter-Probing	51

Module Setup

Loading Saved Setups to an LA module	53
Merging Modules	53
Saving LA Setups	54
Setting Up Analog Multiplexing	55

How to

Acquire	
Acquisition Overview	57
Specifying Multiple Acquisitions	58
Opening a Document or Program after an Acquisition	58
Calibrate	
Calibrating Modules	59
Calibrating Probes	59

Change the Data Window Display	
Developing Custom Color Schemes	60
Comparing Data	
Comparing Data	60
Specifying Stop Conditions for Compare Operations	62
Create New Data Window	
New Histogram Data Page	62
New Histogram Options Page	62
Select Axis Type	63
Axis Type Wizard	64
New Source Data Page	64
New Source Options Page	64
Display Analog Data	
Setting Up Analog Feeds	64
Setting Up System Inter-Probing	65
Routing a Signal from a Logic Analyzer Output	66
Routing a Signal to an Oscilloscope Input	66
Changing Signal Routing	67
Edit	
Edit Protocol Files	68
Editing the Protocol Window Properties	68
Edit Relations Between Protocols	69
Edit Stack Files	69
Editing a Tool on the Tools Menu	70
Removing a Tool from the Tools Menu	70
Editing Symbol Files	70
Export	
Exporting Data from a Protocol Window	71
Exporting Data from a Listing Window	71
Exporting Histogram Data	72
Load Setups and Systems	
Loading a Saved System	72
Loading Portions of a Saved System	73
Loading a Saved Module	73
Adding a Data Source to an Active Listing Window	73
Loading an LA Module with Saved Trigger Settings	74
Adding MagniVu Data to a Listing or Waveform Window	75
Loading Source Code Symbol Information	75
Loading Filters	76
Navigate in Data Windows	
Moving to a Mark (Go To)	76
Modifying the Location of Cursors Using the Overview Markbar	76

Changing the Position of Cursors	77
Print	
Print Dialog Box	77
Save	
Save Dialog Box	78
Saving a System	78
Saving a Module	79
Saving Repetitive Acquisition Data	79
Saving Repetitive Module Setups and Data	80
Saving Repetitive System Setups and Data	80
Search	
Setting the Search Options.....	81
Searching for Specifics in a Listing or Waveform Window.....	81
Searching for Source Data	82
Searching in the Protocol Window.....	83
Set Properties	
Setting the Waveform Style.....	83
Setting the Waveform Limits	84
Determining the Data Source of the Selected Waveform.....	84
Selecting the Radix of Busform Readouts	85
Changing the Zoom Options.....	85
Setting the Default Trigger Tab	86
Changing the Start-Up Default Settings.....	86
Changing the Color of a Graph Mark.....	86
Changing the Data Series Range	87
Changing the Graph Drawing Style	87
Changing the Graph Window Properties	87
PG Run Properties Dialog Box.....	88
Waveform Properties	88
Waveform Window Properties	89
Set Up System	
Selecting the Sample Size: A General Approach	90
Store	
Storing LA Module Setup and Hold Violations	91
Setting up Transitional Storage for the LA Module.....	91
Storage Definition Dialog Box	92
Trigger a Logic Analyzer	
Triggering a Logic Analyzer Using the Trigger Tab Page	92
Adding a State to a Trigger Program	93
Defining If/Then Clauses in a Trigger Program	93
Defining Multiple Transition Recognizers	94
Selecting Signals for an LA trigger Channel Event.....	95

Selecting When to Store Data	95
Positioning the Trigger	95
Triggering an LA Module on a Glitch	96
Triggering an LA Module on a Setup and Hold Violation	96
Triggering an LA Module on Timer Values	97
Triggering an LA Module on Counter Values	97
Triggering an LA Module on an Internal Signal	98
Triggering an LA Module on a Word Event	99
Triggering an LA Module on a Snapshot Event	99
Triggering on Group Data Values	100
Triggering on Group Data Ranges	100
Triggering an LA Module on a Single Channel	101
Triggering an LA Module on a Group	102
Arming other Modules from an LA	103
Triggering other Modules from an LA	103
Troubleshooting LA PowerTrigger Programs	104
Timer Event Target Value Limit	105
Use Data Measurements	
Data Measurement Types and Definitions: Count	105
Data Measurement Types and Definitions: Jitter	107
Data Measurement Types and Definitions: Minimum/Maximum	108
Data Measurement Types and Definitions: Time	109
Using LA Data Measurements	110
Statistic Definitions for Data Measurements	111
Adding Dual Channel Measurements	112
Use Data Filters	
Copying Filters	112
Creating Filters	113
Opening Filters	113
Use the Listing Window	
Adding Reference Data to a Listing Window	113
Adding an Opcode Mark	114
Showing Qualification Gaps in a Listing Window	114
Showing Glitches in a Listing Window	115
Going to a Specific Column	115
Creating Symbol Files	115
Symbol File Header	117
Coloring Data	118
Use the Oscilloscope	
Display Analog Waveforms	
Adding a Waveform	118
Changing Waveform Appearance	119

Selecting the Waveform Color.....	119
Displaying Waveform Readouts.....	120
Changing the Waveform Time Scale (Time/Division).....	120
Changing Waveform Window Properties	120
Controlling Whether a Waveform is Visible.....	121
Selecting the Violation Color.....	122
Making Violations Visible	122
Selecting the Active Data Window Color Scheme.....	122
Selecting the Background Color.....	123
Selecting the Graticule Color.....	123
Making Graticules Visible	124
Changing the Size of a Waveform	124
Jump Over Time Gaps in the Waveform Window.....	124
Setting Waveform Options.....	125
Viewing an LA Waveform in Magnitude Mode	125
Scrolling Between Samples in a Waveform Window.....	126
Set Up the Oscilloscope	
Selecting the DSO Horizontal Settings.....	126
Take Measurements with Your Oscilloscope	
Setting Up Waveform Measurements.....	126
Removing Selected Analog Waveform Measurements	127
Taking Voltage Measurements.....	127
Taking Delta Volts Measurements	128
Taking Time Measurements	129
Taking Delta Time Measurements	129
Taking Logic Analyzer Measurements	131
Trigger an Oscilloscope	
Selecting the Trigger Action and Target for an External Oscilloscope.....	131
Use the Waveform Window	
Adding Reference Data to a Waveform Window.....	132

General Application Information

Running Extended Diagnostics	133
Defining Reference Data.....	133
Customizing the System Tools menu.....	134

Data Window Information

Adding a Data Source to an Active Waveform Window.....	135
Opening a Saved Data Window.....	135
Creating an Overlay Waveform.....	135
Locking Windows	136
Aligning Time of Correlated Data Sources.....	136

Adding a Mark to a Data Window	137
Moving Marks	137
Sending Stacked Marks to the Back	137
Going to a Specific Mark	138
Expand Signals.....	138
Collapse Signals	139
Removing Data from a Listing or Waveform Window	139
Removing Waveforms from an Overlay Waveform	139
Storage.....	140
Storage Selection for an LA Module	140
Storing DSO Data in Memory.....	141
Waveform Cursors.....	141
Time per Division	142

LA Trigger Information

Trigger Tab Page.....	143
Trigger Window	143
LA Triggers	145
System Trigger.....	146
System Trigger Placement	148
Setting the System Trigger Source.....	148
PowerTrigger Overview	149
PowerTrigger Detail.....	149
Data Sample Storage Selections	150
Trigger Position for an LA Module.....	151
Trigger Programming Tips for an LA Module	151
Range Recognizers and Hardware Order	153
Signed Decimal Values	153
Module Arming Using an LA	154
Internal Signaling.....	154
Trigger Action Signal Restrictions	155
Change Detect Dialog Box	157
Select Signal Dialog Box	157
Transition Definition Dialog Box	158
Add Transition Dialog Box.....	159
Word Definition Dialog Box	159
Add Word Dialog Box	160
Detecting Violations.....	161
63-Sample Blocks Mode Details.....	162
Define Glitch Detection.....	163
Clause Definition Dialog Box.....	163
Define Snapshot Dialog Box	165

System Trigger and Reference System Trigger Relationship	165
Group Radix.....	166
LA Trigger Event Table.....	167
LA Trigger Action Table.....	168
LA Trigger Resources.....	169
Force Main Prefill	171
MagniVu.....	172
MagniVu Storage Rate	172
MagniVu Trigger Position	173
Multiple Samples per Clock Trigger Programs	173
Entering a Symbol Plus an Offset in a Trigger	175
Triggering Problems.....	175

Serial Analyzer Trigger Window

Serial Analyzer Trigger Window	177
Trigger Tool Bar	177
Creating a Serial Analyzer Trigger Program	178
Serial Analyzer Trigger Events	178
Defining Packets in Trigger Windows.....	179

LA Trigger Program

Trigger Program Structure for an LA Module.....	181
EasyTrigger Program List	183

LA Listing Window Information

Listing Window	185
Add Column/Waveform Dialog Box	187
Lock Windows Dialog Box.....	187
Export Data Dialog Box	188
TLA Binary File Format.....	189
Export Data Options Dialog Box	190
Compare Definition Dialog Box	190
Repetitive Properties Dialog Box.....	191
Open Dialog Box	193
Define Search Dialog Box	193
Search Definitions Dialog Box.....	195
Time Alignment Dialog Box	195
Go To Dialog Box	195
Mark Opcode Dialog Box	196
About Data Properties.....	196
Column Properties.....	196
Marks Properties.....	197

Listing Window Properties	197
Disassembly Properties	198
Error Message Dialog Box	199
Define Suppression Dialog Box.....	199
Show Samples Dialog Box	199
Suppress Samples Dialog Box	200
Sample Suppression Dialog Box.....	200

Waveform Window Information

Waveform Window	203
Time Gaps	205
Selecting Objects	206
Waveforms	
Overlay Waveform.....	206
Overlay Waveform Properties	208
Setting the Overlay Waveform Offset	209
Individual Waveform Properties	209
Individual Waveform Properties within an Overlay Waveform	211
Multi-Signal Busform LA Waveforms	213
Multi-Signal Magnitude LA Waveforms.....	214
Sample Waveforms	215
Single-Signal LA Waveforms	215
Waveform Labels	216
Waveform Options.....	216

Histogram Window Information

Histogram Window	219
Histogram Window Properties	220
Data Source Properties	221
Ranges Properties.....	222
Histogram Scale Dialog Box	224
Histogram Add Counts Dialog Box	224
Export Histogram Dialog Box	224
Export Histogram Options Dialog Box	224
Histogram Tasks	
Measuring Histogram Data	225
Defining Histogram Ranges	225
Selecting the Histogram Data Source.....	226

Source Window Information

Source Window	227
Moving Through Source Files	229
Scan Listing Mode.....	230
Step Forward and Step Backward Buttons.....	231
Next Mark and Previous Mark Buttons	231
Active Cursor Readout	232
Source Files Edit Dialog Box	233
Source Go To Dialog Box	234
Scan Listing Direction Dialog Box	234
Source Window Search Dialog Box.....	234
Source File Locator Dialog Box	235
Source Window Properties	236
Source Files Properties.....	237
Source Window Data Properties	239
Source Window Marks Properties	240
Source-Relative Cursor Positioning	240
Listing-Relative Cursor Positioning.....	241
Source Window Tasks	
Tracing the Execution of Source Code	241
Getting Out of a Loop in a Source Window	241
Setting a Trigger from Source Window Data.....	242

Packet Structure Definition Language (PSDL)

Packet Structure Description Language	243
Syntax.....	243
Packet Structure Definition	245
Operators	245
Control Statements	247
Loop Statements	248
Sample PSDL Program	249
Type Definitions	
Type Definitions	250
ASN.1 Types	252
PSDL Additions to ASN.1.....	258
PSDL Types	258
Value Definitions.....	261
Length and Value Expressions	261
Tags.....	262
Miscellaneous Commands	262
Reserved Words	
Reserved Words (Ab to Con).....	263

Reserved Words (Cou to Max)	264
Reserved Words (Min to Ta).....	264
Reserved Words (To to Wi).....	265
Lexical Items	265
Protocol Stack and Packet Structures	
Protocol Stack, SPZ Files, Packet Structure Overview	266
Defining a Protocol	268

Protocol Analysis

Protocol Window Information	
Protocol Window	271
Compilation Result Pane.....	272
Frames and Framers	273
Protocol Filter Setup Dialog Box	274
Protocol Properties: About Data Tab	275
Protocol Window Properties Tab.....	275
Protocol Column Properties	275
Protocol Color Properties Tab	276
Protocol Marks Properties Tab	277
Protocol Framing Options Properties Tab.....	277
Protocol Search Dialog Box	278
Export Protocol Data Dialog Box.....	278
New Protocol Display Window	279
Select Bus Page	280
Define Bus Dialog Box.....	280
Stack Definition Pane	281
Select Stack Dialog Box	282
Select Stack Page	282
Generic Framer Example	282
Protocol Window Tasks	
Setting Up the Protocol Window	283
Selecting Stacks	283
Filtering the Protocol Display	284
Viewing Decoded Data in a Protocol Window	284
Protocol Designer Information	
Protocol Setup Window Properties	285
Protocol Relations Properties.....	285
Stack Definition Display Properties	286
Protocol Designer Window	286
Protocol Editor Pane.....	287
Add Protocol Dialog Box	288
Replace Protocol Dialog Box.....	288

Save Protocol or Stack Dialog Box	289
Open Protocol or Stack Dialog Box	289
Delete Protocol or Stack Dialog Box	289
Import Stack Dialog Box	290
Export Stack Dialog Box	290
Protocol Designer Tasks	
Add Protocols	291
Define Relations Between Protocols	291
Go To Protocol Dialog Box	292
Delete Protocols	292
Define Protocol Setup	292
Create New Protocol Files	293
Create New Stack Files	294
Replace Protocol Files	294

Analysis Tools

Deskew Analysis	
Deskew Analysis Window	295
Load Deskew Analysis Setup Dialog Box	298
Deskew Analysis Setup Dialog Box	299
Deskew Analysis Properties Dialog Box	301
Deskew Analysis Setups Dialog Box	302
Apply Deskew Analysis Results Dialog Box	303
Data Measurements	
Data Measurements Overview	304
LA Measurement Setup	305
Data Measurement Statistics	306
Graph Window Information	
Graph Window	307
Graph Window Properties	310
Series Data Properties	310
Series Format Properties	311
Hit Frequency Properties	311
New Graph Data Window	312
New Graph Window Select Series	312
Add Series Dialog Box	313
Graph Window Go To Dialog Box	314
Graph Window Tasks	
Adding a New Series	314
Taking Cursor Measurements	315
Going to a Specific Series	315
Going to a Specific Graph Window Mark	315

Zooming a Graph	315
Viewing Persistence Data	316
iVerify Information	
iVerify Window	316
iVerify Setups Dialog Box	318
Load iVerify Setup Dialog Box	319
iVerify Setup Dialog Box	320
iVerify Properties Dialog Box	322
iVerify Properties - About Data Tab	322
iVerify Properties - iVerify Window Tab	323
iVerify Properties - Eye Coloration Tab	325
iVerify Statistical Analysis Dialog Box	327
iVerify Setup Dialog Box: Analysis Configuration Tab	327
iVerify Setup Dialog Box: Channel Tab	329
iVerify Setup Dialog Box: Clock/Qualifier Tab	330
iVerify Setup Dialog Box: Inter-Probing Tab	331
Protocol Window	
Protocol Designer	

Microprocessor Support

Microprocessor and Bus Support Package Basic Operation	333
Load Support Package	333
Group Definitions for Microprocessor and Bus Support	333
Microprocessor and Bus Support Package Triggering	333
Disassembly Display Formats	334
Saved System Timing Format File	334
Custom Sampling	334
Clocking Problems	335
Incorrect Data	335

Troubleshooting/Diagnostics

Extended Diagnostics Properties	337
Troubleshooting Triggering	338

Reference

Symbol Files	
Symbol Files	341
Symbol Value Width	342
Range Symbol Files	342
Pattern Symbol Files	343
Function Range Symbols	343
Variable Range Symbols	344

Source Range Symbols.....	345
Symbols.....	346
Symbols Dialog Box	346
Load Symbols Options Dialog Box.....	348
Exporting Symbol Tables	350
TLA Symbol File Format Overview.....	350
TLA Symbol File Format - TSF Headers	351
TLA Symbol File Format - TSF Pattern Symbols	352
TLA Symbol File Format - TSF Range Symbols	353
TSF Range Symbols - Color.....	354
TSF Range Symbols - Function.....	356
TSF Range Symbols - Source	356
TSF Range Symbols - Variable.....	359
System Overview	
System Window.....	359
New Listing/Waveform Data window.....	362
New Data Window Name.....	363
Name Data Window Dialog Box.....	363
Load System Options Dialog Box	363
Status Monitor Dialog Box	366
External Signal Selection	367
Internal Signal Selection.....	367
Selection Dialog Box.....	368
MagniVu Data.....	368
System Trigger (Trigger All Modules).....	369
High-Level Language support.....	369
Properties	
Start-Up Properties	369
Power-On Diagnostics Properties.....	370
Self Calibration Properties	371
Mainframe Properties.....	372
Modules Properties	372
Module Specifics Properties	372
Merge Modules Properties	372
Preferences Properties.....	373
System Source Files Properties.....	374
Signals Properties	375
Color Properties.....	377
Software Properties	378
Defaults Properties.....	378
System Options	
Save Color Scheme Dialog Box.....	378

Presets Properties	378
Tools Properties	379
Tool Properties Dialog Box	379
Shortcut keys	
General Purpose Data Window Shortcuts	380
Menu Shortcut Keys	381
Waveform Window Shortcuts	382
Listing Window Shortcuts	383
Source Window Shortcuts	384
Histogram Window Shortcuts	385
Triggering Overview	
DSO Trigger Settings	385
Select Status Bits Type Dialog Box	386
Saved Triggers	386
Theory	
Sampling and Digitizing a Signal	386
Intermodule Interactions and Time Correlation	387
High Speed Timing	388
Listing Data Concepts	389
Waveform Data Concepts	390
LA Versus DSO Module and External Oscilloscope Waveforms	391
Sampling Resolution	391
Signal Resolution and Signal Duration	392
Displaying Waveforms	392
DSO Module	
Autoset (DSO Module)	392
DSO Module Horizontal Settings	393
DSO Module Vertical Input Channel Settings	393
DSO Module Setup	395
DSO Measurement Setup	396
Oscilloscope Waveforms	399
Logic Thresholds	400
Probe Calibration	400
Navigation Tips	

Index

Technical Support

Tektronix welcomes your comments about products and services. Contact Tektronix via mail, telephone, or through the Web site. Click [Contacting Tektronix](#)) for more information.

Tektronix also welcomes your feedback. Click [Customer Feedback \(see page 2\)](#) for suggestions for providing feedback to Tektronix.

Several software enhancements are included to support your TLA product. Refer to the release notes on the TLA Application CD for the latest information. You can also access the release notes by selecting:

Start > All Programs > Tektronix Logic Analyzer > TLA Release Notes

About Tektronix Logic Analyzers

The Tektronix family of logic analyzers consists of the TLA7000 Series Logic Analyzers and the TLA5000B Series Logic Analyzers. The TLA7000 series offers the highest performance for today's demanding applications and consists of portable and benchtop modular mainframes with expansion capability. Instrument modules include logic analyzers and serial analyzers. The TLA5000B offers a selection of standalone logic analyzer instruments that make 500 ps-deep memory timing resolution available to designers of today's mainstream embedded systems. A full line of complementary support products for popular processors and buses is available for the entire TLA family.

NOTE. *Unless otherwise indicated, all references to the TLA5000B Series Logic Analyzers apply to both the TLA5000B and TLA5000 instruments.*

Logic Analyzer Modules

The logic analyzer modules support the iLink tool set, which provide tools to take digital and analog measurements through a single logic analyzer probe, display time-correlated views of digital and analog waveforms, and analyze multi-signal buses and eye diagram validation. Logic analyzer modules also feature MagniVu, an acquisition technology that enables each of the logic analyzer modules to offer 125 ps timing resolution on all signals. MagniVu technology gives you a high-resolution window into the actual signal activity for each channel.

Serial Analyzer Modules

The serial analyzer modules provide tools to analyze serial buses using mid-bus or slot probes. Two different serial analyzer modules are available with different types of probes. Both modules can be used with PCI Express Generation 1 and Generation 2 serial buses.

Customer Feedback

Tektronix values your feedback on our products. To help us serve you better, please send us any suggestions, ideas, or other comments you may have regarding the TLA Series Logic Analyzer.

Direct your feedback to us by email to:

techsupport@tektronix.com

In the subject field, please indicate TLA Customer Feedback.

Or FAX at (503) 627-5695, and include the following information:

- TLA Logic Analyzer
- TLA Application software version
- TLA modules (for example, TLA7BB4)
- Microprocessor or bus support package and version.

For specific information on your TLA Series Logic Analyzer, go to the System menu, select System Properties, and click on the appropriate property page.

The following items are important, but optional:

- Your name
- Your company
- Your mailing address
- Your phone number
- Your FAX number

Enter your suggestion below. Please be as specific as possible.

Please indicate if you would like to be contacted by Tektronix regarding your suggestion or comments.

TIP. To copy the above information to your email text editor, right-click the mouse and click Copy. The above information will be copied to the clipboard where you can paste it into any text editor.

To include screen shots of the user interface, press the Alt key and the Print Screen key simultaneously to copy the image to the clipboard. Paste the image into Microsoft Paint and save the image as a .BMP file. You can then attach the file to your email (depending on the capabilities of your email editor).

Overview of Connection Choices

You can connect to your instrument in three different ways: Remote Host, Remote Desktop, or Local. In addition, you can choose to work offline, without connecting to the instrument, to modify setups or view data files previously saved to your PC.

Connecting as a remote host allows you to control the instrument remotely, and then store the data locally on your PC.

Remote Host mode is best for the following:

- Connecting to an instrument over a LAN (must be using V5.0 software or above).
- Sending data to where the application is running.

NOTE. *When you open the application on the remote computer, the analyzer name is shown in the title bar. A TLA7016 is always connected as remote host because there is no internal computer in a TLA7016.*

Connecting as a remote desktop means that you are running the application on the instrument from your PC and storing the data on the instrument.

Remote Desktop mode is best for the following:

- Connecting site-to-site over a network.
- Troubleshooting from a remote location.
- Viewing data stored on the analyzer.

NOTE. *You can use remote desktop to start the TLA Server, and then connect as remote host.*

Connecting locally means that you are working directly on the analyzer.

Using Local if you want to do the following:

- Run the application and store data on the analyzer.
- Connect a printer to the analyzer.

Related Topics

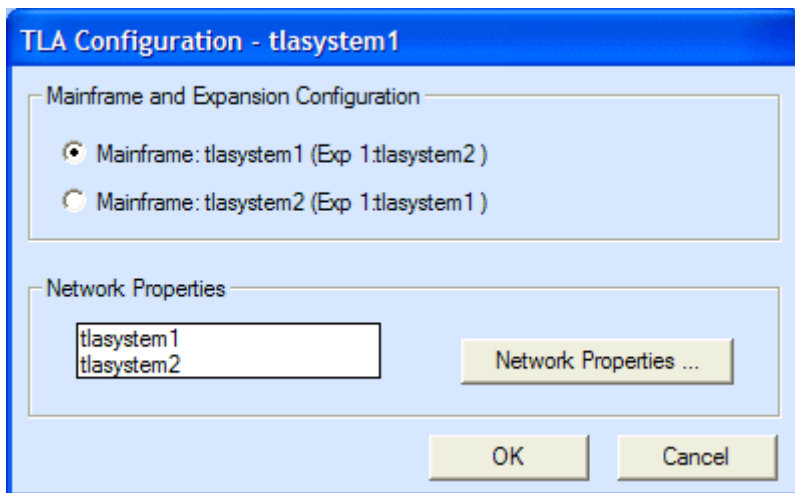
[Connect to a Networked Instrument \(see page 5\)](#)

[Configuring your TLA7000 without a TL708EX \(see page 4\)](#)

Configuring your TLA7000 without a TL708EX

Overview

Use this dialog box to change the configuration of your TLA system when you only have two mainframes connected with a TekLink cable. You can change which TLA is the mainframe and which one is the expansion mainframe using this dialog box. For TLA7016 instruments you can also open the Network Properties dialog box from here.



NOTE. This dialog is only accessible if the system has no more than two TLA7000 mainframes connected with a TekLink cable.

To use the TLA Configuration dialog box, perform the following steps:

1. Select Choose TLA from the File menu to open the TLA Connection dialog box.
2. Select an instrument, and then click the TLA Configuration button.
The TLA Configuration dialog box opens.
3. Select the desired configuration under Mainframe and Expansion Configuration.
4. To edit the network properties, select the TLA7016 instrument, and then click the Network Properties button.

Related Topics

[Connect to a Networked Instrument \(see page 5\)](#)

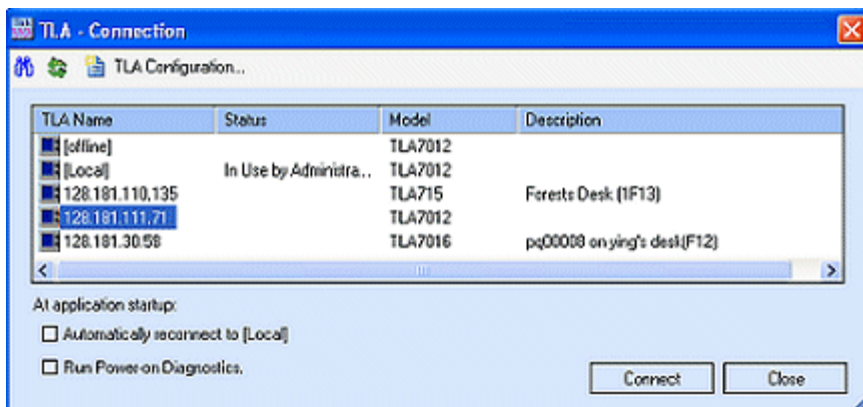
[Network Properties \(see page 8\)](#)

Connect to a Networked Instrument

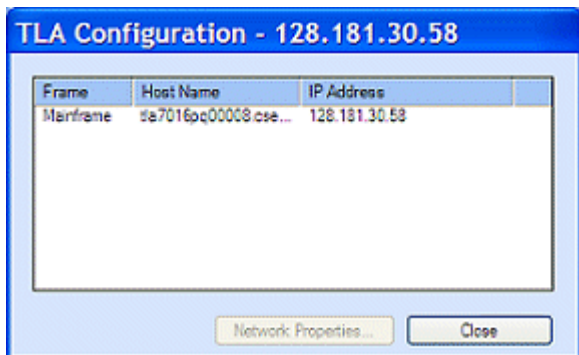
From the TLA Connection dialog box, select the instrument that you want to connect to and click the Connect button. You can customize your startup by specifying whether to run power-on diagnostics and whether to automatically connect to a selected instrument each time you login.


NOTE. *If you choose to automatically reconnect to an instrument, the next time that you start the TLA application, it will open in the offline mode.*

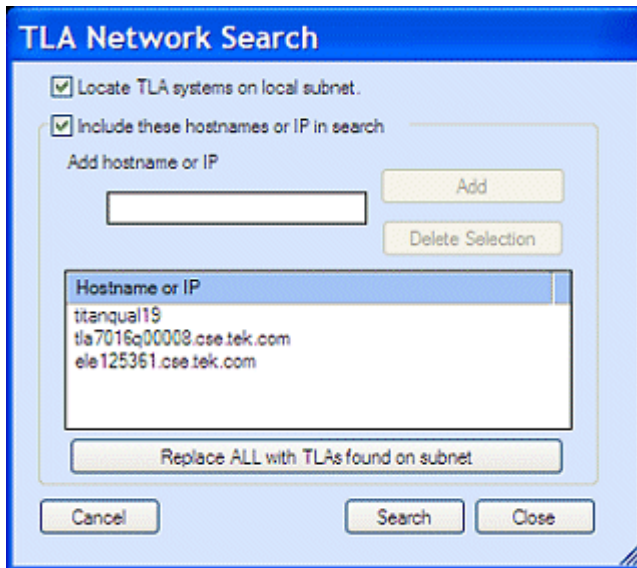
To change your customized settings, select Choose TLA from the File menu. Note that the TLA Server must be running on your instrument to populate the TLA Connection dialog box. To start the TLA Server, click the TLA Server Control icon in the Windows system tray and select Start TLA Server. The TLA Server does not run on your PC.



If you want to view the configuration of your TLA system, select an instrument, and then click the TLA Configuration button. (This button is not active until you have selected a TLA.)



The instruments that are listed in the TLA Connection dialog box are determined by the network search properties. You can change these properties by clicking the  icon and entering your preferences in the Network Search Preferences dialog box.




TIP. If you do not know the host name or IP address, select the *Locate TLA systems on local subnet* check box. This will find any TLA instrument that is connected to the same subnet as your host PC in the search.

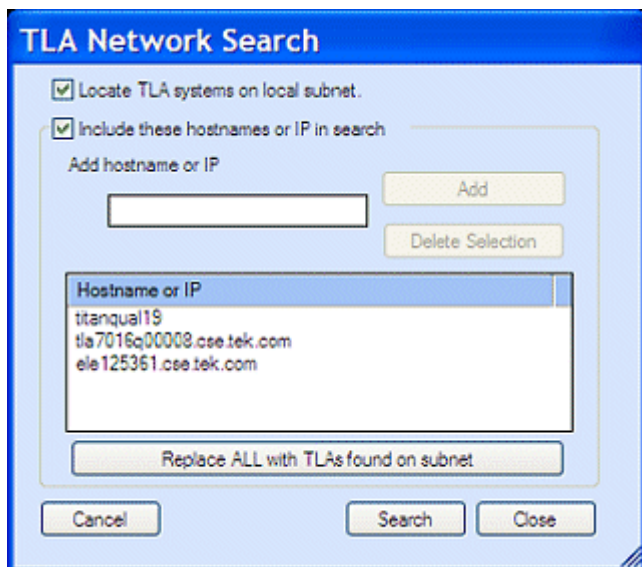
What would you like to do next?

[Learn About the LA Setup Window. \(see page 13\)](#)

[Learn About the LA Trigger Window. \(see page 143\)](#)

Network Search

Use the Network Search dialog box to set the preferences for the search criteria that the Connection dialog box uses. You can open this dialog box by clicking  in the Connection dialog box.



1. Select Locate TLA systems on local subnet to include any TLA system connected to the same subnet as your host PC in the search.
2. Select Include these hostnames or IP in search to include the instruments listed in the Hostname or IP text box in the search. If you know the name of an instrument, you can add it directly by entering the name in the Add hostname or IP text box and then clicking the Add button. This adds the instrument to the search but not to the Connection dialog box. The instrument must be found in the search to be added to the Connection dialog box.
3. To remove an instrument from the search, select it from the Hostname or IP text box, and then click the Delete Selection button.
4. To update the Hostname or IP text box, click Replace All with TLAs found on subnet.
5. Click the Search button to search the subnet according to the criteria that you have set, and update the Connection dialog box. When the search is completed, the Network Search dialog box closes and the Connection dialog box is updated.

Related Topics

[Connect to a Networked Instrument \(see page 5\)](#)

[Overview of Connection Choices \(see page 3\)](#)

Network Properties

Overview

Use the Network Properties dialog box to configure the Host Name and IP address of your networked instruments.

Behavior

This dialog box appears when you click the Network Properties... button in the TLA Configuration dialog box. The Default name is a combination of the TLA7016 Interface Module model number and the serial number. Select Use the following Name to assign a unique name. The TLA7016 Interface Module will attempt to get its IP address automatically when Obtain Address Automatically is selected.

NOTE. This dialog is only supported on TLA7016 mainframes.

The screenshot shows a dialog box titled "TLA Network Properties - TLA7016". At the top, it displays "Product: TLA7016, Serial Number:". Below this, there is a section for "Host Name" with two radio buttons: "Default name (tla7016)" and "Use the following Name". The "Use the following Name" option is selected, and a text box below it contains "TLA7016". To the right of the radio buttons is a "Lookup IP Address" button. Below the Host Name section, there is a paragraph of text: "You can get a network address automatically if your network supports DHCP. Otherwise, you must get an IP address from your network administrator." Below this text are two radio buttons: "Obtain Address Automatically" (selected) and "Use the following Address". Under the "Use the following Address" section, there are three text boxes for "IP Address" (0 . 0 . 0 . 0), "Subnet Mask" (255 . 255 . 0 . 0), and "Gateway" (192 . 168 . 1 . 1). To the right of the IP Address box is a "Lookup Host Name" button. At the bottom of the dialog are "OK" and "Cancel" buttons.

To Use

1. Select either Default name or enter one of your choosing in the Use the following Name text box.
2. Click the Look Up IP Address button to find an IP address.
3. To specify an IP address, select Use the following Address, and then enter the IP Address, Subnet Mask, and Gateway.

Options of Powering On and Off

To power on the analyzer without the TLA application starting, perform the following steps:

1. Access the System folder using the following path: Program Files\TLA\System.
2. Click TLA700SU.bat and use your right mouse button to click Edit.
3. Enter the characters REM in front of the following line:

```
C:\WINDOWS\COMMAND\START /MAX .\TLA700.EXE.
```


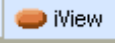
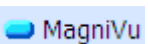
You can set how you want the analyzer to power off through a program labeled Mainframe Utilities located in the Windows Control Panel.




The utility also provides a screen dim mode where you can reduce the brightness by 60%. Refer to the help provided with the utility for more information on both functions.

TIP. Access the Mainframe Utilities by clicking the Windows Start button, pointing to Settings and Control Panel, and then clicking Mainframe Utilities.

Examine the Waveform

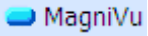
You can use the Waveform window controls to view and measure the waveforms. Here are some suggestions for getting started:


- Use the scroll bars to display data that is currently off-screen.
- Click  in the toolbar and select a mark to jump to.
- Select Cursor 1 and drag the cursor handle to a new location. The handles shown along the top of the display are for fine control; those at the bottom are for coarse control. Note the change in the cursor time readouts in the Cursor toolbar.
- Click the  button to display an external oscilloscope waveform.
- Click the  button to display a MagniVu waveform.

- To zoom data, click and drag the mouse over the area that you want to zoom. The window will zoom to the data when you release the mouse button.
- To increase the size of the area in which the data is displayed, select Full Screen from the View menu.
- Click  (Zoom Out) or  (Zoom In) in the toolbar to change the horizontal scale.
- Select a waveform header and the whole row is highlighted, making it easier to view.
- Double-click  (or any other waveform label). You can change the color and size (size is also changeable directly) of the waveform in this property page. Click OK to complete the changes and return to the Waveform window.
- Click the + on the waveform label to expand the waveform into the individual signal elements.
- Drag the split box (above the top arrow of the vertical scroll bar) down to display a second data area. You can now view two different portions of the same acquisition memory.

TIP. If you are using a portable mainframe, use the front panel knobs to change the horizontal scale and to scroll the data.


Viewing Other Types of Waveforms

Add the MagniVu version of the data at any time by clicking the  button or use the following steps. ([MagniVu Data \(see page 368\)](#) is always acquired with your other data.)

1. Click  (Add Waveform) in the toolbar.
2. From the Data Source list, select LA1 - MagniVu.
3. Select a MagniVu waveform or group (for example, Mag_A3), and click Add.
4. Click the Close button when you have finished adding waveforms.

When viewing LA groups, you can view the data as busforms or as magnitude waveforms by right-clicking the mouse over a data label and selecting either Display Busform or Display Magnitude. You can also expand the group into the individual busforms by clicking the + on the label.

Or,


1. Double-click the label of a signal waveform. (If you do not have a group waveform, click , select an LA Data Source, and choose a group to display. Confirm and exit the dialog box.)
2. From the Waveform Properties tab, click the Options button.
3. Select Busform or Magnitude, and click OK to close the Waveform Options dialog box.
4. Click OK. The waveform is displayed accordingly.

Related Topics

[Multi-Signal Magnitude LA Waveforms \(see page 214\)](#)


[Multi-Signal Busform LA Waveforms \(see page 213\)](#)


Viewing a Magnitude Waveform Sample

1. Click  in the toolbar to start the New Data Window wizard.
2. Click Waveform and click the Next button.
3. Click Add Data Source.
4. Enter the following path into the File name box, or click browse to navigate to the SampleSystem.tla file:
C:\Program Files\TLA700\Samples\TLA700 Samples\Data Files\SampleSystem.tla
5. Click the Add button. The SampleSystem data source is added to the list of data sources.
6. Click LA under SampleSystem.tla and click the Next button.
7. Enter a name for the new data window or use the default name and then click the Finish button.
8. Choose one of the following procedures:
 - a. Right-click the mouse over any data label and select Display Magnitude.Or
 - a. Double-click the new Down Count label.
 - b. Click Options and select Magnitude, click OK twice to close the dialog boxes.
 - c. Click the Zoom buttons to view a sawtooth waveform. The waveform is a graphic representation of counter data.



Examine the Listing Data

Here are some suggestions for viewing Listing window data:

1. Use the scroll bars to display data that is currently off-screen.
2. Click  in the toolbar and select Cursor 1.
3. Drag the cursor handle to a new location. The handles shown along the left of the display are for fine control; those at the right are for coarse control. Note the change in the cursor sample number and delta time readouts in the Cursor toolbar.
4. Click the toolbar Font buttons to change the display font size.

5. Drag the split box (beside the left arrow of the horizontal scroll bar) over to display a second data area. You can now view two different portions of the same acquisition memory.
6. Double-click an address or data column label (for example, ). You can change the attributes of the column in this property page.
7. From the Column property page, try loading a symbol file: click the radix button box, choose the Symbolic radix, click the Symbol File button, and choose a symbol file.
8. Confirm and exit the property page.

Searching Selected Data

1. Double-click a data entry in one of the columns of the Listing window. The data is now selected. Click  in the toolbar to copy the data onto the clipboard.
2. Click  in the toolbar.
3. From the Search Definition drop-down list, select a saved search definition from the selected data window or type in a new search name.

NOTE. To add a search definition from another data source, click the Load button to open the Load Search dialog box.

4. Click the Start from drop-down list to select where you want to start searching and enter the number of occurrences to find in the occurrence text box.
5. Click the Search Options... button to open the Search Options dialog box and select what action you want the logic analyzer to take when the search is successful.
6. Type a Description of the search in the Description text box.
7. From the Search drop-down list, select the data source.
8. Paste the data that you copied from the Listing window into the appropriate text box in the Find area by pressing CTRL+V (the paste shortcut key combination). You can use the same method for pasting a quick entry into a trigger program definition.
9. Use the Group drop-down list to select the column from which you copied the data.
10. Click Search Forward. The logic analyzer displays the next instance of the specified data. Once you have defined the search criteria, you can click the Search Forward or Backward toolbar buttons for additional searches.

LA Setup Window

The primary function of the Setup window is to configure the LA module for compatibility with the system under test. Use this window to do the following tasks:

- Set thresholds
- Set global setup and hold sampling
- Specify signal groups
- Name signals
- Select filter and analysis tools

To Use

1. From the TLA toolbar, click the Setup button, and then select your module, (for example LA 1, when there is more than one LA module).
2. Specify the type of [sampling \(see page 14\)](#).
3. Set the [memory depth \(see page 27\)](#).
4. Specify the [storage option \(see page 25\)](#).
5. Associate meaningful [signal names \(see page 18\)](#) to probe channels.
6. Create meaningful [groups \(see page 22\)](#).
7. [Deskew the probes \(see page 16\)](#).

NOTE. *After defining the LA Setup window, you can immediately acquire data. To improve your acquisition results, revise your Trigger window and adjust your data window properties.*

Behavior

The LA Setup window and the associated Trigger window determines what data will be acquired. Each module has a Setup window and Trigger window; they are set up individually and used separately for their particular purposes. Configure the Setup window before the Trigger window; many of the Setup window settings affect Trigger window selections.

TIP. *To duplicate the current setup for another LA module, open to the File menu and click Save Module.*

For quick access to the Setup window on the TLA7012 Portable Mainframe, press the Setup button on the front panel. Continue to press the Setup button to cycle through all the acquisition module Setup windows in the system.

Base Signal

A memory storage location for acquisition data.

Prime Signal

An expanded memory location for high speed data acquisition. The logic analyzer creates prime signals for storage when the speed of acquisition exceeds that available for base signals alone. Prime signals have an exponent added to the base name.

Creating a prime signal increases the memory available and removes the base signal from the Signal Mapping area.

Sampling

The logic analyzer supports two basic types of sampling: asynchronous and synchronous, each displayed in its own tab on the sampling screen. A third type of sampling, [custom sampling \(see page 334\)](#) is available only when a [support package](#) is loaded on the logic analyzer. The tab label includes the name of the support package.

When you use asynchronous sampling, the logic analyzer generates its own clock signal at regular, fixed intervals to sample data from the system under test. Acquisitions taken with asynchronous sampling are often referred to as timing acquisitions, and are best viewed in the Waveform window.

When you use synchronous sampling, a clock from the system under test samples the signal. This clock can be a fixed frequency, or it may be highly variable. Acquisitions taken with synchronous sampling are often referred to as state acquisitions and are best viewed in the Listing window.

To Use

1. From the TLA toolbar, click the Setup button, and then select your module, (for example LA 1).
2. Click the Asynchronous, Synchronous, or support package tab.
3. Adjust the controls as needed for your situation.

NOTE. *Sampling and state speeds will vary by module.*

Microprocessor or Bus Support Package

A microprocessor or bus support package is a software application that allows the logic analyzer to acquire and disassemble microprocessor data or data from a bus.

Some support packages include a hardware probe adapter that is attached directly to the microprocessor in the system under test to capture microprocessor signals.

The LA Setup window shows which support package is currently installed.

Synchronous Sampling: Define Single Clock

The default controls for the Synchronous tab establish global settings for setup and hold across all signals in the module. You can identify a single clock from the system under test for your synchronous clock. You can change the settings for individual signals in the [Signal Mapping area \(see page 18\)](#).

To Use

1. From the TLA toolbar, click the Setup button for your module.
2. Click the Synchronous tab.
3. If necessary, click the drawer arrow (<|||) to open the Define Single Clock panel.
4. Select the clock source from the Single Clock Source list.
5. Adjust the Global Default Threshold. This setting should match most of the signals that you are sampling with the current module.
6. Enter a maximum clock rate or the minimum clock period for your module.

Setting this value helps scale the T_s and T_h elements in the diagram. Enter a value that is likely to be greater than your maximum clock rate; an accurate value is not needed.

7. Select the clock edge and adjust the busform to match your SUT setup and hold times. You can drag the whole busform or just the edges.

Behavior

When you select the DDR check box, the logic analyzer automatically selects the appropriate sampling points based on the Max Clock Rate and currently selected sample points. If you manually select two points, the DDR check box is automatically selected.

Synchronous Sampling: Define Compound Clock

Use the Define Compound Clock controls to set up a compound synchronous clock. This is a global setting for the module. You can change the settings for individual signals in the [Signal Mapping area \(see page 18\)](#).

To Use

1. From the TLA toolbar, click the Setup button for your module.
2. Click the Synchronous tab.
3. If necessary, click on the drawer arrow (|||>) to open the Define Compound Clock panel.

4. In the Master Sampler table, define the synchronous clock sources by selecting the clocks and qualifiers from the drop-down list within each table cell. The graphic shows the resultant settings.
5. Adjust the Default Threshold. This setting should match most of the signals that you are sampling with the current module.
6. In the Speed drop-down list box, select a bandwidth for the module you are using. If you select Max, and reload a different module, the default setting uses the maximum speed of the module (only available with TLA7000 series logic analyzer modules).
7. Set the Pipeline Delay for all acquired signals.
8. Enter the minimum clock period for your module.

Behavior

When you display the Define Compound Clock panel, the logic analyzer uses the default values you assigned in the Define Single Clock panel, including Min Clock Period, T_s and T_h , and the number of sample points.

If you defined a DDR sampling structure in the Define Single Clock panel, the Sampler Definition area indicates two sample points corresponding to the DDR definition. However, you will note that the Compound Clock diagram differs significantly from the Single Clock diagram.

NOTE. To configure individual channels with varying numbers of sample points, use Compound Clocking and then adjust each signal definition using the [Signal Sampling Screen](#) (see page 24).

Deskew Panel

Deskew allows you to compensate for propagation delays in your circuit traces that differ from your design due to the location of the probe header. You can perform the deskew on all signals in the module by clicking the Deskew button. You can also [deskew individual signal groups](#) (see page 17).


This procedure applies deskew to all signals in the module.

1. From the TLA toolbar, click the Setup button for your module.
2. Click the Synchronous tab.
3. Use the Search Range and Accuracy controls to adjust deskew settings for all signals for the connected modules.
4. Click the Deskew button.

The logic analyzer displays the Deskew progress bar in the Status area during deskew.

Group Deskew

This procedure deskews selected signals within a defined group based on parameters you set in the Deskew panel. After you make any changes to selected signals in the group, the Global Values check box will be cleared. This indicates that these signals will not track changes to the parameters in the Global Sampling diagram.

1. From the TLA toolbar, click the Setup button for your module.
2. Click the Synchronous tab.
3. In the Group Management area, adjacent to the Signal Mapping area, [create and name a signal group \(see page 22\)](#).
4. In the group card, click the Local Sampler Definition thumbnail  in the group card header to open the Signal Sampling screen.
5. In the Signal Sampling screen, verify that the signals that you want to deskew are highlighted in the Signal tree.
6. Click the Deskew button to deskew the highlighted signals in the group.

The logic analyzer displays the Deskew progress bar in the Status area during deskew.

To return signals to the global value, select the Use Global Values check box.

Behavior

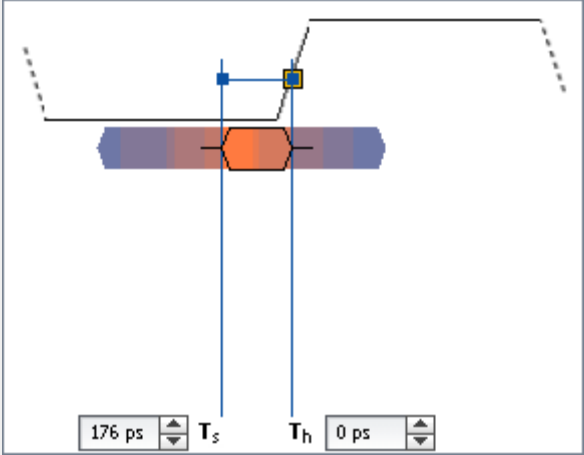
Before deskew, the logic analyzer displays all of the signals in the local Setup and Hold diagram on top of one another. After the deskew, the logic analyzer displays the signals as a [fuzz band](#) in the Setup and Hold diagram.

Related Topic

[Signal Sampling Screen: Adjusting Setup and Hold Values \(see page 24\)](#)

Fuzz Band

The logic analyzer displays signals as a fuzz band within the Setup and Hold diagram. The display shows the signal frequency as a temperature gradient where the warmer or brighter colors represent a higher density of signals. The cooler or darker colors represent areas where fewer signals overlap. Click an area in the fuzz band to select a signal from the tree. You can adjust the Setup and Hold values for the selected signal or move the signal within the fuzz band.



The diagram illustrates a 'Fuzz Band' visualization. It shows a horizontal bar with a color gradient from dark blue on the left to bright orange in the center, representing signal density. Above the bar, a signal trace is shown with a blue square and a yellow square. Below the bar, two vertical lines represent the Setup (T_s) and Hold (T_h) times. At the bottom, there are two input fields: '176 ps' for T_s and '0 ps' for T_h , each with a small up/down arrow icon.

Signal Mapping Area

The Signal Mapping area allows you to view the physical resources of the module. It also provides a centralized area for viewing signal activity, signal type and characteristics, sampling configuration, and [base](#) and [prime](#) signal participation.

Use the Signal Sampling screen to configure the threshold, polarity, type, and the sampling definitions for individual signals. Access the Signal Sampling screen either through the right-click menu (by clicking the activity indicator in a channel cell for the signal), or by clicking the Local Sampler Definition thumbnail



next to the probe section header

Behavior







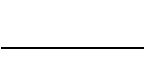
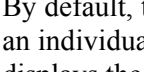
The Signal Mapping area is a collection of cells organized into a two-dimensional matrix. The columns represent the probe leads (consisting of probe sections in a 7 – 0 order, plus the clock and qualification leads). The rows represent probe sections, with adjacent rows representing paired probe sections. Columns and rows cannot be individually sized, and header cells cannot be renamed.

The Signal Mapping area may display logical signals that have no physical counterpart (such as status bits or static bits).

Sampling Definition Thumbnails. When a module is sampling in synchronous sampling mode or custom sampling mode, the Thumbnail column displays images representing the sampling definition of the probe signals. In addition to identifying the characteristics of the probe section sampling characteristics, the thumbnail is a button used to display the Signal Sampling screen.

NOTE. *The logic analyzer does not display thumbnails in the asynchronous sampling mode.*

Click the thumbnail image to open the Signal Properties screen where you can set the sampling offsets for [base](#) and [prime](#) signals.

Thumbnail type	Description
	Basic sampling, rising or falling edge, single sample point
	Compound sampling, one or more rising or falling edges, no qualifier
	Same as above with additional sample point
	Basic sampling, rising or falling edges, two sample points
	Compound sampling, one or more rising and falling edges, no qualifier
	Compound sampling, one or more edges with one or more qualifiers.
	Compound sampling, one or more rising and falling edges, one or more qualifiers
	Varies (variable sampling), signals in a group or section have different sampling characteristics The clocking figure may have any of the images shown above; the rising edge shown for illustration purposes only

By default, the displayed thumbnail corresponds to the Global Sampling Definition. When you change an individual signal sample point (either manually or as a result of the deskew function), the thumbnail displays the Varies image, indicating that at least one signal in the section has a different sample point.

Thumbnails appear for base signals and group cards and are not available for prime signals, status bits, or custom sampling contexts.

Probe Section Header

The logic analyzer uses the Probe Section Header to identify [base](#) (true sources) and [prime](#) (cloned data streams from the base source) signals.

The Probe Section Header displays the name of the probe section (A0, A1, A2). Base probe sections (true sources) display the name left-justified in the header cell.

Prime probe sections (data streams cloned from the base source) are labeled with the base name and an exponent (A0¹, A0²). The labels are right justified in the header cell. Prime probe sections are always placed sequentially below their Base probe section (A0 is followed by A0¹, A0²).

Signal Column Header


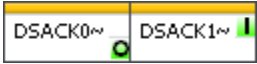


Each probe lead is assigned a cell below its signal number. The Signal column header is a display item only and has no interactive behavior.

Clock / Qual Header

Two columns identify the clock and qualification lead for each probe section that has one of these additional leads. The left column permanently displays the resource name, for example CK0(), and the right column contains the Channel / Signal cell. The Channel / Signal cell initially shows the resource name, but you can enter your own name. For probe sections that do not have a clock, qual, or both, both cells in the column are disabled.

Group Participation Bar


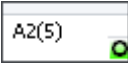
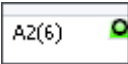
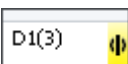
Running horizontally across the top of each Signal cell, is a color-pattern bar that indicates how signals participate in a selected group or multiple groups. The following table shows examples of two adjacent cells.

Appearance	Description
	Default appearance. The channels do not participate in any group.
	Group participation. The signals participate in the currently selected group.
	Multiple group participation. The signals participate in the current selected group and at least one other group.
	Other group participation. The signals participate in a group other than the selected group.

Activity Indicator

The rightmost region of a base Channel / Signal cell displays activity information and a means to modify the operating characteristics of the signal. Instead of activity indicators, polarity indicators are displayed for prime signals, status bits, static bits, or custom signals using the crossbar (custom storage signals). Although you cannot change most of the operating characteristics of these types of signals, you can still change the polarity by right-clicking the polarity indicator and by selecting the Invert Signal Polarity from the right-click menu.

The following table shows examples of the polarity indicators and activity indicators.

Appearance	Description
	Normal, high polarity. The indicator appears as a vertical bar in the upper cell.
	Normal, low polarity. The indicator appears as a zero in the lower cell.
	Inverted, low polarity. The indicator appears as a zero in the upper corner. Similarly for the inverted, high polarity, the indicator appears as a vertical bar in the lower cell.
	Transitioning signal.

Click the green and yellow activity indicator to display the Signal Sampling panel, where you set the signal threshold value and polarity.

Static and Status Bits

These are additional signals that you can use in group definitions for triggering.

Related Topic

[Group Management Area \(see page 22\)](#)

Group Management Area

Use the Group Management area to organize the probe signals to match the configuration of the system under test. Depending on your application, match the [signal groups \(see page 28\)](#) to the address and data buses, or match the signal groups to other signals of interest. Each signal group is displayed as an individual group card with a group name and grid. Group cards contain additional elements, such as an activity indicator and thumbnail to help you define signal properties within the signal group.

To Use

1. From the TLA toolbar, click the Setup button for your module.
2. To create a new group, click the + icon at the top of the Group Management area.
 - a. Enter a [group name](#) for easy identification.
 - b. In the Signal Mapping area, select individual signals or entire probe sections, and then drag and drop the signals to the group card.
 - c. To rearrange the order of signals within a group, drag and drop them from one location to another within the group card grid. Alternatively, right-click the mouse and select Reverse Signal Order to change the MSB-LSB order of the signals.
3. After creating multiple group cards, you can rearrange them by dragging and dropping them in the Group Management area.
4. Click the thumbnail or the activity indicator in a group card to access the [Signal Sampling screen \(see page 24\)](#) to adjust signal thresholds or to adjust the setup and hold requirements for signals.

NOTE. *Activity indicators and thumbnails might not display depending on the sampling you have defined.*

5. Expand and collapse group cards to make room for more cards on the screen using the right-facing arrow in the upper left corner of the group card.

Behavior

You can create any number of group cards. Each group can contain any combination of module signals; you are not restricted from mixing or repeating signals from various groups.

Groups are used in other displays and setup controls, such as the Waveform and Listing windows, and the Trigger window. The order of the groups in this table determines the presentation order of groups in other parts of the application.

Support Packages

If a microprocessor support package is loaded, the signal groups are already defined. You should not change or delete these groups; doing so can lower the accuracy of disassembly analysis. However, you still have the option of adding and deleting new groups.

Related Topic

[Signal Mapping Area \(see page 18\)](#)

Group Name

Each group must have a unique name. When you use the same name as a previous group, the application prefixes your name with a number. For example, if you try to name a group, MyGroup, and another group already exists with the that name, the application will redisplay your group name as 1:MyGroup.

You can use the default name, or you can enter another name as long as it is unique. The names will appear in the Listing and Waveform windows. There is no limit to the number of groups you can create.

Signal Sampling Screen: Setting Thresholds

Use the Signal Sampling screen to set the probe signal characteristics appropriate to your system under test.

To Use

1. From the TLA toolbar click the Setup button for your module.
2. In the Signal Mapping area, select one or more cells.
3. Click the Activity Indicator in one of the selected cells. Alternatively, from the right-click menu, select Signal Sampling.
4. Confirm the signals you want to work with are highlighted in the Signal tree. Clicking an item removes the existing highlighting. (Press and hold CTRL+Click or SHIFT+Click to highlight multiple items in the Signal tree.)
5. Use the slider or Preset controls to set threshold values for the highlighted items.
6. Click Normal or Inverse to set the polarity for the highlighted items.

NOTE. If you highlight a *prime* signal in the Signal tree, no corresponding threshold control becomes available (no slider highlights in the threshold graph). Prime signals do not have thresholds.

Behavior

Changes you make to the threshold are immediately applied to the signals that you [highlighted](#) in the Signal tree.

NOTE. *You can highlight a different set of signals from the "selected" set of signals (the signals you selected in the Signal Mapping area).*

Use one of the buttons adjacent to threshold graph to assign the threshold value to more signals than the ones you've highlighted. For example, if you have highlighted only one signal in the tree, (and you have several "selected") click the Selected Signals button to assign the threshold to the selected set of signals. Similarly, click the Probe Sections button to assign the threshold to all signals in a probe section to which a highlighted signals belongs. The buttons are arranged top to bottom in increasing scope.

NOTE. *Not all buttons will be enabled for all types of highlighting and selecting. Depending on your module type, you might have fewer or more buttons available.*


Related Topic

[Signal Sampling Screen: Adjusting Setup and Hold Values \(see page 24\)](#)

Signal Sampling Screen: Adjusting Setup and Hold Values

Use the Signal Sampling screen to adjust setup and hold times for signals at the local (probe section or group) level when working with synchronous or custom sampling.

To Use

1. From the TLA toolbar, click the Setup button for your module.
2. Select the Synchronous tab (or custom sampling if you want to adjust signals for a loaded support package).
3. Click the Local Sampler Definition thumbnail  in the probe section column in the Signal Mapping area or in the group card to open the Signal Sampling screen (the image on the thumbnail differs depending on your setup).
4. Click to select a signal in the tree (CTRL or SHIFT to select multiple signals) in the Signal tree. Alternatively, you can click on the busform in the clocking diagram. Repeated clicking on a busform will cycle through the signals in the order they are displayed in the Signal tree.
5. Adjust the local setup and hold values for your signal by dragging the Setup or Hold indicators, dragging the entire busform, or by entering specific values in the boxes.

When you select more than one signal (within the same sample or across samples) the logic analyzer displays an open selection outline around each of the signals.

Interactions Between Global and Local Sampling Definitions

When you first open the Signal Sampling screen, the default settings are based on the global default values. When you change a local setting, the setting is no longer affected by changes that you make at the global level.

If you work in a local context and you change the global settings, any local settings that you have not manually changed will be updated to the global settings. To reestablish the local value connection with the global values, click the Use Global Values check box. The check box affects the T_s and T_h values of the current [highlighted signals](#) in the Signal tree. The check box is disabled when no signals are highlighted.

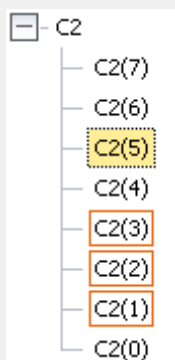
Related Topics

[Signal Sampling Screen: Setting Thresholds \(see page 23\)](#)

[Group Deskew \(see page 17\)](#)

Highlighted vs. Selected Signals

In the sample tree structure below, signals C2(3), C2(2), and C2(1) were selected and highlighted when you first opened the Signal Sampling screen. Any changes that you made to the highlighted signals were automatically applied.



When you click signal C2(5), the signal becomes highlighted while signals C2(3), C2(2), and C2(1) remain selected, but not highlighted. Any changes that you make in the Signal Sampling screen will be applied to highlighted signals, in this case, just C2(5). This gives you the flexibility to change signals by highlighting them without having to close and reenter the Signal Sampling screen.

Highlighting and selection apply to both sides of the Signal Sampling screen: changing thresholds and changing setup and hold times. In the case of thresholds, the threshold slider highlights based on which signals you have highlighted. When you move the slider, only the highlighted signals are affected.

When used with thresholds, the highlighted signals determine which buttons on the left side of the Signal Sampling screen become available for changing thresholds.

Acquisition Options

The acquisition options determine the data to store. The options that are available depend on the sampling type that you specify.

To Use

1. From the TLA toolbar, click the Setup button for your module.
2. Select your sampling type (synchronous, asynchronous, or custom).
3. Select Samples to store only the samples you acquire (default).
4. Select 63-Sample Blocks to store 31 samples before your stored sample and 31 samples after your stored sample. (Use conditional storage in Trigger to store only events of interest).
5. Select Samples + Glitches (Asynchronous) or Samples + Violations (Synchronous) to store both the acquired sample and glitch or violation data.

Behavior

Sampling type	Acquisition options and resulting data
Asynchronous	<p>Samples - stores samples only</p> <p>63-Sample Blocks - stores blocks of data around each stored sample</p> <p>Samples + Glitches - stores both signal and glitch data</p>
Synchronous	<p>Samples - stores samples only</p> <p>63-Sample Blocks - stores blocks of data around each stored sample</p> <p>Samples + Violations - stores both samples and setup & hold data</p>

NOTE. *Selecting Samples + Glitches or Samples + Violations decreases the maximum memory depth by half and decreases the maximum sampling speed.*

Related Topics

[Sampling \(see page 14\)](#)

[LA Setup Window \(see page 13\)](#)

[63-Sample Blocks Mode \(see page 162\)](#)

Samples + Violation Storage Mode

The Samples + Violation storage mode captures both setup and hold violation data and regular data for each signal. This mode limits memory depth to half of the memory depth.

Samples + Glitches Storage Mode

The Samples + Glitches storage mode captures both glitch data and regular data for each signal. This mode limits memory depth to half of the maximum depth and limits the sample period to 4 ns or greater for TLA7000 Series logic analyzer modules and TLA5000B Series logic analyzers.

Sample Storage

Use the Samples per Signal setting to specify the total number of samples acquired for each signal by the logic analyzer.

To Use

1. From the TLA toolbar, click the Setup button for your module.
2. In the Setup window, use the slider to set the Samples per Signal.

Behavior

For a given number of samples, there is a trade-off between the sample rate and the sample duration, and the time period over which samples are taken. A faster sample rate provides a more detailed record of a shorter sampling duration.

If you do not require the maximum number of samples, select a lesser value. This will improve the response time for search and zoom operations.

NOTE. *Selecting Samples + Glitches reduces the available memory by one half. You can double the available memory by increasing the sample rate below a certain value in Asynchronous sampling. In Synchronous sampling, you can double the available memory by increasing the maximum clock rate of the SUT clock rate by reducing the clock period.*

Related Topics

[Selecting Memory: A General Approach \(see page 90\)](#)

[Acquisition Options \(see page 25\)](#)

Merged Module Synchronous Sampling

If you use synchronous sampling with a merged module formation and you use a single clock source, you only need to connect the clock lead from the master module to the clock source.

This condition is also true for qualifier lines.

Related Topic

[Merge Modules Properties \(see page 372\)](#)

Highlight Signal Names Dialog Box

Use this dialog box to highlight signals by name.

1. From the toolbar, open the Setup menu and select your module.
2. In the signal mapping area, right-click and select Highlight Signal Names.
3. In the Highlight Signal Names dialog box, enter the text you want to find. For example, enter RESET to highlight any signal name that contains RESET.

NOTE. *You can enter multiple strings, separated by spaces. The space-separated strings are logically ORed. That is, the name that matches any one of the strings will be highlighted.*

4. Select the highlight criteria.
5. Click OK.

Grouping

Group cards list the signals that make up each group. Use the group cards to do the following:

- Create arbitrary collections of signals by dragging them from the Signal Mapping area.
You can type the signal name in an empty cell in the group card grid. You can also type the signal name in the [Keystroke Entry field](#).
- Arrange the signals in order of significance by clicking and dragging them within the group card; the Keystroke Entry field automatically updates when you rearrange signals.
- Establish operating conditions on the logical groups to establish proper operating conditions of the logic analyzer.
- Provide the appropriate logical relationship to the system under test.

There are several ways to delete signals within a group card:

- Delete the signal name from the Keystroke Entry field.
- Select a signal name within a cell and delete it.
- Select one or more signals and press the Delete key.
- Select an individual signal and drag it to another group or to the Signal Mapping area.

When you delete a signal from a group card, the signals on the least-significant bit side of the deleted signal move to fill the gap created by the deleted signal.

Keystroke Entry Field

Use the Keystroke Entry field to enter valid expressions for signals in the group card. You can enter hardware section notation variants (such as A3, A2(), E1(0–4)) as well as logical names. The field automatically updates the section notation to individual hardware names. The field also applies logical names to signals, if they are available, such as when your logic analyzer has a bus support loaded.

iCapture Analog Feeds

Use the iCapture Analog Feeds dialog box to assign a digital data signal to a TLA7000 Series logic analyzer module analog output.

You can assign data signals in the following ways:

- Select an individual channel; then assign it to an analog output channel.
- Select a group of channels; then select the analog output channels to which you want them assigned.

The Physical Module field, which is located at the top of the dialog box, allows you to select which logic analyzer module channels you want to direct to an oscilloscope.

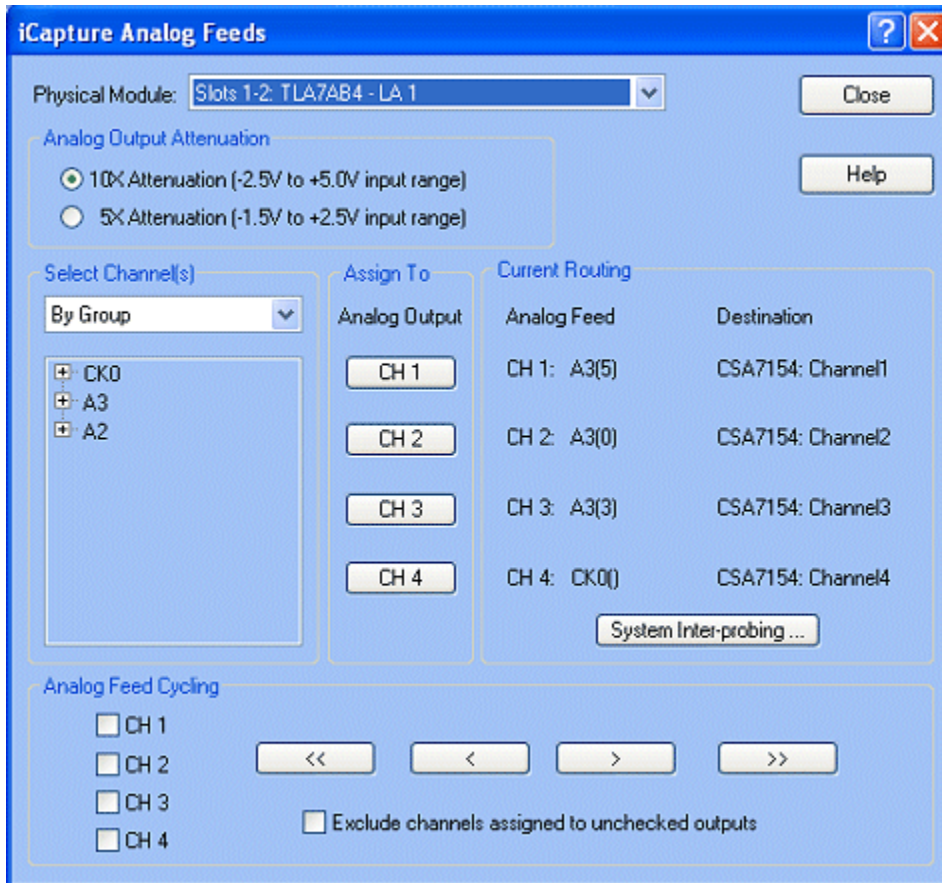
NOTE. *iCapture Analog Feeds are only available in TLA7000 Series logic analyzer modules.*

To Use

From the TLA toolbar, click the iCapture button.

Behavior

Under Analog Output Attenuation, select an attenuation level based on the amplitude of the signals being assigned. 5X Attenuation is best suited for small amplitude, high-performance signals for improved signal to noise ratio. Please note that waveforms displayed on your oscilloscope will be displayed at twice their normal height because the attenuating factor is 5X and not 10X.



After selecting which module contains signals that you want to output, you use the contents of Select Channels to choose one or more specific signals to use as feeds for analog outputs. Select Channels contains a drop-down list that you can use to choose the manner in which digital signals are displayed.

The drop-down list provides the following display options:

- **By Group.** This displays signals organized by group. Only groups that are defined in the LA Setup window can be displayed using this option. In addition, a group can only be expanded if it contains signals from the selected physical module.
- **By Probe.** This displays signals that are alphanumerically organized by individual signal. All signals are displayed when this option is selected, regardless of whether they are part of a group that is defined in the LA Setup window.
- **By Name.** This displays signals by name assigned to them from the LA Setup window. Signals that have not been assigned custom names are not listed when using this selection.

Selecting Individual Signals

After you have chosen a digital signal to output, you can assign it to an analog output signal by clicking a signal button under Assign To. You can confirm that the digital signal was correctly routed by reviewing the analog feed assignment information under Current Routing.

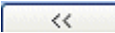
Selecting Groups


When you select multiple digital signals to output, you can assign them to one or more analog output signals by selecting the check boxes associated with the target signals. If you are interested in more digital signals than you have analog outputs, use the Analog Feed Cycling buttons to cycle the selected digital signals through the selected analog outputs. You can additionally direct the system to exclude signals assigned to unchecked outputs, which guarantees that signal assignments to unselected outputs are preserved. To do so, select the check box labeled Exclude signals assigned to unchecked outputs.


NOTE. *The system reassigns digital signals only when the assigned digital signal is a member of the signals that you specified for output in Selected Channels and the analog output signal to which the digital signal is assigned is not selected.*


If one or more of the selected digital signals are already assigned to an analog output, those signals are highlighted under Select Channels. As you feed digital signals to the selected analog output signals, this highlighting changes to reflect current signal routing.

Use the following Analog Feed Cycling buttons to cycle the selected digital signals to the selected analog output signals.

 Assigns up to the first four (depending on number of analog output signals selected) digital signals to the selected analog output signals.

 Assigns the selected digital signals to the selected signal outputs in groups equal to the number of signal outputs selected. The order in which the digital signals are assigned is from the bottom of the list to the top.

 Assigns the selected digital signals to the selected signal outputs, in groups equal to the number of signal outputs selected. The order in which the digital signals are assigned is from the top of the list to the bottom.

 Assigns up to the last four (depending on the number of analog output signals selected) digital signals to the selected analog output signals.

The Current Routing group box displays read-only details about the assignment of individual data signals to analog outputs and the oscilloscope input signals to which they are physically connected. It additionally provides a System Inter-probing button that you can click to reassign oscilloscope destination of data signals.

TIP. *To acquire the best possible signal-to-noise ratio on live waveforms through the analog probe outputs, use an external oscilloscope when the logic analyzer acquisition is idle.*

The iCapture Analog Feeds dialog box retains the signal assignment settings for each TLA7000 Series logic analyzer module represented in the System window. When using Analog Feed Cycling, you can assign signals using one of the previously described control buttons, then close the dialog box. Later, when you reopen the dialog box, you can resume assigning digital signals to analog outputs. However, if you change either the setup of groups or any signal names, signal assignment settings will return to defaults the next time the dialog box is opened.

When using the logic analyzer's demultiplexing capabilities to acquire digital data, the analog output data may not match the digital data. The analog output connector always gets the signal directly from what is present at the probe tip. For a demultiplexed destination signal, the digital data comes from the demux source signal and not from the probe tip. Looking at the LA Setup dialog box signal grid, you can easily tell which analog output signals will not match the digital data: Demultiplexed destination signals (pink background) and signals forced to 1 or 0 (yellow background) will not match.

Related Topics

[System Inter-Probing \(see page 51\)](#)

[Changing Signal Routing \(see page 67\)](#)

[Setting Up Analog Feeds \(see page 64\)](#)

Filter Definitions Dialog Box

From the toolbar, click the Filters button for your module.

Overview

Use this dialog box to display and manage all filters in the logic analyzer.

Behavior

The dialog box displays all modules and filters currently used by the logic analyzer. Use the tree controls to expand and collapse the filter contents of the modules. Modules are listed first, followed by any loaded data sources, and then the filters. Loaded data sources consist of a file name and a module name (for example, System1.tla:LA-1).

To Use

Manage the filters using the toolbar buttons at the top of the dialog box, the right-click menu, or the keyboard. By selecting a module in the dialog box, you can attach a new filter to the module, load an existing filter, or paste a new filter using the clipboard contents and attach it to the module. By selecting a filter, you can cut, copy, paste filter contents, load another filter's contents from the [Load Filter Dialog](#)

[Box \(see page 34\)](#), or you can view or edit the filter’s contents through the [Define Filters dialog box \(see page 33\)](#).

Define Filter Dialog Box

Overview

The logic analyzer normally displays all data acquired by the probes from the target system. Frequently, it displays data that you are not interested in. You can use data filters to display only the data that you want to see in the data window.

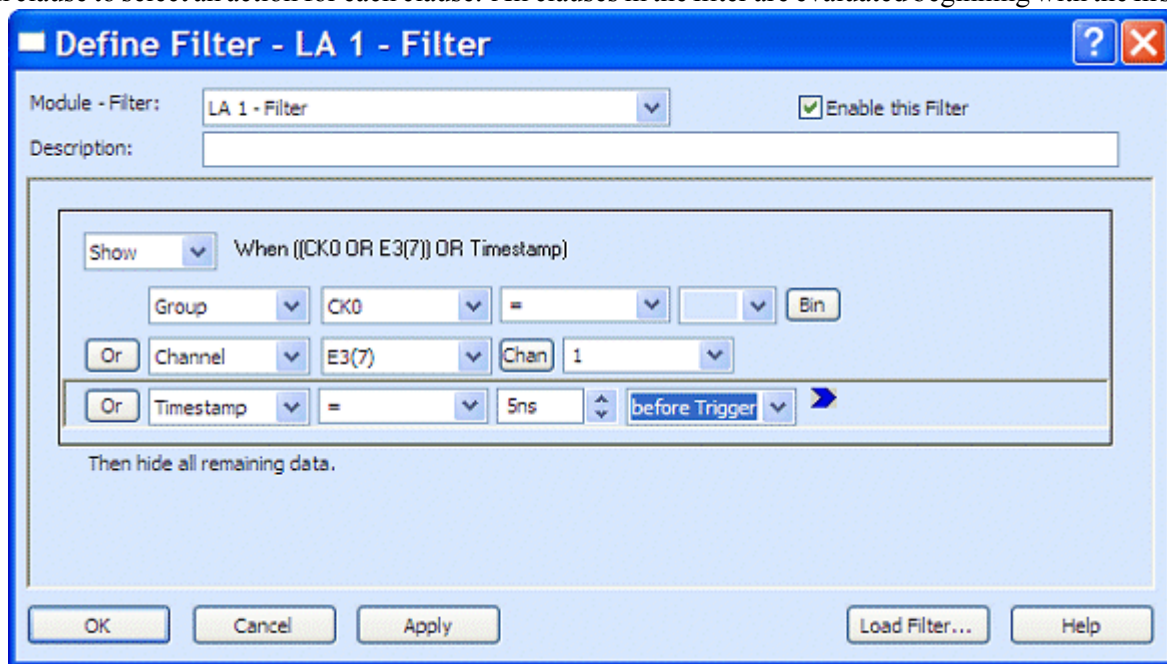
Use filters to do the following:

- Hide specific data. For example, you can define a filter to hide data when all of the data in a group is equal to FFFF.
- Show specific data. For example, you can define another filter to show data from a specific address.
- Color specific data. For example, you can define a filter to display data to another address with a specified color.

Behavior

Filters are made up of clauses and events and are very similar to trigger definitions. Use the drop-down list located at the upper left corner of each clause to select an action for each clause. All clauses in the filter are evaluated beginning with the first

clause.



When you select a Show or Hide action, you can fill in the event fields by using the drop-down lists or text boxes to define the event that you want to either show or hide.

When you select a Color action, a color options button appears where you can click the color options for your filter. After you select the color options, you can fill in the event fields to further define the data that you want to filter for the specific event.

You can edit events and clauses by clicking the mouse at the right-end of the clause and either placing the mouse over the indicator or right-clicking the mouse to display a context-sensitive menu.

Filters are saved as part of the module setups. You can load filters when you need them. You can also edit and rename filters. When you apply a filter to a data window, the filter name is appended to the data label (for example LA1: FilterA: A3 indicates that FilterA is attached to the A3 group of module LA1).

To Use

Use this dialog box similarly to the trigger window by defining events and clauses for the filter. By default the filter is enabled if it is not enabled, click the Enable this Filter check box.

Load Filter Dialog Box

Overview

Use this dialog box to load filter contents from any current or saved module setups.

Behavior

This dialog box loads filter contents from any saved module setup files or system setup files. The load operation replaces all settings in the selected filter (specified in the title bar). If no filter is selected, the load operation creates a new filter and attaches it to a module.

When a module setup name or system setup name appears in the File Name text box, all modules and filters associated with that setup are listed in the center of the dialog box. If a module has a filter attached, you can select and load that filter.

To Use

Enter a module setup or a system setup name in the File Name text box or click the Browse button to locate a module or system setup file. Select a filter from the setup to enable the Load button; click the load button to load the filter. You can also double-click the filter to automatically load the filter.

NOTE. *If you load a filter that uses groups or signals that are not included with the module, the events will be discarded. A message box will display, explaining what happened.*

Related Topic

[Load Filter Browser Dialog Box \(see page 35\)](#)

Load Filter Browser Dialog Box

Use this dialog box to find TLA module or system setup files that contain filters. This dialog box is similar to the standard Open Dialog Box. The main exception is that the Content area describes whether the module has any filters defined.

Use the dialog box to browse the file system for filters. After you select a file name and click the Open button, the dialog box closes and adds the file name to the File Name field of the [Load Filter Dialog Box \(see page 34\)](#).



TIP. *If you don't see the file that you want, make sure that the file is a valid TLA module setup file or system setup file. These files have the .tla suffix.*

Applying Filters

When you apply a filter to a data window, the filtered data appears as a data source. The filter name is appended to the data source name in a data window. Define the filter before you can apply it to a data window.


1. In the Listing window, click the column name; in the Waveform window, click a waveform name.
2. Right-click the mouse and elect Apply Filter followed by the filter name.

You can add a new column or waveform with filtered data by clicking the Add Column or Add Waveform icons and selecting a filtered data source:

1. In the Listing window, click  (the Add Column icon); in the Waveform window, click  (the Add Waveform icon), .
2. In the new dialog box, select a data source that contains filtered data. For example, LA1:Filter1.
3. Select the data group that contains the filtered data.
4. Click the Add button followed by the Close button.

Color Options Dialog Box

Use this dialog box to control the color of groups in the data window.

1. Select a filter in the Filter Definitions dialog box and then click  to open the filter.
2. Select Color from the filter definition.
3. Click the button to the right of the drop-down list to open the Color Options dialog box.
4. Determine whether you want to use separate colors for an individual group or the same color for all groups.
5. Click the color button to select the foreground and background color from the color matrix.

In addition to using a basic color, you can click More Colors... at the bottom of the color selection box to define custom colors.

6. Click OK to close the dialog box.

Signal Import and Export Formats

The Signal import and export uses a structured text file format to organize the import and export information. Only formatted text files can be processed by the parser.

The import or export file is separated into multiple lines by EOL (EOL can be CR, LF, or CRLF). During importing or exporting, the parser processes one line at a time. The following lines are allowed in the file:

[Comment Lines \(see page 40\)](#)

[Control Lines \(see page 41\)](#)

[Information Lines \(see page 43\)](#)

Click [Sample File \(see page 38\)](#) to view an Import/Export file.

Related Topics

[Signal Setup Import Dialog Box \(see page 38\)](#)

[Signal Setup Export Dialog Box \(see page 37\)](#)

[Signal Setup Import Behavior \(see page 36\)](#)

Signal Setup Import Behavior

Be aware of some of the behavior of the logic analyzer when it imports signal setup files:

- The logic analyzer detects the delimiters used in the imported file. Therefore it is important to use only one delimiter in the file. If you need to embed the delimiters in a signal name or group name, enclose the name with quotes.
- The import operation will update the signal setup parameters related to the selected fields. For example, if a user-defined signal name is selected, the old name will be replaced by the new name without any

warnings. Similarly, if there are two lines in the imported file setting for the same signals, the second line receives the new name.

- The import operation will not change the signal setup parameters related to unselected fields. A skipped field (an empty field name between two delimiters) acts as an unselected field.
- User-defined signal names must be unique. If an imported user-defined signal name exists, the two signals will exchange their user-defined names (for example, if you have A0(0):A and A0(1):B, and you set A0(1) to A, the result will be A0(0):B and A0(1):A).
- If an imported file contains a definition for a signal that does not exist, the definition will be ignored and a warning message will be saved in the error log file. You may experience this when importing a file to a narrow module.
- If an imported file does not contain definitions for all the signals of the logic analyzer, the definitions of the missing signals remain unchanged. This may happen when you import a file to a wide module.
- The parser processes one line at a time. It will stop processing on any unrecognizable line and then display an error message.
- All warning and error messages will be written to a file named ImportExportError.log in the same location as the import file.

Signal Setup Export Dialog Box

Overview

Use this dialog box to export signal setups to a text file. The setups can then be imported by another logic analyzer.

Behavior

This dialog box allows you to save the setups to a text file listed in the Export Path box. You can also click the Browse button to save the file to a folder of your choosing. You have the choice of exporting individual signal information or group information by selecting the appropriate Export Selections. If you select Export Signal Information, you can specify the type of information that you want to export by selecting one or more of the check boxes. You can select one of four field delimiters to separate the fields within the file; an empty field is represented by two consecutive field delimiters in the file.

Any warning or error messages are saved in a log file in the same location as the exported file.

To Use

Select Export Signal Setup... from the File menu. Specify a file name and the path to the file. Use the other selections in the dialog box to define the signal information and delimiters. Click the Export button to export the file.

Related Topics[Signal Setup Import Dialog Box \(see page 38\)](#)[Signal Import and Export Formats \(see page 36\)](#)[Signal Setup Import Behavior \(see page 36\)](#)

Signal Setup Import Dialog Box

Overview

Use this dialog box to import signal setups from a saved location.

Behavior

This dialog box allows you to specify the signal setup information that you want to import. Specify the file location in the Import Path box or click the Browse button to browse the file system for the file. The setups are normally saved in a text file.

Select the Automatically detect from file to import all fields from the import file. Select Custom options to specify the fields that you want to import by selecting one or more custom options. If you try to import a field that does not exist in the import file, the logic analyzer will display a warning message. Click the Import button to import the setups. Warning and error messages are saved in a log file in the same location as the imported file.

To Use

Select Import Signal Setup... from the File menu while the Setup window is active. Locate the import file, specify the import options in the dialog box, and then click the Import button.

Related Topics[Signal Setup Export Dialog Box \(see page 37\)](#)[Signal Import and Export Formats \(see page 36\)](#)[Signal Setup Import Behavior \(see page 36\)](#)

Sample Signal Setup File

The following file shows an example of the different parts of a Signal Import/Export file.

```
////////////////////////////////////
```

```
// TLA Signal Import/Export File
```

```
// Created on June 10, 2007 at 16:05:27
// by TLA Import/Export Utility Version: 1.0.001
// (Field Delimiter: <Tab>)
////////////////////////////////////////////////////////////////
////////////////////////////////////////////////////////////////
// Remove all user-defined signal names
//+ResetAllSignals
//
// Remove all groups
//+RemoveAllGroups
////////////////////////////////////////////////////////////////
////////////////////////////////////////////////////////////////
// signal definition format
//+SignalFormat : HWName UserName Polarity Compare
////////////////////////////////////////////////////////////////
C0(7) NMI - TRUE
C0(6) IGNNE# - TRUE
C0(5) EADS# - TRUE
C0(4) SRESET - TRUE
C0(3) HOLD - TRUE
C0(2) KEN# - TRUE
C0(1) LOCK# - TRUE
C0(0) RDY#= - TRUE
Q3 Q3 - TRUE
Q2 Q2 - TRUE
Q1 Q1 - TRUE
Q0 Q0 - TRUE
CK3 CLK - TRUE
CK2 RDY# - TRUE
CK1 BRDY# - TRUE
CK0 BRDYC# - TRUE
```

```

////////////////////////////////////
// Group definition format
//+GroupFormat :  GroupName ChannelList
////////////////////////////////////
Data D31 D30 D29 D28 D27 D26 D25 D24 D23 D22 D21 D20 D19 D18 D17 D16 D15 D14
D13 D12 D11 D10 D9 D8 D7 D6 D5 D4 D3 D2 D1 D0

Control BLAST# AHOLD HLDA BOFF# M/IO# D/C# W/R# SMIACT#

DataSize BS8# BS16# BE3# BE2# BE1# BE0#

Misc CLK= ADS#

Cache KEN# EADS# FLUSH#

Misc2 BREQ# INTR LOCK# PLOCK# RDY#= BRDY#= BRDYC#= HOLD IGNNE# FERR# RESET
SRESET NMI

```

Comment Lines

All lines starting with "/" are treated as comments, unless the very next character is a '+', which signifies a control line. The "/" can also be used to initiate a comment on the end of "non-comment" line. In this case, a delimiter is needed before the "/", all text between the "/", and the end of the line is treated as a comment (ignored by the parser). Examples of using comments lines are shown below:

```

////////////////////////////////////
// TLA Signal Import/Export File
// Created on June 10, 2007 at 16:05:27
// by TLA Import/Export Utility Version: 1.0.001
// (Field Delimiter: <Tab>)
////////////////////////////////////

```

Related Topics

[Control Lines \(see page 41\)](#)

[Information Lines \(see page 43\)](#)

[Sample Signal Setup File \(see page 38\)](#)

[Signal Setup Import Behavior \(see page 36\)](#)

Control Lines

All lines starting with "//+" are treated as control lines. This special character sequence is used to mark instructions to the parser. Each word in a control line is considered as a keyword, which can be recognized by the parser. Only delimiters and predefined keywords are allowed in a control line. Each control line signifies a structure, which is used to organize the information in the following information lines.

The first keyword in a control line specifies the name of the defined structure. The keywords after the structure name specify the names of the structure items. Currently, there are four structure names available, **ResetAllSignals**, **RemoveAllGroups**, **SignalFormat**, and **GroupFormat**. The keywords are not case sensitive.

ResetAllSignals

This keyword works as a command to remove all user-defined signal names as shown in the following example:

```
////////////////////////////////////
// Remove all user-defined signal names.
//+ResetAllSignals
////////////////////////////////////
```

RemoveAllGroups

This keyword works as a command to remove all of the groups.

```
////////////////////////////////////
// Remove all groups.
//+RemoveAllGroups
////////////////////////////////////
```

SignalFormat :

This is a structured line containing the following fields: HWName, UserName, Polarity, Threshold, and Compare.

HWName. This is the hardware name of the signal. It is required in all SignalFormat structures. The hardware name of the signal should never be empty in the information lines. For normal acquisition signals, the syntax is the hardware pod name followed by the signal number enclosed in parentheses, (for example, A0(1) or A0(2)). The syntax for clock and qualifier signals is the type identifier "CK " or " Q" respectively, followed by a number (for example, CK0, CK1, Q0, or Q1). This field is not case sensitive.

UserName. This is the user-defined name of the signal. This field **is** case sensitive. The user name should be unique among signals; otherwise the latest defined name will replace the pre-existing name. The user

name cannot contain embedded spaces, commas, or tabs. The TLA application allows semicolons in names. When using semicolons as delimiters, embed the name and the delimiter within double quotes ("sample;name"). Similarly, if you want to embed "/" in the name, the name and the "/" symbol must be enclosed in double quotes.

Polarity. This is the polarity of the signal. The values for this field are + and -.

Threshold. This is the threshold voltage of the signal. The syntax is the voltage value and the unit, (for example, 2.5V, or 100mV). This field is not case sensitive.

Compare. This field contains the compare enable/disable information of the signal. Values for this field are TRUE and FALSE. This field is not case sensitive.

```

////////////////////////////////////
// Signal definition format
//+SignalFormat: HWName UserName Polarity Threshold Compare
////////////////////////////////////
C1(7),RESET,-,1.5V,TRUE

```

GroupFormat

This is a structured line containing the following fields: GroupName and SignalList .

GroupName. This field defines the group name. This field is case sensitive. The group name must be unique; otherwise the later-defined name replaces the previous-defined name. The group name must not contain embedded tabs. When using delimiters with the name, embed the group name and the delimiter within quotes ("sample name").

SignalList. This field is a list of signal names assigned to the group. This field is case sensitive. The names can be user-defined names or probe-based names. When using probe-based names, there are several ways to specify the signals. For individual signals, the syntax is the hardware pod name followed by the signal number enclosed in parentheses (for example A0(1) or A0(2)). Groups of contiguous signals can be specified using a range of signal numbers within the parentheses (for example, A0(7-0) or A1(3-5)). When you want to specify all signals in a hardware pod, you can use a shorthand notation using an empty pair of parentheses (for example, A0() or A1()). The syntax for clock and qualifier signals is the type identifier "CK " or " Q" respectively, followed by a number (for example, CK0, CK1, Q0, or Q1). Multiple signals or groups must be separated by delimiters. Embedded spaces are not allowed.

The number of the names is unlimited. The parser reads the delimiter-separated names until it finds an EOL character or a "/" sequence. You can break a long line into several lines by adding a delimiter followed by the '\ ' character before you add each EOL character. The parser treats those lines as a single line.

```

////////////////////////////////////
// Group definition format
//+GroupFormat:GroupName SignalList
////////////////////////////////////
Control,BLAST,AHOLD,HLDA,BOFF,M/IO,D/C,W/R,SMIACT

```

Data,D31,D30,D29,D28,D27,D26,D25,D24,D23,D22,D21,D20,D19,D18,D17,D16,,\D15,D14,D13,D12,D11,D10,D9,D8,D7,D6,D5,D4,D3,D2,D1,D0

Address,A0(),A1(),A2(),A3()

Related Topics

[Comment Lines \(see page 40\)](#)

[Information Lines \(see page 43\)](#)

[Sample Signal Setup File \(see page 38\)](#)

[Signal Setup Import Behavior \(see page 36\)](#)

Information Lines

Lines not starting with "/" or "/" are treated as information lines. An information line can be separated into several fields by a pre-defined delimiter. The orders of the fields are declared in the control line defined earlier. For more information, see the [Sample File. \(see page 38\)](#)

Related Topics

[Control Lines \(see page 41\)](#)

[Comment Lines \(see page 40\)](#)

[Sample Signal Setup File \(see page 38\)](#)

[Signal Setup Import Behavior \(see page 36\)](#)

Serial Analyzer Setup Window

The Setup window provides access to all controls needed to configure the serial analyzer module to capture data from a particular serial data stream or link.

A dedicated status area at the top of the window provides basic status information for the current link. Controls at the left side of the window define the link, define the reference clock, and to specify how to store data.

The signal activity indicators in the Setup window provide a visual indication of the signal activity at the probe. The module always acquires data and updates the status indicators. The legend items in the status area near the top of the Setup window describes the various signal states.


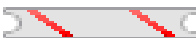


Status Area

The status area is located near the top of the serial analyzer Setup window. It provides status information for the Setup window and includes the following components:

- Text messages
- Command buttons
- Channel activity indicator legend
- Link status readout

Text messages provide informational, warning, or violation messages about the status of the Setup window and the module. Some messages include an Acknowledge button. These messages appear when you make a change to the analyzer settings that may require your further interaction. Click the Ack button to dismiss the message.

The signal activity indicator legend defines the signal status of the signals and the system under test (SUT) clock signals. The following table lists the possible values and their indication.

Indicator	Meaning	Description
	No signal	No signal activity and none was expected
	Missing signal	No signal activity. The signal is connected to a lane on the system under test and a signal is expected.
	Signal	Signal activity. A symbol lock was not achieved.
	Data signal	Signal activity. The serial analyzer has achieved symbol lock. In case of the SUT clock, a valid clock was detected at the front panel connector.

If the indicators appear different than what you expect, this could indicate a possible problem with the probe or the signals at the system under test. The legend is always on the screen unless another status message is displayed.

The right side of the status area shows key information about the current condition of the link. The top readout displays the link width during the previous update interval and the bottom readout displays the transfer rate. The readout displays “Link not Trained” when the link is not trained or when no signals are active. If the link width changes during the update interval, the readout displays “Varies.”

Autoset Status

When a link has trained, the serial analyzer can detect the training results for each lane. When this occurs, the serial analyzer displays a status message along with the Ack button and the Autoset button. When you click the Autoset button, the serial analyzer automatically completes the following setups for the link:

- Link direction
- Link width. The highest lane number among the trained lanes plus one.
- Channel-to-lane associations. The Setup window displays the associations with lines connecting from the lanes to the activity indicators.
- Channel polarity

The serial analyzer can only detect changes in the transfer rate when Transfer Rate is set to Track Training. If the Transfer Rate is set to one of the fixed rates, the serial analyzer cannot detect the signals at the link when they train to a different rate. These changes are reported in the status area.

To disregard the changes and dismiss the status message, click the Ack button.

Left Side Control Area

The Left-side controls configure the behavior of the serial analyzer. The area contains three groups of controls: the link definition group, reference clock source group, and the link data storage group.

Link Definition Group

The name of the link appears on the tab at the upper left side of the control area. You can rename the link by clicking text and entering a new name.

Select one of the Acquire buttons to identify the upstream and downstream lanes for capturing data. The selection that you make impacts the appearance of the signal-lane assignment area on the right side. For example, if you acquire downstream data, the upstream signal-lane assignments become unavailable.

Select the maximum link width from the drop-down list. The link widths depend on the other settings in the Link Definition group, such as the number of signals in the module and the Acquire selections. It is a good idea to select the largest width to which the link can train because the link can down-train during operation. If the serial analyzer detects changes that may require further user interaction, it displays a message in the status area.

Select the transfer rate from the drop-down list. Use Track Training when you want the serial analyzer to watch the TS1 and TS2 training sequences to determine the transfer rate that the link is trying to establish or to anticipate and to follow any rate changes.

Reference Clock

The serial analyzer can synchronize its reference clock to one of two sources.

Select Recovered from Data to synchronize the serial analyzer with a clock signal embedded in the data signal.


Select SUT to use an external clock from the system under test and then select one of the clock rates from the drop-down list. The serial analyzer receives the clock signal from the external clock cable connected to the system under test.

Data Storage Area

Use the Data Storage area to specify the data format and the amount of data to acquire and the type of filtering. The data volume is expressed in terms of symbols per lane with a maximum of 32 M symbols per lane.

Use the Trig Pos control to specify where the trigger will appear in memory (1% accuracy within the data record).

In addition to specifying the amount of data to acquire, you can also filter data during an acquisition cycle.

Use the filters from the drop-down list or define your own by clicking  (the Define Filter button). The filtering applies to both upstream and downstream data and does not restrict any triggering.

Use the Define Filter function to define a filter that meets your needs. After you create a filter, you can save it for future use. The new filter appears in both the upstream and downstream lists. The last filter that you use becomes the selected filter in the lists.

Channel-Lane Assignment Area

The Channel-Lane Association area of the serial analyzer Setup window provides controls for making logical connections between module signals and the logical lanes of the Listing and Waveform windows. The area consists of graphical representations of the module bordered by the upstream and downstream lanes. The downstream and upstream lanes provide connection points to the module.

Connector groups (of four Channel Activity Indicators) represent the connectors on the front of the serial analyzer. Each group has four signals consisting of a signal emblem, a polarity indicator, and two signal connection points (one on each side of the emblem). The SUT Clock emblem shows the activity of the clock when you select the SUT clock as the reference clock source.

Use the polarity indicators to change how the serial analyzer senses the differential signal on a signal. The plus and minus signs represent the normal or inverted polarity. The buttons show the last trained value after you click the Autoset button. However, the values do not automatically track training.

When the probes are connected to the system under test, the appearance of the emblems represent the status of the signals at the probe tips. Refer to the legend at the top of the Setup window to understand the signal status.

The easiest way to connect the lanes to the module connectors is to click the Autoset button after the serial analyzer has detected training. You can also manually connect the lanes to the module connector by clicking the mouse on a lane and dragging a line to the desired module connector.

Setting Up External Oscilloscope Communication

The iView External Oscilloscope Setup wizard provides instructions to connect your Tektronix logic analyzer to an external Tektronix oscilloscope, enabling communication between the two instruments.

- If the external oscilloscope that you are adding does not appear in the list, then the logic analyzer does not have the files necessary to support your model of external oscilloscope. In this event, go to the Tektronix Web site and download the latest external oscilloscope support files.
- After you complete the process of adding your external oscilloscope to the logic analyzer system, you can set up your oscilloscope from the logic analyzer System window.
- You can specify horizontal and vertical settings on external oscilloscope. In addition, if the external oscilloscope is triggering the TLA, you can specify the trigger settings on the external oscilloscope.

iView Setup Window

Overview

A Setup window is available to assist you in verifying, changing, and testing the external oscilloscope settings.

Before acquiring and displaying a waveform, you can first establish a connection between your Tektronix Logic Analyzer and external oscilloscope using the Add iView External Oscilloscope setup wizard.

To Use

To access the setup wizard from the System menu, click Add iView External Oscilloscope. Then step through the external oscilloscope setup process, following the provided directions that guide you through physically connecting the analyzer, choosing both your GPIB address and where to display your external waveform data, and initializing the system trigger.

The Connections tab contains diagrams that help you to confirm that your logic analyzer and external oscilloscope are physically connected correctly. Diagrams are available for confirming the physical connection of both your logic analyzer and your specified external oscilloscope. Click the TLA Connections button to display the logic analyzer diagram, or click the TDS Connections button to display the diagram for the external oscilloscope that you specified in the setup wizard.

To open the iView Setup window from the System window, click the iView Setup button.

Related Topic

[Troubleshooting Connections \(see page 48\)](#)

Troubleshooting External Oscilloscope Communication

If you experience difficulty in establishing communication between your logic analyzer and external oscilloscope, the following checklist will assist you in identifying and correcting the problem.

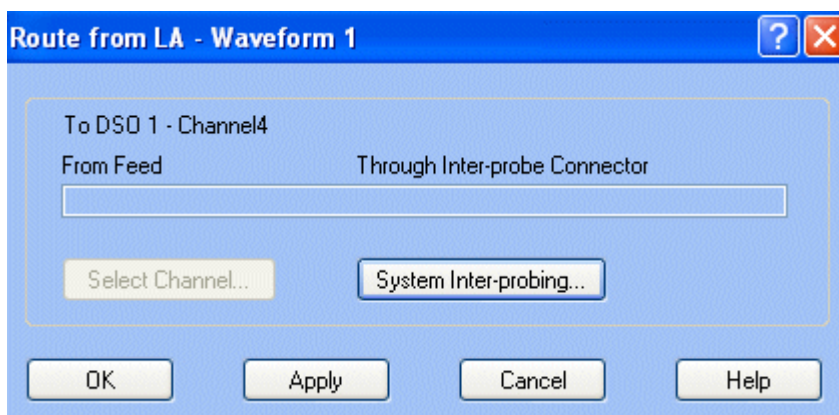
Diagnostic Checklist

- Is your external oscilloscope turned on?
- Is the GPIB configuration on your external oscilloscope set up correctly? (See your oscilloscope's online help.)
- Are your cable connections securely attached?
- Did you select the correct model number for the external oscilloscope to which you are connecting?
Use the Back button to return to the window where you specified the external oscilloscope model number and make changes as necessary.
- Have you correctly connected the cables between your logic analyzer and external oscilloscope?
Use the Back button to return to the windows that describe the proper cable configuration between your logic analyzer and external oscilloscope. Alternatively, you can cancel the setup wizard and begin the setup process again.
- If you specified the external oscilloscope GPIB address instead of letting the setup wizard automatically detect it for you, did you correctly select the GPIB address where your external oscilloscope is set to communicate?
Use the external oscilloscope user interface to ensure that the GPIB address you selected in the setup wizard agrees with the GPIB address at which the external oscilloscope is set to communicate.
Once you correct the communications problem, the error message will disappear when you press the Next button to establish communication.

Signal Routing from the Logic Analyzer

Overview

Use the Route from LA dialog box to send probe data from a logic analyzer signal to the currently selected oscilloscope signal. To display the Route from LA dialog box, select Route from LA from the right-click menu associated with a selected oscilloscope signal waveform.



Behavior

The Route from LA dialog box displays the currently defined inter-probe connection and the feed for the oscilloscope signal. This item cannot be selected and is displayed for information only. If no inter-probe connection is defined for the oscilloscope channel, the field is empty.

To Use

Click the System Inter-probing button to change the contents of the scroll list by adding or changing the connection to the logic analyzer.

Click the Select Channel button to select a new logic analyzer channel to feed the inter-probe connection to the oscilloscope channel. This button is made available when an inter-probe connection is defined.

Related Topics

[Signal Routing to the Oscilloscope \(see page 49\)](#)

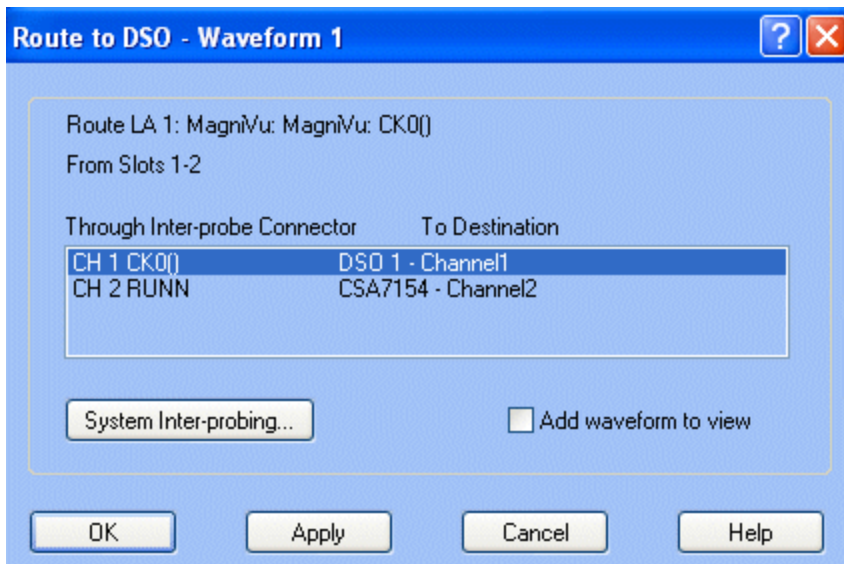
[Routing a Signal from a Logic Analyzer Output \(see page 66\)](#)

[Routing a Signal to an Oscilloscope Input \(see page 66\)](#)

Signal Routing to the Oscilloscope

Overview

Use the Route to DSO dialog box to send probe data from the selected logic analyzer signal to one of the oscilloscope channels. To display the Route to DSO dialog box, select Route to DSO from the right-click menu associated with a selected LA signal waveform. The Route to DSO right-click menu item is only enabled if the selected waveform is a TLA7000 Series logic analyzer signal.



Behavior

The Route to DSO dialog box provides a list of oscilloscope channels that have inter-probe connections defined to the logic analyzer. If no inter-probe connections are defined, the list is empty. If the oscilloscope list contains the desired LA signal, then that list item is picked by default.

The check box named Add waveform to view is unchecked by default because the waveform view typically contains waveforms for the oscilloscope of interest. Selecting this check box and clicking OK will add a waveform of the selected oscilloscope destination to the view.

To Use

Select an oscilloscope channel and click OK to feed the logic analyzer signal to the inter-probe connection attached to the oscilloscope channel.

Click the System Inter-probing button to change the destination list contents by adding or deleting the connections for the logic analyzer.

Related Topics

[Signal Routing from the Logic Analyzer \(see page 48\)](#)

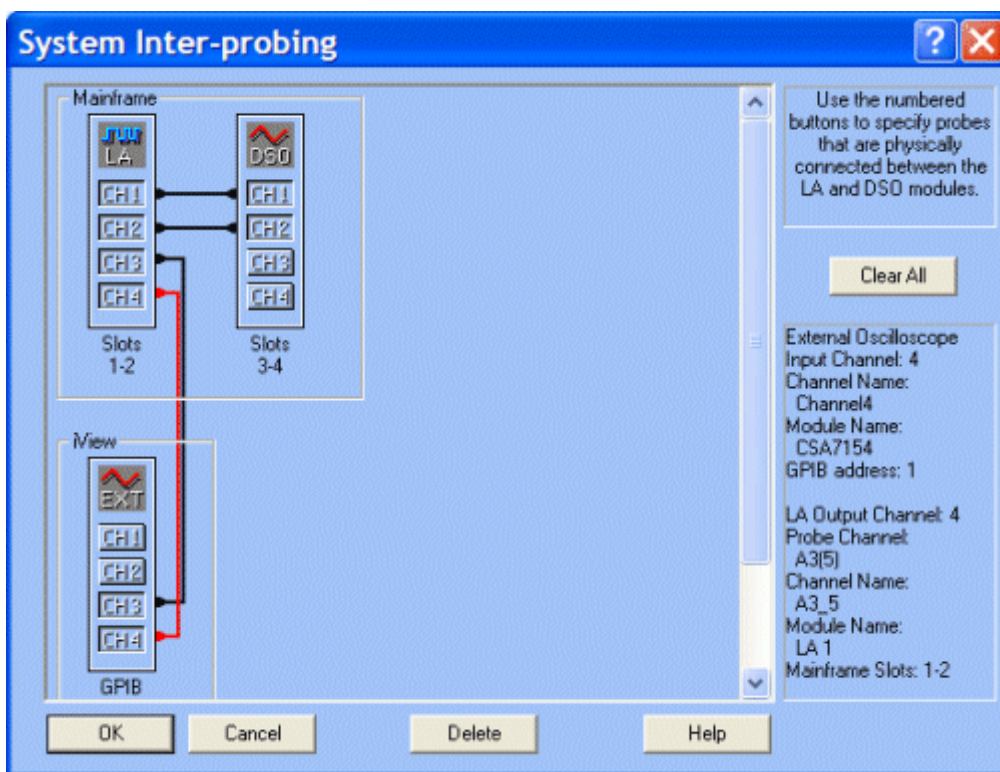
[Routing a Signal from a Logic Analyzer Output \(see page 66\)](#)

[Routing a Signal to an Oscilloscope Input \(see page 66\)](#)

System Inter-Probing

Overview

Because the TLA application cannot sense probe physical connect/disconnects, it is unable to manage logic analyzer or oscilloscope mapping automatically. Therefore, you can use the System Inter-probing dialog box to manually specify which connectors between the logic analyzer and oscilloscope are fitted with probe connections.



Behavior

When an inter-probing connection is created, the oscilloscope signal name changes to reflect the logic analyzer signal that is feeding it. The signal name is visible in both the DSO Setup channel tabs and as the waveform element label. The rule that the system uses for generating the DSO signal name is to make it identical to the LA signal name. The system also automatically adjusts the time alignment dialog to incorporate whatever signal path delay is determined for the oscilloscope individual signal. The delay is a single time value, based on both the cable length and the particular path of the logic analyzer signal feed.

NOTE. Your ability to use the LA Setup, Time Alignment, and Waveform windows depends on correct set up and maintenance of system inter-probing.

To Use

To create an inter-probing connection, click either an oscilloscope or logic analyzer signal button. The system responds by changing the cursor to the shape of a probe connector. Then click a button corresponding to the opposite end of the connection to establish the connection. Upon establishing an inter-probing connection, the cursor changes back to the normal shape.

NOTE. *If you click anywhere other than on a legal endpoint for the connection, then no connection is made and the cursor changes back to the normal shape.*

To delete a system inter-probing connection, click either endpoint button of the existing connection. When a connection is deleted, the oscilloscope signal name changes to the normal default and the oscilloscope individual signal time alignment is set to zero.

Related Topic

[Setting Up Analog Multiplexing \(see page 55\)](#)

Loading Saved Setups to an LA module

1. In the System window, click the LA Setup button.
2. From the File menu, select Load Module.
3. Click Browse to search for the system or module with the setup you want. You can also enter the path in the File name text box.
4. Select the file and click Open; the Load Module dialog box appears with a list of modules in the saved file.
5. Select the module you want and click Load.

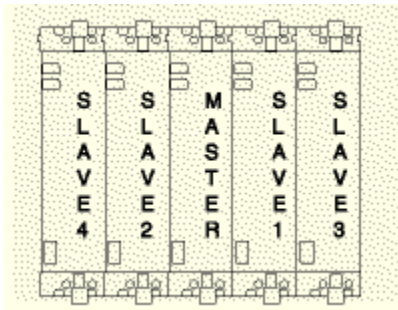
If the setup does not load, refer to [Load System Options Dialog Box \(see page 363\)](#) for a list of possible problems.

Merging Modules

Before you can merge modules in the application, modules must be physically connected as described in the TLA Series Logic Analyzer Installation Manual.

Follow these rules and restrictions when merging modules:


- TLA7BB2 (68-channel) modules can be merged. For other modules, 102 channel and 136 channel modules can be merged.
- TLA7Bxx modules cannot be merged with TLA7Lx, TLA7Mx, TLA7Nx, TLA7Px, TLA7Qx, or TLA7Axx modules.
- TLA7Axx/TLA7NAx modules cannot be merged with TLA7Lx, TLA7Mx, TLA7Nx, TLA7Px, TLA7Qx, or TLA7Bxx modules.
- TLA7Nx modules can be merged together, or with either TLA7Px or TLA7Qx modules.
- TLA7Nx, TLA7Px and TLA7Qx modules cannot be merged with TLA7Lx and TLA7Mx modules.
- Any combination of TLA7Axx and TLA7NAx modules (up to five) can be merged.
- Modules containing different memory depths can be merged but will default to the shallowest memory depth being used.
- The maximum merged combinations are five TLA7Bxx modules, five TLA7Axx modules, five TLA7NAx modules, three TLA7Nx/Px/Qx modules, or two TLA7Lx/Mx modules. The graphic shows how to orient modules in a multiple module merge.



- The number of channels in the master module must be equal to or greater than the number of channels in the modules with which it is merged. If there is a second slave module, the first slave must have greater than or equal to the number of channels in the second slave module. Slave 3 cannot have more channels than the master, slave 1, or slave 2. Slave 4 cannot have more channels than the master, slave 1, slave 2, or slave 3.
- Modules cannot be merged between a mainframe and an expansion mainframe; merged modules must be in the same physical mainframe.
- To merge a module to an established merged set, the established merged set must first be unmerged through software. Only unmerged modules can be used to create a merged configuration.

Use the following steps to merge logic analyzer modules.

1. From the System menu, click System Configuration.
2. Click the Merge Modules tab.

3. Click the merge button.  The button closes and the module icons become one to indicate that the modules are merged.
4. Click the OK button

Related Topics

[Merge Modules Properties \(see page 372\)](#)

[Calibrating Modules \(see page 59\)](#)

Saving LA Setups

1. In the System window, click the LA Setup button.
2. From the File menu, click Save Module.
3. Select the folder.
4. Enter the file name if you want a new name.

5. Enter any comments you want to save with the file.
6. Click the option button associated with the save option you want: Save All Data, Save Only Unsuppressed Data, or Don't Save Data
7. Click Save.

Setting Up Analog Multiplexing

1. From the LA Setup window, click the iCapture button. The system responds by displaying the iCapture Analog Feeds dialog box.

NOTE. *Analog multiplexing is available only in TLA7000 Series logic analyzer modules.*

2. Specify signal routing between the LA module's ANALOG PROBE OUTPUT connectors and both the DSO module's and external oscilloscope's Analog IN connectors (as applicable).
3. Set up signal routing from the LA probe input channel to the LA module's ANALOG PROBE OUTPUT connectors.
4. Click the following links to complete the analog multiplexing setups.

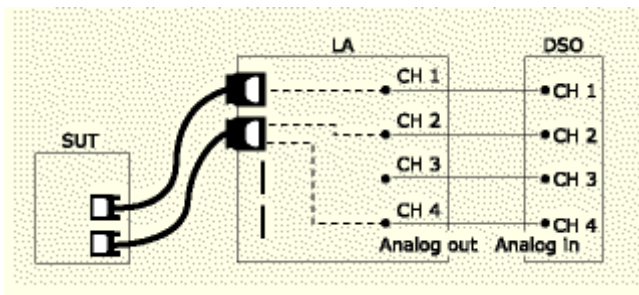
[Setting Up System Inter-Probing \(see page 65\)](#)

[Setting Up Analog Feeds \(see page 64\)](#)

[Changing Signal Routing \(see page 67\)](#)

The following figure depicts the logical process of setting up analog multiplexing.

Establishing a Logical Connection

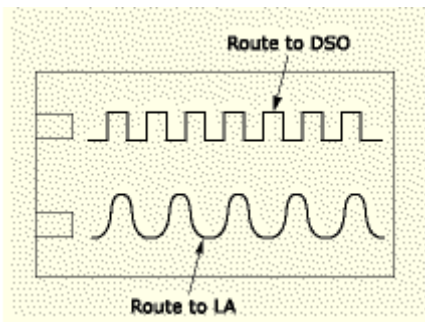


Use the following procedure to establish a logical connection between your logic analyzer and oscilloscope from the Setup window.

1. Connect probe from the channels on system under test to the Tektronix Logic Analyzer.

2. Establish logical connections between the logic analyzer and the DSO, using the [System Inter-probing \(see page 51\)](#) dialog box.
3. Determine which channels to route from the logic analyzer input to the Analog Output, using the [Analog Feeds \(see page 29\)](#) dialog box.

Reconfiguring an Existing Logical Connection



Alternately, you can use the following procedures to reconfigure the logical connection between your logic analyzer and oscilloscope from the Waveform window.

Reconfigure logical connections from the logic analyzer to the oscilloscope:

1. Right-click the waveform label associated with a digital waveform being routed to the oscilloscope.
2. Select [Route To DSO \(see page 49\)](#) from the shortcut menu to display the Route to DSO dialog box.
3. Use the Route to DSO dialog box to modify probe data being sent from the logic analyzer to the oscilloscope.

Reconfigure logical connections from the oscilloscope to the logic analyzer:

1. Right-click the waveform label associated with an analog waveform being routed from the logic analyzer.
2. Select [Route from LA \(see page 48\)](#) from the shortcut menu to display the Route from LA dialog box.
3. Use the Route from LA dialog box to modify probe data being sent from the logic analyzer to the oscilloscope.

Acquisition Overview

Overview

When you start an acquisition, all modules start acquiring data together. (Exceptions are when one module has been programmed to arm another or when a module has been turned off.) Modules stop acquiring data individually, according to their trigger programming.

Starting and Stopping an Acquisition

In the Control bar, click Run to start an acquisition or press the Run/Stop button on the front panel. The Tek icon on the right animates while the logic analyzer runs. Point at this icon with the mouse to display a tool tip showing the analyzer status.

There are two ways to acquire data: Single-run or Repetitive mode. Within Repetitive mode, you can specify the following three conditions:

- Save Module and Data
- Stop if Compare with Reference is Equal or Not Equal
- Stop After N Acquisitions

When comparing acquisition data against reference data, you can view the results in either a Listing window or in a Waveform window.

Single-Run Mode

In Single-run mode, the logic analyzer automatically stops acquiring and displays data when it fulfills the setup conditions. Use Single-run mode to find and display a specific event.

During acquisition, the logic analyzer monitors the data, looking for the events you specified in the Trigger windows. When the specified events occur, the logic analyzer responds according to the selections you made in the Setup and Trigger windows.

Repetitive-Run Mode

In Repetitive-Run mode, the logic analyzer keeps acquiring data until you click Stop or until it fulfills the stop conditions.

You can use Repetitive-Run mode as follows:

- To observe the same waveform or listing for a period of time
- To stop after a set number of acquisitions for a period of time and look for differences

After each acquisition you can have the logic analyzer do the following tasks:

- Save the system or module setups and data

- Export the data to separate files or overwrite the same file
- Compare the acquisition data against data in another LA module or a saved LA module file
- Open a file or execute a set of defined tasks after the acquisition is complete. For example, you can execute a command such as sending email or paging you that the logic analyzer has stopped.

Select Repetitive Properties from the System menu to select the different options for the Repetitive mode.

NOTE. You can minimize the time between acquisitions by specifying compare conditions under the LA Setup window, deleting (not just minimizing) all data windows, and then starting the logic analyzer. When the logic analyzer fulfills the compare conditions, you can create a new data window with the New Data Window wizard to view the data.

Related Topics

[Setup Window \(see page 13\)](#)

[Compare Definition Dialog Box \(see page 190\)](#)

Specifying Multiple Acquisitions

1. From the System menu, click Repetitive Properties.
2. Select the Stop After check box.
3. Enter the number of acquisitions to execute.
4. Click OK.

TIP. Click the button to the right of the Run button to set the repetitive run mode.



To improve the response time of the logic analyzer during repetitive operations, you can delete any unnecessary data windows. This allows the logic analyzer to spend time on processing data rather than redrawing data in data windows.

Related Topic

[Repetitive Properties Dialog Box \(see page 191\)](#)

Opening a Document or Program after an Acquisition

1. From the System menu, click Repetitive Properties.

2. Select the Open: <Name of program or document> check box.
3. Click the Open button.
4. In the Open dialog box, click the Browse button to locate the document or application you want to open.
5. Specify any arguments and file locations, if needed.
6. Click OK to close the Open dialog box.
7. Click OK.

Calibrating Modules

1. Save your system before performing calibration. Calibration invalidates acquired data.
2. From the System menu, click Calibration and Diagnostics.
3. Click the Self Calibration tab.
4. Select the module or modules to calibrate.
5. Click Run. Calibration time can take up to several minutes to complete, depending on the type of modules chosen.

If the calibration fails, refer to the service manual for that particular module or contact your local Tektronix service center.

Related Topics

[Self Calibration Properties \(see page 371\)](#)

[Technical Support \(see page 1\)](#)

Calibrating Probes

For a DSO module:

1. In the System window, click the DSO Setup button.
2. Click a channel tab.
3. Attach probes that you want to calibrate to the PROBE COMPENSATION output through a dual-male BNC adapter and Probe-cal Adapter board.

NOTE. *Passive or unknown probes are not calibrated.*

1. Click Probe Cal.

2. Click Calibrate Chan < n> to calibrate the probe attached to the selected channel.
3. Click Calibrate All to calibrate the probes attached to all channels that can be calibrated.

The channel number and probe calibration status for each probe are displayed in the dialog box.

For an external oscilloscope:

Use the Vertical menu on the external oscilloscope to calibrate the probes.

Developing Custom Color Schemes

1. With the System active, go to the System menu and select Options.
2. Click the Colors tab.
3. Select the element you want to change.
4. Select a color for the element. This will be the default.
5. Click Save Scheme. Enter a new name; you cannot save changes to a default color scheme.
6. Click OK.


Related Topic


[Changing Defaults \(see page 86\)](#)

Comparing Data

You can use the logic analyzer to compare acquired data against saved reference data. You can use color to highlight the results in the Listing or Waveform window.

1. Decide which signals to compare and set them up in groups in the LA Setup window.
2. Go to a Listing or Waveform window and add the groups to that window.

3. In the Listing or Waveform window, click  Define Compare (the Define Compare button) in the TLA toolbar to open the Define Compare dialog box.
 - a. Select the Enable Data Compare box, if it is not already selected.
 - b. Select a reference data source from the list, or click Add Data Source to add reference data from a saved file.
 - c. Select one of the following compare regions:
 - Use All Data Possible: Compares the entire acquisition against the reference data.
 - Partial Data around Ref Begin: Compares a specified number of samples after the beginning of the reference data.
 - Partial Data around Ref Trigger: Compares a specified number of samples before or after the trigger mark in the reference data.
 - d. Specify the data alignment.

Align the data with respect to the beginning of memory or to the trigger location. You can also specify the number of samples to offset the data.
 - e. Click OK to close the dialog box.
4. Click  (the properties toolbar button) in the Listing or Waveform window.
 - a. Select either the Listing or Waveform window tab.
 - b. Select the color for memory differences (Acq != Ref).

If you want to display that the acquisition data equals the reference data (memory equalities), select Acq = Ref and the desired color.
 - c. Click OK.
5. Click Run to perform an acquisition.

The compare data (memory differences or equalities) will be highlighted in the regions of the display window that you specified in the Define Compare dialog box.

Follow these guidelines for comparing data:


- Acquisition modules and reference modules must be the same width.

- You cannot compare glitch data, disassembler group data, MagniVu data, or oscilloscope data. You can compare only the main logic analyzer data.
- Although you cannot compare disassembler group data directly, you can compare the raw, nondisassembled data by using groups as defined in the Setup window. Add the groups in a Listing window.

Specifying Stop Conditions for Compare Operations

1. From the System menu, click Repetitive Properties.
2. Select the Stop criteria.
3. Specify additional parameters as needed for your application, such as executing a program after meeting stop conditions or saving information to a file.
4. Click OK.

NOTE. Select *Enable Compare* in the *Compare Definition* dialog box to stop the acquisitions when the compare conditions are met.

TIP. Click the button to the right of the Run button to set the repetitive run mode. 

Related Topic

[Repetitive Properties Dialog Box \(see page 191\)](#)

New Histogram Data Page

Use to specify the data source for the new Histogram window:

1. Click Data from an LA in the system to use data from a currently installed logic analyzer or from a saved logic analyzer.
2. Click the No data to create an empty Histogram window; you can add the data later.
3. Click Next.

New Histogram Options Page

Use to select specific data options for the new Histogram window:

1. Select a data source. The data source can be from a logic analyzer or from saved TLA files.

2. Select a group or counter/timer from the selected logic analyzer.
3. Click Next.

Related Topic

[Data Source Properties \(see page 221\)](#)

Select Axis Type

Overview

Use this page to specify the X-axis and Y-axis for the empty Graph window.

Behavior

For either axis, you can select one of the following:

- **Sample Number.** Use the drop-down list to select Begin, Trigger, or End.
- **Row Number.** This selection is the same as the Sample Number but does not include any gaps due to filtering.
- **Timestamp.** Use the drop-down list to select either an Absolute timestamp or the Previous timestamp.
- **Value of Group.** Use the drop-down list to select an Address, Data, or Control group.

The axis label will be the same as the type that you specify from the above list. After you set up the window, you can use the [Graph Window \(see page 307\)](#) to further define each axis of the graph.

NOTE. *Once you have specified the axis type using the wizard, you cannot change the type without creating a new Graph window.*

To Use

Select the axis type and click the Finish button to finish the wizard.

Related Topic

[Add Series Dialog Box \(see page 313\)](#)

Axis Type Wizard

Overview

Use the Axis Type wizard to add a data series to an empty graph window. The wizard appears when you click the Add Series toolbar button or select Add Series from the Edit menu in an empty Graph window. It allows you to select the data source for the Graph window and then specify the X-axis and Y-axis types for that data source.

In an empty Graph window, click the Add Series toolbar button to start the wizard.

New Source Data Page

Use to specify the data for the new Source window:

1. Select An existing Listing window to use data from an existing Listing window.
2. Select the No Listing window to create an empty Source window; you can add the data later.
3. Click Next.

New Source Options Page

Use to select specific data options for the new Source window:

1. Select a Listing window, data source, and listing column. The Source window will use the data from these selections.
2. Select a symbol file to use.
3. Click Next.

Related Topic

[Source Data Properties \(see page 239\)](#)

Setting Up Analog Feeds

Analog feeds functionality allows you to assign multiple digital signals to the DSO using multiple analog channel outputs. You can access the iCapture Analog Feeds dialog box from the LA Setup window by clicking the iCapture button.

NOTE. *Analog feeds functionality is only available in TLA7000 Series logic analyzer modules.*

Use the following procedure to set up this functionality from the iCapture Analog Feeds dialog box.

1. Select the LA module containing digital signals that you want to route to a DSO from the Physical Module list box.
2. Select an analog output attenuation level.
3. From the Select Signals list box, choose a method of displaying signals: By Group - lists groups as defined in the Setup window. By Probe - lists signals by the probe name. By Name - alphabetically lists signals by their name.
4. Select the digital signals that you want to route through analog outputs to the DSO.
5. Select the check boxes associated with the analog outputs, located under Analog Feed Cycling.
6. Click a button in the Analog Feed Cycling group box to assign the selected signals to the selected outputs, in groups equal to the number of outputs selected.
7. Optionally select the check box labeled Exclude signals assigned to unchecked outputs.
8. Click the System Inter-probing button to display the System inter-probing dialog and select the DSO inputs to which the signals will be routed.

Related Topics

[Setting Up System Inter-Probing \(see page 65\)](#)

[Signal Signal Routing \(see page 67\)](#)

[Analog Feeds \(see page 29\)](#)

Setting Up System Inter-Probing

Only TLA7000 Series logic analyzer modules, DSO modules, and external oscilloscopes can be displayed in the System Inter-probing dialog box. If a TLA7000 Series logic analyzer module and either a DSO module or external oscilloscope exist but none are available, the dialog box can still be opened but is empty.

1. From the System menu, click System Inter-probing to display the System Inter-probing dialog box.
2. Click an LA, DSO or external oscilloscope channel button to begin the inter-probing connection. The system responds by changing the cursor to the shape of a probe connector.
3. Click a channel button corresponding to the physical connection at the opposite end of the probe to establish the probe connection. Upon establishing an inter-probing connection, the cursor changes back to its normal shape.

NOTE. *If you click the mouse anywhere other than on a legal endpoint for the connection, no connection is made and the cursor changes back to its normal shape.*

4. Repeat steps 1 and 2 for each LA module-to-DSO module or external oscilloscope physical connection that you want to specify.

TIP. *To remove an inter-probing connection, click one of its endpoints or use the Delete button.*

You can use the Clear All button to remove all probe connections simultaneously.

Routing a Signal from a Logic Analyzer Output

1. From the Waveform window, right-click the waveform label associated with the oscilloscope to which you want to route the digital signal.
2. In the right-click menu, click Route from LA. The Route from LA dialog box displays.
3. Click the Select Channel button. The Select Signal dialog box displays.

NOTE. *The Select Signal button is disabled when you have not previously setup logical system inter-probing connections. To perform this setup, click the System Inter-probing button.*

4. Select a method of displaying signals.
5. Choose the signal to route to the oscilloscope.

Related Topic

[Routing a Signal to an Oscilloscope Input \(see page 66\)](#)

[Setting Up System Inter-Probing \(see page 65\)](#)

Routing a Signal to an Oscilloscope Input

1. From the Waveform window, right-click the waveform label associated with the logic analyzer module from which you want to route the digital signal.
2. In the right-click menu, click Route to DSO. The Route to DSO dialog box displays.
3. Select the logic analyzer signal that you want to send to the oscilloscope.

NOTE. No logic analyzer signals display in the Route to DSO dialog box until you set up logical system inter-probing connections. To perform this setup, click the System Inter-probing button.

4. When you want to add a new waveform to the waveform window containing the rerouted signal, select the check box labeled Add waveform to view.

Related Topic

[Routing a Signal from a Logic Analyzer Output \(see page 66\)](#)

[Setting Up System Inter-Probing \(see page 65\)](#)

Changing Signal Routing

Use the following procedure to change digital signal routing from either a DSO channel input or a logic analyzer output that is displayed on the Waveform window.

Routing from the LA

1. Right-click the label of the DSO channel waveform.
2. From the right-click menu, click Route from LA. The Route from LA dialog box displays.
3. Click the Select Signal button. The Select Signal dialog box displays.

NOTE. The Select Signal button is disabled when you have not previously setup logical system inter-probing connections. To perform this setup, click the System Inter-probing button.

4. Select a method of displaying signals.
5. Choose the signal to rout to the oscilloscope.

Routing to the Oscilloscope

1. Right-click the label of the LA waveform.
2. From the right-click menu, click Route to DSO. The Route to DSO dialog box displays.
3. Click the Select Signal button. The Select Signal dialog box displays.

NOTE. The Select Signal button is disabled when you have not previously setup logical system inter-probing connections. To perform this setup, click the System Inter-probing button.

4. Select a method of displaying signals.
5. Choose the signal to route to the oscilloscope.

Related Topics

[Setting Up System Inter-Probing \(see page 65\)](#)

[Setting Up Analog Feeds \(see page 64\)](#)

Edit Protocol Files

1. To open a protocol for editing from the Stack Definition pane, do one of the following:
 - Double-click the protocol you want to edit. The protocol is displayed in the Protocol Editor pane.
 - From the Protocol Editor pane select the File menu, and then select Open Protocol.
2. Type your changes directly in the Protocol Editor pane.

TIP. Use the Replace function to help you locate and replace text.

1. From the Protocol Editor pane, select the Edit menu and then select Replace.... The Replace dialog box opens.
2. Type the text you want to change in the Find What text box.
3. Type the new text in the Replace with text box.
4. If the text is case sensitive, select the Match case check box.
5. Click Replace to replace the next instance of the text in the Find What text box with the new text in the Replace text box. Click Replace All to replace all instances. Click Find Next to find the next instance but not replace the text.

Editing the Protocol Window Properties

Follow these steps to edit the Protocol Window Properties:

1. Place the cursor in the Protocol window and click View > Properties to display the Protocol Properties dialog box.
2. Click the About Data tab to view the current data source information.
3. Click the Protocol window tab to adjust the display settings of the Protocol window.
4. Click the Column tab to adjust the column settings in the Packet view.
5. Click the Color tab to adjust the color settings in the Decode View.

6. Click the Marks tab to adjust the properties for the marks that you placed.
7. Click the Frammer options tab to set the port address and packet identifier for framing the packets.
8. Click OK.

Related Topic

[Protocol Window Properties \(see page 275\)](#)

Edit Relations Between Protocols

1. In the Stack Definition pane, right-click the protocol for which you want to edit the relations. The Protocol shortcut menu appears.
2. From the Protocol shortcut menu, select Define Relations. The Protocol Setup dialog box appears, displaying the Relations tab.
3. To remove any protocol from the Current Relations list, select the protocol, and then click the Remove button.
4. To add any protocol to the Current Relations list, select the protocol you want to add from the Available Protocols, and then click the Add button.
5. Click Up or Down to determine the order of the relations, and then click OK.
6. If you specify the Type of Relation as Single Parameter or Raw Data, enter hexadecimal values preceded by 0x or binary values preceded by 0b. For example, to enter the 3 in hexadecimal, enter 0x3; to enter it in binary, enter 0b0011.

Edit Stack Files

1. To open an existing stack file, select Open Stack from the File menu.
2. In the Open dialog box, select the stack to open and click OK.
3. To see how to edit the stack files, click the following links:

[To Replace a Protocol from the Stack \(see page 294\)](#)

[To Remove a Protocol in the Stack \(see page 292\)](#)

[To Define a Protocol in the Stack \(see page 292\)](#)

[To Modify Relations Between the Protocols in the Stack \(see page 69\)](#)

Editing a Tool on the Tools Menu

1. From the System menu, click Options and then click the Tools tab.
2. In the User-Defined Tools box, select the tool that you want to edit.
3. Perform one or more of the following actions:
 - To move the tool up one position in the menu, choose the Move Up button.
 - To move the tool down one position in the menu, choose the Move Down button.
 - To change the menu text, command line (tool path and file name), command-line arguments, or the initial directory, click the Edit button and then type the new information in the appropriate text box.
 - To specify a letter in the menu title as an accelerator key, precede that letter in the Menu Name box with an ampersand. The first letter in the title is the accelerator key by default.

Related Topics

[Removing a Tool from the Tools Menu \(see page 70\)](#)

[Adding a Tool to the Tools Menu \(see page 134\)](#)

Removing a Tool from the Tools Menu

To remove a tool from the Tools menu:

1. From the System menu click Options, and then click the Tools tab.
2. In the User-Defined Tools box, select the tool you want to delete and click the Delete button.

Related Topics

[Editing a Tool on the Tools Menu \(see page 70\)](#)

[Adding a Tool to the Tools Menu \(see page 134\)](#)

Editing Symbol Files

You can edit any symbol file supplied by the support package. It is best to make a copy of the file and then edit the copy.

NOTE. *Do not change signal assignments. If you do so, the disassembler may not function correctly.*

Most support packages provide two types of ASCII symbol files: pattern and range. Make a copy of the type you need and edit it using an editor.

NOTE. *Save the symbol file as ASCII. If saved in any other format, the symbol file cannot be used.*

Related Topics

[Symbol Tables \(see page 341\)](#)

[Color Properties \(see page 377\)](#)

Exporting Data from a Protocol Window

Follow these steps to export data from Protocol window

1. Place the cursor in the Protocol window, and select Export... from the Data menu to display the Export Data dialog box.
2. In the Export Data dialog box, specify the file name and location that you want to export data to.
3. Under Export Range, select either Packet View or Decode View, and then select the export range.
4. If you want to include the Hex View, select the Append Hex View check box.
5. If you selected Packet View, you should also select the export options, including the delimiters you want to use.
6. Click Save.

Related Topic

[Export Data Dialog Box \(see page 278\)](#)

Exporting Data from a Listing Window

1. In the System window, click the Listing window icon.
2. From the File menu, click Export Data.
3. In the Export Data dialog box, select the folder you want to export data into.
4. Enter a name for the exported file.
5. Click Options to specify the format options.
6. Select one of the Export ranges.
7. Click Save.

NOTE. Use a Tab or Semicolon as a delimiter when exporting data from a Listing window. If a Space or Comma is chosen as a delimiter, the timestamp values are erroneously exported from the Listing window in picoseconds without units.

Exporting Histogram Data

1. In the System window, click the Histogram data window icon.
2. From the File menu, select Export Histogram.
3. In the Export Data dialog box, select the folder you want to export data into.
4. Enter a name for the exported file.
5. Determine how many rows of data you want to export and click the appropriate button.
6. Click Save.

NOTE. Click Options to change the field delimiters or to change how the bounds will be exported.

Loading a Saved System

1. From the File menu, click Load System.
2. Navigate to the directory where you saved your system.
3. Select the file and click Open.

NOTE. The saved system must exactly match the current system in order for all parts of the system to load. If the saved system does not match, you can load parts of the saved system using the Load System Options dialog box.

The current version of software can load files saved under older versions of software.

Related Topics

[Saving a System \(see page 78\)](#)

[Load System Options Dialog Box \(see page 363\)](#)

Loading Portions of a Saved System

1. From the File menu, click Load System.
2. Navigate to the folder where the saved file is located.
3. Click Options. The Load System Options dialog box opens.
4. Click a Saved System module and drag the module to a compatible module in the current system. Reference data sources and data windows in the saved system load automatically.

NOTE. *The icon itself will not drag. The cursor will change to a drag cursor. If you attempt to drag a saved module to an incompatible current module the cursor changes to a no drop shape.*

5. Click OK.

Related Topic

[Open Options \(see page 363\)](#)


Loading a Saved Module

1. From the System window, click the Setup or Trigger button of the destination module.
2. From the File menu, click Load Module.
3. Navigate to the directory where you saved your file.
4. Select the file name and click Open.
5. Select the module you want to load in the Load Module dialog box.
6. Click Load.


Adding a Data Source to an Active Listing Window

1. In the System window, click the appropriate data window icon.
2. From the Edit menu, click Add Column.
3. Select the Data Source or click Add Data Source if the source is not listed.

4. Select the columns and click Add.
5. Click Close.

TIP. Click  to open the dialog box.

Loading an LA Module with Saved Trigger Settings

1. In the System window, click the LA Trigger button.
2. Click the  button, located in the trigger toolbar.
3. From the Load LA Trigger dialog box, click the Browse button to search for the trigger file. Then click a filename from the displayed file listing to show all modules in the selected file.

NOTE. Alternatively, you can enter the path name to the desired trigger program.



4. Click the Open button to select the file. The Load LA Trigger dialog box lists the modules included in the saved file. You can select only the module that contains the LA trigger settings.
5. Click Load.

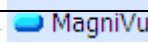
When a trigger program is not compatible with the current setup (for example, different groups), perform the following adjustments, when necessary:

- Groups that are used in group events are matched by name. If a match cannot be found, the event defaults to the first available group.
- Restored values are left-truncated or left-extended with don't cares (XXXX) for word events; left-extended with zeros for range events.
- Channels used in channel events are matched by name. If a match cannot be found, the events are matched by probe name. If a match is still not found, the trigger event defaults to the first available channel.
- Restore event values that are set by group (for example, Setup & Hold or Word events) by matching groups by name. Groups in the current module that are not found in the trigger program being loaded are assigned default settings. Settings for groups not defined in the current module are discarded.
- The Arm Module actions that cannot be present because of target module conflicts are changed to Do Nothing actions.
- Word event definitions are merged with existing word event definitions. If word event definitions have the same names as the current word event definitions then the new definitions will replace the previous ones.
- Channel groups in word events are matched by name. The restored values are left truncated or left extended with don't care values.

- Channel groups in word events in the current module, that are not found in the trigger program being loaded, are assigned default don't care values.
- Settings for groups in word events not defined in the current module are discarded.
- Transition event definitions are merged with existing transition event definitions. If transition event definitions have the same names as the current transition event definitions, then the new definitions will replace the previous ones.
- Channel groups in transition events in the current module, that are not found in the trigger program being loaded, are enabled by default.
- Settings for groups in transition events that are not defined in the current module are discarded.

Adding MagniVu Data to a Listing or Waveform Window


1. In the System window, click the appropriate data window icon.
2. From the toolbar, click  (the Add Column icon) or  (the Add Waveform icon).
3. From the Data Source list, click [MagniVu \(see page 368\)](#). If the source you want is not listed, click Add Data Source and add the one you want to the list.
4. Select the required columns or waveforms.
5. Click Add. The column or waveform appears in the data window.
6. Click Close.

TIP. To quickly display MagniVu data, click  in the data window. Click the button again to remove the MagniVu data from the window.

Loading Source Code Symbol Information


1. In the System window, click the Source data window icon.
2. From the Edit menu, click Properties.
3. Click the Data properties tab.
4. Click the Symbol File button to locate a symbol file in the file system.
5. Choose the symbol file.
6. Verify that the file information in the File Contents box shows that the Type is Range.
7. Click Open.
8. Click OK.

The new symbol file with source code is loaded in the Source window.

TIP. Click  in the Source window to open the Properties dialog box.

You can also load the symbol file from the Symbols dialog box, which is accessible from the System menu.

Loading Filters

1. Click the Filters button in the LA Setup window to open the Filter Definitions dialog box.
2. Click the  Load button to open the Load Filter dialog box.
3. Enter a file name and path in the File Name box for the LA module or system setup that contain the filters that you want to load. You can also click Browse to browse the file system.
4. Select the filter from the Load Filter dialog box.
5. Click the load button to load the filter to the LA module.

Moving to a Mark (Go To)

1. In the System window, click a Waveform or Listing window icon to open the window.
2. From the Edit menu, click Go To.
3. In the Go To dialog box, select the Mark option.
4. Select the mark from the available list.
5. Click OK.

Related Topics

[Adding a Mark to a Data Window \(see page 137\)](#)

[Moving Marks \(see page 137\)](#)

Modifying the Location of Cursors Using the Overview Markbar

1. In the System window, click a waveform or listing icon.
2. Click the cursor handle (in the data area mark bar or the overview mark bar) and drag the mark to the new location.

Related Topics

[Adding a Mark to a Data Window \(see page 137\)](#)

[Sending Stacked Marks to the Back \(see page 137\)](#)

Changing the Position of Cursors

1. In the System window, click the Waveform or Listing window icon.
2. Click the cursor handle (in the data area mark bar or the overview mark bar) and drag the cursor to the new location.

Alternate Method

1. In the System window, click a Waveform or Listing window icon.
2. From the View menu, click Cursor Toolbar if the cursor toolbar is not displayed.
3. Enter the cursor position using either the C1: or C2: cursor spin box.

Related Topics

[Adding a Mark to a Data Window \(see page 137\)](#)

[Moving Marks \(see page 137\)](#)

[Sending Stacked Marks to the Back \(see page 137\)](#)

Print Dialog Box

Use to print the current active window. You can print hard copies of the current data windows and setup windows. Depending on the window, you can specify the print range and the number of copies. You can also print to a file.

When printing Listing, Source, and Waveform windows, you can print data between two marks, such as the trigger point and one of the data cursors. You can cancel printing at any time by pressing the ESC key.

From the File menu, click Print.

Related Topics

[Exporting Data from a Listing Window \(see page 71\)](#)

[Exporting Histogram Data \(see page 72\)](#)

Save Dialog Box

Use to save the system or a logic analyzer module to a file. A system file includes the setup information, the trigger program, and acquired data from all modules (if you select Save Acquired Data). It also includes all data windows and any links to data sources that were added.

A saved logic analyzer module file includes the setup information, the trigger program, and acquired data from the current logic analyzer (if you select Save Acquired Data).

Saved system and module file names have a .tla file name extension. The initial default location for saved files is C:\My Documents.

To open the Save dialog box from the File menu, click either Save System As or Save Module As.

Related Topics

[Loading a Saved System \(see page 72\)](#)

[Loading a Saved Module \(see page 73\)](#)

[Loading an LA Module with Saved Trigger Settings \(see page 74\)](#)

Saving a System

You can save system setups to a [saved system file](#)

Go to the File menu and click Save System.

To save a system and change the name, do the following:

1. From the File menu, click Save System As.
2. Navigate to the directory where you want to save the system.
3. Go to File Name and enter the name you want.
4. Enter any comments you want to include.
5. Select the Save all Acquired Data option if you want the data (including measurement statistics) to be included with the file.
6. Click Save.

Saved System File

A saved system file contains the setup and trigger program information needed to automatically set up the logic analyzer with all installed modules. This file can also contain acquired data from the modules.

Saving a Module

Click a module icon in the System window, then go to the File menu and click Save Module.

To save a module and change the name, do the following:

1. From the File menu, click Save Module As.
2. Navigate to the directory where you want to save the module.
3. Go to File Name and enter the name you want.
4. Enter any comments you want to include.
5. Select the Save all Acquired Data option if you want the data to be included with the file.
6. Click Save.

Saving Repetitive Acquisition Data

1. From the System menu, click Repetitive Properties.
2. Select the Save check box.
3. In the list associated with the Save check box, select Export Data.
4. In the From list, select a data window (this selection is only available with Export Data).
5. Enter a file name or click Browse to specify the location and file name.
6. Select whether to overwrite the existing file or to create a new file for each acquisition.
7. Click OK to close the dialog box and save your settings.

The data will be saved in an ASCII file with the .txt extension under the specified file name.

TIP. Click the button to the right of the Run button to set the repetitive run mode.



Related Topic

[Repetitive Properties Dialog Box \(see page 191\)](#)

Saving Repetitive Module Setups and Data

1. From the System menu, click Repetitive Properties.
2. Select the Save check box.
3. In the list associated with the Save check box, select Save Module and Data.
4. In the From list, select a module.
5. Enter a file name or click Browse and specify the location and file name.
6. Select whether to overwrite the existing file or to create a new file for each acquisition.
7. Click OK to close the dialog box and save your settings.

To improve the response time of the logic analyzer during repetitive operations, you can delete any unnecessary data windows. This allows the logic analyzer to spend time on processing data rather than redrawing data in data windows.

Related Topic

[Repetitive Properties Dialog Box \(see page 191\)](#)

Saving Repetitive System Setups and Data

1. From the System menu, click Repetitive Properties.
2. Select the Save check box.
3. In the list associated with the Save check box, select Save System and Data.
4. Enter a file name or click Browse and specify the location and file name.
5. Select whether to overwrite the existing file or to create a new file for each acquisition.
6. Click OK to close the dialog box and save your settings.

To improve the response time of the logic analyzer during repetitive operations, you can delete any unnecessary data windows. This allows the logic analyzer to spend time on processing data rather than redrawing data in data windows.

Related Topic

[Repetitive Properties Dialog Box \(see page 191\)](#)

Setting the Search Options

The Search Options dialog box defines the action that you want the logic analyzer to take when the search is successful. Access the Search Options dialog box from the Search Options... button in the Define Search dialog box.

To create a new mark upon a successful search:

1. Select the Create new mark at point found check box.
2. Use the existing Mark Name Prefix or enter a new one in the Mark Name Prefix text box.

NOTE. *When a new mark is created, the Mark Name Prefix and a numeric suffix are appended to the mark to ensure that the new mark is unique.*

Searching for Specifics in a Listing or Waveform Window


1. In the System window, click the appropriate data window icon.
2. From the Data menu, click Define Search, and then select Search.
3. From the Search Definition drop-down list, select a saved search definition from the selected data window or type in a new search name.

NOTE. *To add a search definition from another data source, click the Load button to open the Load Search dialog box.*

4. Click Start from the drop-down list to select where you want to start searching and enter the number of occurrences to find in the occurrence text box.
5. Click the Search Options... button to open the Search Options dialog box and select what action you want the logic analyzer to take when the search is successful.
6. Type a Description of the search.
7. From the Search drop-down list, select the data source.
8. Use the drop-down lists under Find to select the criteria for which you want to search.
9. Right-click in the Search event area to open a menu where you can cut, copy, or paste events.

NOTE. You can add and delete sources and enable or disable searches with the short cut edit menu.

10. Click either the Search Fwd or Search Back button. The first occurrence is marked according to how the Search Options are set up.

TIP. Click  to open the dialog box.

Related Topic

[Searching for Source Data \(see page 82\)](#)

Searching for Source Data


In the System window, click the Source data window icon.

1. From the Data menu, click Define Search.
2. Click either All Files to search all files identified by the symbol file, or select This File to search the current file.
3. Enter a text pattern.
4. Click Search Fwd or Search Back. The first occurrence is marked by the active cursor.

NOTE. You can search for any text pattern including leading and embedded white space; trailing white space is ignored.

The search operation follows the order of files as listed in the Source File properties tab. When you search for data in all files, and one or more of the files cannot be found, the Source File Locator dialog box displays, asking you to specify the file location.

If the search is not completed in a set amount of time, the Search Progress dialog box shows the status of the search. You can let the search continue, or you can click Abort to stop the search.

TIP. Click  to open the dialog box.

You can copy text to use as a search pattern from the Source window and paste it into the Source Window Search Definition dialog box.

Related Topic

[Searching for Specifics in a Listing or Waveform Window \(see page 81\)](#)

Searching in the Protocol Window

Follow these steps to search for a data packet or a message pertaining to the packet:

1. Place the cursor in the Protocol window and click Data > Define Search to display the Search dialog box.
2. Select a protocol from a list of protocols. Use the drop-down list to select a protocol.
3. Select the fields to search from the Fields text box.
4. Select operation used to compare field values from the Operation group box.
5. Specify the value and the format for the selected field. The packets that have the specified values are searched for in the Packet View.
6. Select either the << Back or Forward >> button to search before or after the current cursor position.
7. Clear the Enable Fields selection to search the packets based on the selected protocol.

Related Topic

[Protocol Search Dialog Box \(see page 278\)](#)

Setting the Waveform Style

1. In the System window, click the Waveform window icon.
2. Double-click the label of the LA waveform whose options you want to set.

NOTE. *The Options selection is not available if you select a waveform with only one channel.*

3. Click Options.
4. To display LA waveforms as a busform, click Busform.
5. To display LA waveforms as magnitude waveforms, click Magnitude.
6. If you selected Magnitude, select whether you want acquisition data interpreted as signed data.

NOTE. *When you want to interpret the LA waveform as signed data, make sure you select a radix, such as signed decimal, so that the readouts will display the correct information.*

Related Topics

[Setting Waveform Options \(see page 125\)](#)

[Setting the Waveform Limits \(see page 84\)](#)

[Showing Waveform Readouts \(see page 120\)](#)

Setting the Waveform Limits

1. In the System window, click a Waveform window icon.
2. Double-click the label of the waveform whose options you want to set.
3. Click Options.
4. If you want the waveform limits locked to the DSO setup, click Lock to Setup.
5. If Lock to Setup is not checked, set the maximum and minimum value to display in the waveform.

The Max and Min values correspond to the top and bottom of the waveform range.

Related Topics

[Setting Waveform Options \(see page 125\)](#)

[Setting the Waveform Style \(see page 83\)](#)

[Showing Waveform Readouts \(see page 120\)](#)

Determining the Data Source of the Selected Waveform

1. In the System window, click a Waveform window icon.
2. Double-click the label of the waveform about which you want to view attributes.
3. If not selected, select the waveform you want to change from the waveform list.
4. View the source of the waveform from the Source field, which is located in the upper-left corner of the Waveform tab.
5. Click Cancel.

Source is only available when a single waveform is selected.

Related Topics

[Changing Waveform Appearance \(see page 119\)](#)

[Changing the Height of a Waveform \(see page 124\)](#)

[Selecting the Radix of Busform Readouts \(see page 85\)](#)

[Changing the Color of the Selected Waveform \(see page 119\)](#)

[Controlling Whether a Waveform is Visible \(see page 121\)](#)

Selecting the Radix of Busform Readouts

1. In the System window, click a Waveform window icon.
2. Double-click the label of the waveform whose attributes you want to change.
3. If not selected, select from the Waveform list the waveform you want to change.
4. From the Radix list, select the waveform radix.
5. Click OK.

Related Topics

[Changing Waveform Appearance \(see page 119\)](#)

[Changing the Height of a Waveform \(see page 124\)](#)

[Determining the Data Source of the Selected Waveform \(see page 84\)](#)

[Controlling Whether a Waveform is Visible \(see page 121\)](#)

Changing the Zoom Options

You can change the zoom options by changing the system preferences.

To change the zoom options, select Options from the System menu. Under the Preference tab, click the Click & Drag Zoom option to display a drop-down list with the following selections:

- Zoom Immediately
- Show Context Menu
- Show Rectangle Only

Zoom Immediately

This is the default setting. To zoom data in a Waveform window, click and drag the mouse over the area that you want to zoom. The window will automatically zoom after you release the mouse button. If you click and drag from the left to the right, the window zooms to the selection. If you click and drag from the right to the left, the window repeats the previous zoom operation.

When you zoom data in a Listing window, a right-click menu appears after you click and drag the mouse. The right-click menu gives several choices where you can copy text, bitmaps, move cursors to the edge of the rectangle, show, or suppress data.

Show Context Menu

Use this setting to show the right-click menu as soon as you click and drag the mouse over the zoom area. Select one of the choices from the right-click menu.

Show Rectangle Only

Use this setting to draw a rectangle by clicking and dragging the mouse. You can then resize the rectangle so that you can focus on the data of interest. Right-click the mouse to display a right-click menu with several options for handling the data inside the rectangle.

NOTE. *You cannot suppress or show oscilloscope data. The Suppress and show selections only function for logic analyzer data.*

Setting the Default Trigger Tab

1. From the System menu, click Options to display the Options dialog box.
2. Click the Defaults tab.
3. Click the Trigger Window Style field and then select the desired trigger tab from the list box.

Changing the Start-Up Default Settings


1. From the System menu, click Options.
2. Click the Defaults tab.
3. Click the option and change the setting.
4. Click OK.

The settings here are used as defaults when creating a new data window.


Changing the Color of a Graph Mark

1. In the System window, click the Graph data window icon.
2. Double-click the mark that you want to change to open the Marks property page.
3. Click the Color button and select a new color.
4. Click OK.


Changing the Data Series Range

1. In the System window, click the Graph data window icon.
2. Click  in the toolbar of the Graph window to open the Graph window property pages.
3. Click the Series Data tab.
4. Enter new values for the minimum and maximum data range setting for each axis.
5. Click OK.

Changing the Graph Drawing Style

1. In the System window, click the Graph data window icon.
2. Click  in the toolbar of the Graph window to open the Graph window property page.
3. Click the Series Format tab.
4. Select the series type.
5. Use the drop-down lists in the Line section of the property page to change the line style, color, and thickness. Use the drop-down lists in the Data Points area of the property page to change the data point style, color, and thickness. The graphic in the property page provides a brief example of what your changes will look like.
6. Click OK to close the property page.

Changing the Graph Window Properties

1. In the System window, click the Graph data window icon.
2. Click  in the toolbar of the Graph window to open the Graph window property pages.
3. Click the Graph Window tab.
4. Make the changes that you want to the:

- Graticule

Select Show Graticule check box. You can set the color, style, and thickness using the drop-down lists. If you want to hide the graticule, clear the Show Graticule check box.

- X and Y axis

Enter new values for the minimum and maximum data range setting for each axis. If desired, enter an new label for each axis.

- Axis readout

Select the Show Readout check box. Select the color for the readout from the Color drop-down list. If you do not want the readout to display, clear the Show Readout check box.

- Background color

Click the Background color button at the bottom of the property sheet and select a background color.

5. Click OK.

PG Run Properties Dialog Box

The Pattern Generator Run Properties dialog box selections determine if the TLA Start button will start and stop the pattern generator (PG) modules.

The When the TLA Runs list box selections are:

- **Ignore PG.** Do not start or stop the pattern generator modules.
- **Start PG after TLA.** When the TLA start button is pressed the LA modules will start before the pattern generator modules.
- **Start PG before TLA.** When the TLA start button is pressed the pattern generator modules will start before the LA modules.

The two check box options are:

- **Stop PG when TLA stops.** When this box is checked the pattern generator modules will be forced to stop after all LA modules are finished acquiring data.

If this box is not checked the system will wait for all LA and PG modules to complete before indicating that the system (PG and LA modules) are stopped.

- **If repetitive run, restart at each iteration.** Normally the pattern generator will run once and stop even in repetitive mode.

If this box is checked, the pattern generator will be restarted each iteration (Repeat count).

Waveform Properties

Overview

Use the Waveform tab to setup, change, or view the properties of a waveform. The properties displayed in the Waveform tab depend on the waveform that you select in the Waveform window.

Behavior

The following are properties that are common to all waveforms, regardless of whether you choose an individual waveform, a waveform group, or an individual waveform within a waveform group:

- **Waveform:** lists the label that displays in the Waveform window. You can select a different waveform from the list. The list contains the names of all waveforms in the active window. The initial waveform is the selected waveform or the first waveform if no waveform is selected. The list box is empty if multiple waveforms are selected.
- **Height:** controls the height of the waveform. Height can range from 10 to 500 pixels. The default height of waveforms is fixed at 18 pixels.

To Use

To change waveform properties from the System window, click a Waveform window icon, double-click the label of a waveform, and then click the Waveform tab.

Related Topics

[Individual Waveform Properties \(see page 209\)](#)

[Group Waveform Properties \(see page 208\)](#)

[Individual Waveform Properties within a Waveform Group \(see page 211\)](#)

Waveform Window Properties

Overview

Use to format the current Waveform window.

- Acq != Ref determines if data [differences](#) will be displayed during data compare operations.
- Acq = Ref determines if data [equalities](#) will be displayed during data compare operations.

Behavior

To display differences and equalities, define a valid data reference and enable the comparison.

- Show Graticule determines if a graticule will be displayed. Graticule color determines the color of the graticule.
- Show Violations determines if glitches or setup and hold violations will be displayed. Violation color determines the color of glitches and violations. Glitches or setup and hold violations must be stored before they can be displayed.
- Background determines data area background color for the active data window.

- Color Scheme determines the named color scheme for the active data window.
- Snap cursors/marks to transition edges allows you to place the cursors or marks quickly on a transition edge.

When you select a new color scheme, the color of all waveform data window elements reset to the new color scheme (All color selections in this dialog box will change to the colors defined by the new color scheme).

To Use

To change Waveform Window properties from the System window, click a Waveform window icon, double-click a waveform label and then click the Waveform Window tab. You can also access this tab from the toolbar, menu bar, or from the right-click menu on the waveform label.

Related Topics

[Changing Waveform Window Properties \(see page 120\)](#)

[Waveform Tab \(see page 88\)](#)

[Changing Waveform Properties \(see page 119\)](#)

[Waveform Options \(see page 216\)](#)

[Color Settings \(see page 377\)](#)

[Making Violations Visible \(see page 122\)](#)

Selecting the Sample Size: A General Approach

To make the most effective use of memory, use sample suppression to limit the number of samples acquired.

From the LA Setup window follow these steps to set the sample size:

1. Select the desired number of samples.
2. Click the Suppress button in the toolbar to activate the Sample Suppression dialog box.
3. Make a selection in the dialog box to limit the number of samples initially displayed after each acquisition.

For example: Show 1024 Samples around trigger.

During data acquisition memory is filled up to the selected depth (maximum in this case). After acquisition only 1024 samples will be delivered to all data displays.

If the acquisition you are looking for is not visible, show more samples using the Show dialog box or right-click menu in the Listing or Waveform window. This can be done without reacquiring data.

If too much data is delivered to all data displays, remove samples using the Suppress dialog box or right-click menu in the Listing or Waveform window.

Related Topics

[Suppress Samples Dialog Box \(see page 200\)](#)

[Sample Storage depth \(see page 27\)](#)

Storing LA Module Setup and Hold Violations

1. Select the LA Setup window.
2. Select synchronous sampling or if you have previously loaded a support package, use the Custom sampling selection.
3. Select the Samples + Violations (Setup and Hold) storage option.

NOTE. *When you select the maximum acquisition length under the Samples storage and you change to Setup and Hold storage, the logic analyzer changes the acquisition length to half the original.*

4. If you use synchronous sampling, adjust the global setup T_s and hold T_h values in the sampling area of the LA Setup window.
You can adjust the values of individual groups or signals in the Signal Sampling screen.
5. If you use Custom Sampling, adjust values in Signal Sampling screen.

Setting up Transitional Storage for the LA Module

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click Transitional from the Storage list box, located in the trigger toolbar.
4. Click the Change Detect button.
5. From the Change Detect dialog box, select the groups in which you want the logic analyzer to detect a change in state for the transitional storage.
6. Click OK.

When you start the logic analyzer, the logic analyzer module will begin storing data when it detects a change in any one of the selected groups.

Conflicts are possible when groups share signals. Conflicts are noted in the boxes. Conflicts do not prevent acquisition. Value settings closer to the top of the list override later settings.

On TLA7000 Series logic analyzer modules and TLA5000B Series mainframes, the first transition state indication is always false. On TLA700 Series logic analyzer modules, the first transition state always shows a transition indication (even if there are no probes plugged in). One way to avoid triggering on a

false first transition indication is to set the first state to "If Anything Goto Next State". This will consume a state, but will ensure that there is no false triggering on the first transition indication.

TIP. *An easy method for setting up a transitional storage trigger is to load one of the EasyTrigger programs (Store on all transitions, Store on signal transitions, or Store on group transition). If necessary you can modify the program from the PowerTrigger tab.*

Storage Definition Dialog Box

Overview

Use this dialog box to define the condition when storage occurs after Conditional storage is selected in the Trigger window toolbar. This dialog box defines an If event the same way as does the Clause definition dialog box. The Then action stores a sample; this action cannot be changed.

To Use

To open the Storage definition dialog box, perform the following:

1. From the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Set Storage to Conditional and click the If/Then button in the Storage state.

A few things to keep in mind when using this dialog box:

- Counter and timer events are not available in storage definitions.
- Violation Storage is not available for Prime signals.
- Storage Definition is not available from the EasyTrigger tab.
- Snapshot events are not available in storage definitions.

Related Topic

[Clause Definition Dialog Box \(see page 163\)](#)

[LA trigger Event Table \(see page 167\)](#)

[Storage Selection for an LA Module \(see page 140\)](#)

Triggering a Logic Analyzer Using the Trigger Tab Page

Use the Trigger tab page to quickly add one of the following trigger programs:

- Channel edge
- Bus Value
- Glitch
- Trigger on Anything
- Channel Value
- Multi-Group Value
- Setup and Hold

To use the Trigger tab page, do the following:

1. From the data window, click the Trigger tab page.
2. Drag a trigger from the Add Trigger group box to the waveform that you want to trigger on.
3. Edit the trigger expression under Trigger Details in the Trigger tab page.
4. Click Run.

Adding a State to a Trigger Program

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. From the Edit menu, click Add State.

NOTE. *The state is added after the currently selected state. If a state is not selected, then the state is added as the last state.*

TIP. *Add State is also available on the toolbar of the Trigger window.*



Defining If/Then Clauses in a Trigger Program

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the If/Then button of the clause you want to edit.
4. Make changes in the If/Then clause dialog box.
5. Click OK.

On TLA7000 Series logic analyzer modules and TLA5000B Series mainframes, the first transition state is always false. On TLA700 Series logic analyzer modules the first transition state always shows a transition indication (even if there are no probes plugged in). One way to avoid triggering on a false first transition indication is to set the first state to "If Anything Goto Next State". This will consume a state, but will ensure that there is no false triggering on the first transition indication.

If transitions are used as events, there will be a false indication of a transition. The first state needs to be set up as "If Anything Then Next State."

Defining Multiple Transition Recognizers

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the state clause you want to define.
4. In the event definition drop-down list box, click Transition and specify the related transition condition.
5. Optionally, from the Clause Definition dialog box, click the Add button to specify additional transition events.
6. Click the Define Transition button to display the Transition Definition dialog box.
7. Click the Add button to add a new transition definition and specify the groups in which transition events will be monitored.

NOTE. You can add as many transition event definitions as you want within the Transition dialog box. However, you can only use eight events of any kind within an event clause. The events that are selected when you close the Transition dialog box are used in the event clause.

8. In the event definition drop-down list box, click Transition and specify the related event action.
9. Optionally, click the Define Transition button to define a unique transition name.

NOTE. You can define up to 16 trigger events and two counters or timers for a given trigger setup.

TIP. An easy method for defining multiple transition recognizers is to load the Trigger on Nth occurrence of edge from the Easy Trigger programs. If necessary, you can modify the program from the PowerTrigger tab.

Related Topics

[Add Transition Dialog Box \(see page 159\)](#)

[Transition Definition Dialog Box \(see page 158\)](#)

Selecting Signals for an LA trigger Channel Event

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the If/Then button of the clause you want to edit.
4. Select Channel in the first box of the If section.
5. Click the Select Channel button.
6. Do one of the following:
 - Double-click the desired group in the Select Signals dialog box and select the signal.
 - Click By Probe and select the signal.
 - Click By Name and select the name of the signal you assigned in the Setup window.
7. Click OK.

Selecting When to Store Data

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Specify the desired storage mode in the Storage box, which is located in the LA Trigger dialog box toolbar. Refer to [Data Sample Storage Selections \(see page 150\)](#) for definitions of the storage modes.

NOTE. *This functionality is only available when defining trigger programs using the PowerTrigger tab. Otherwise, the Storage box is disabled.*

Positioning the Trigger

1. In the System window, click the LA Trigger button.
2. Position the trigger using one of the following methods available from the trigger toolbar:
 - Move the Trigger Pos slider until the desired percentage of data before the trigger shows in the text field.
 - Enter the desired percentage in the text field.
 - Use the controls in the list box to reach the desired percentage.

Triggering an LA Module on a Glitch

NOTE. Only Asynchronous sampling allows glitch selection. Glitch storage is not available with Prime signals.

1. In the TLA toolbar, click the Trigger button for your module.
2. Click the EasyTrigger tab.
3. Locate the Trigger on glitch EasyTrigger program under Simple Events.
4. Click the Define Glitches button.
5. Select the groups for which you want to detect a glitch.
6. Click OK.

TIP. An easy method for setting up a glitch trigger program is to drag-and-drop a glitch trigger into the Waveform or Listing window from the Trigger tab page. Then make any necessary changes to the trigger expression, under Trigger Details, to set up the trigger.

Triggering an LA Module on a Setup and Hold Violation

NOTE. Setup and hold violation storage is unavailable when using Prime signals. Violations can only be selected with synchronous or custom sampling. Custom is available when a support package is loaded.

1. From the LA Setup window select synchronous sampling. If you have previously loaded a support package, the Custom selection is correct.
2. Open the LA Trigger window.
3. Click the EasyTrigger tab.
4. Select the Trigger on group setup & hold violation program from the Simple Event list.
5. Click the Define Violation button to display the Setup and Hold Event dialog box.
6. Select the groups for which signals that you want to detect a setup and hold violation.
7. Enter the values in the corresponding group setup and hold boxes.
8. Click Close.

TIP. An easy method for setting up a setup and hold violation trigger program is to drag-and-drop a Setup and Hold trigger from the Trigger tab page into a Waveform or Listing window. Then make any necessary changes to the trigger expression, under Trigger Details, to set up the trigger.

Related Topics

[Loading an LA with Saved Trigger Settings \(see page 74\)](#)

[Using the Trigger Tab Page \(see page 92\)](#)

Triggering an LA Module on Timer Values

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the clause you want to edit.
4. Click Timer in the If section of the Clause Definition dialog box.
5. Select the specific Timer.

NOTE. Counters and Timers share resources. For example if Counter 1 is selected, Timer 1 is not available as an action or event.

6. Select the operation.
7. Select the value.
8. Click either Trigger or Trigger All Modules in the Then section of the Clause Definition dialog box. Selecting Trigger will trigger the current module; Trigger All Modules triggers all modules, which is the same as a system trigger.
9. Click OK.

Related Topic

[Triggering an LA Module on Counter Values \(see page 97\)](#)

Triggering an LA Module on Counter Values

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.

3. Click the corresponding If/Then button of the clause you want to edit.
4. Click Counter in the If section of the Clause Definition dialog box.
5. Select the specific counter.

NOTE. *Counters and Timers share resources. For example if Counter 1 is selected, Timer 1 is not available as an action or event.*

6. Select the operation.
7. Select the value.
8. Click either Trigger or Trigger All Modules in the Then section of the Clause Definition dialog box. Selecting Trigger will trigger the current module; Trigger All Modules triggers all modules, which is the same as a system trigger.
9. Click OK.

Related Topic

[Triggering an LA Module on Timer Values \(see page 97\)](#)

Triggering an LA Module on an Internal Signal

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the clause you want to edit.
4. Click Signal in the If section of the Clause Definition dialog box.
5. Select the signal.
6. Select the signal logic level.
7. Click either Trigger or Trigger All Modules in the Then section of the dialog box. Selecting Trigger will trigger the current module; Trigger All Modules triggers all modules, which is the same as a system trigger.
8. Click OK.

Related Topics

[Internal Signaling \(see page 154\)](#)

[Trigger Action Signal Restrictions \(see page 155\)](#)

[Module Arming Using an LA Module \(see page 154\)](#)

Triggering an LA Module on a Word Event

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the clause you want to edit.
4. Click Word in the If section of the Clause Definition dialog box.
5. Click the Define Word button.
6. Enter the word values for each group.
7. Click the box to the left of the group to enable or disable each group.
8. Click OK.
9. Click either Trigger or Trigger All Modules in the Then section of the Clause Definition dialog box. Selecting Trigger will trigger the current module; Trigger All Modules triggers all modules, which is the same as a system trigger.
10. Click OK.

TIP. Define multiple word events by clicking Add in the Word Definition dialog box and assigning a name to the definition. You can then select the definition from the Clause Definition dialog box directly.

An easy method for setting up a word event trigger is to load the Trigger on word value EasyTrigger program. If necessary you can make modifications to the program from the PowerTrigger tab.

Triggering an LA Module on a Snapshot Event

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the clause you want to edit.
4. In the Clause Definition dialog box, If section, select Snapshot from the event selection list.
5. Click the Define Snapshot button.
6. Click the box to the left of the group to enable or disable each group.
7. Click OK.

8. In the Then section of the Clause Definition dialog box click either:
 - Trigger - to trigger the current module.
 - Trigger All Modules - to trigger all modules, which is the same as a system trigger.
9. Click OK.

Triggering on Group Data Values

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the clause you want to edit.
4. Click Group in the If section of the Clause Definition dialog box.
5. Select the group.
6. Select the operation for word recognition. Valid selections are = and Is Not.
7. Enter the value to recognize.
8. Click either Trigger or Trigger All Modules in the Then section of the Clause Definition dialog box. Selecting Trigger will trigger the current module; Trigger All Modules triggers all modules, which is the same as a system trigger.
9. Click OK.

TIP. An easy method for setting up a group data value trigger is to load the EasyTrigger programs Trigger on group value. If necessary you can make modifications to the program from the PowerTrigger tab. Alternatively, you can drag-and-drop a Multi-Group Value trigger from the Trigger tab page onto a waveform. Then make the necessary selections in the trigger expression, under Trigger Details, to set up the trigger.

Related Topic

[Using the Trigger Tab Page \(see page 92\)](#)

Triggering on Group Data Ranges

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the clause you want to edit.
4. Click Group in the If section of the Clause Definition dialog box.

5. Select the group.
6. Select the operation for range recognition. Valid selections are:
<, <=, >, >=, Is In and Is Not In.
7. Enter the values for the range.
8. Click either Trigger or Trigger All Modules in the Then section of the dialog box. Selecting Trigger will trigger the current module; Trigger All Modules triggers all modules, which is the same as a system trigger.
9. Click OK.

TIP. An easy method for setting up a group data value trigger is to load one of the EasyTrigger programs (Trigger on group value outside of a range or Trigger on group value within a range). If necessary you can make modifications to the program from the PowerTrigger tab. Alternatively, you can drag-and-drop a Multi-Group Value trigger from the Trigger tab page onto a waveform. Then make the necessary selections in the trigger expression, under Trigger Details, to set up the trigger.

Related Topic

[Using the Trigger Tab Page \(see page 92\)](#)

Triggering an LA Module on a Single Channel

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the clause that you want to edit.
4. Select Channel in the If section of the Clause Definition dialog box.
5. Click the Select Channel button to display the Select Channel dialog box.
6. Select both the selection method and the signal that you want to trigger on, and then click OK.
7. Select the operation.
8. Select the logic level.
9. Click either Trigger or Trigger All Modules in the Then section of the dialog box. Selecting Trigger will trigger the current module; Trigger All Modules triggers all modules, which is the same as a system trigger.
10. Click OK.

TIP. An easy method for setting up a single channel trigger is to load one of the EasyTrigger programs (Trigger on signal low/high or Trigger on signal transition edge). If necessary you can make modifications to the program from the PowerTrigger tab. Alternatively, you can drag-and-drop a Channel Value or Channel Edge trigger onto a waveform from the Trigger tab page. Then make the necessary selections in the trigger expression, under Trigger Details, to set up the trigger.

Related Topics

[Signal Selection \(see page 157\)](#)

[Using the Trigger Tab Page \(see page 92\)](#)

Triggering an LA Module on a Group

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the clause you want to edit.
4. Click Group in the If section of the Clause Definition dialog box.
5. Select the group.
6. Select the operation for group recognition.
7. Select the group value or pattern.
8. Click either Trigger or Trigger All Modules in the Then section of the dialog box. Selecting Trigger will trigger the current module; Trigger All Modules triggers all modules, which is the same as a system trigger.
9. Click OK.

TIP. An easy method for setting up a group trigger is to drag and drop a Multi-Group Value trigger from the Trigger tab page onto a waveform or listing window. Then make the necessary selections in the trigger expression, under Trigger Details, to set up the trigger.

Alternatively you can load one of the EasyTrigger programs (Trigger on group value or Trigger on group transition). If necessary, you can modify the program from the PowerTrigger tab.

Related Topic

[Using the Trigger Tab Page \(see page 92\)](#)

Arming other Modules from an LA

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the clause you want to edit.
4. Select the event type in the If section of the Clause Definition dialog box.
5. Click Arm Module in the Then section of the Clause Definition dialog box.
6. Select the module you want to arm.
7. Click OK.

Related Topics

[Internal Signaling \(see page 154\)](#)

[Trigger Action Signal Restrictions \(see page 155\)](#)

[Module Arming Using an LA \(see page 154\)](#)

Triggering other Modules from an LA

1. In the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click the corresponding If/Then button of the clause you want to edit.
4. Select the event type in the If section of the Clause Definition dialog box.
5. Click [Trigger All Modules](#) in the Then section of the Clause Definition dialog box.
6. Click OK.

NOTE. All trigger actions in the trigger program must be the same and can be either Trigger or Trigger All Modules.

TIP. An easy method for triggering all modules is to load the EasyTrigger program Trigger all modules when A occurs. If necessary, you can modify the program from the PowerTrigger tab.

Trigger All Modules

Trigger All Modules is the same as a system trigger.

Troubleshooting LA PowerTrigger Programs

If you have problems with your PowerTrigger program, try these suggestions:

- Look at the trigger clauses. The clauses, although appearing to be evaluated simultaneously, have precedence, and are given priority from top to bottom in the active state. Therefore, you will not trigger when you think you should if you have a program like the following examples.

If Anything

Increment Counter 1

Else If Counter 1 > 5

Then Trigger

A trigger will never occur because the first clause is always true.

If counter 1 > 5

Then Reset Counter 1

And Increment Counter 2

Else if Counter 1 <= 5

Then Increment Counter 1

Else if Counter 2 > 10

Then trigger

A trigger will never occur because counter 1 will either be >5 or <=5.

- Check for a bad probe lead: go to the Setup window and check the activity indicators for activity at the probe tip.
- When setting up to trigger on group ranges, check that the signals in the group are assigned in descending order in the Setup window. For example:

A3(7), A3(6), A3(4) or A3(7-0) is correct.

A3(6) A3(7), A3(4) or A3(0-7) is incorrect.

The values triggered on in the incorrect example will not be correct in the data windows. For example, if the incorrect group above is in the range of F0 to 70 the only possible values to cause a trigger would be F0 and 70. Values in the range of F0 through 70 will not cause a trigger.

- Check that the probe order is correct. Verify that you have the correct probe or probe leads connected.
- Check the progress of the trigger program in the Status monitor to see which state may be causing the trigger problem.
- On the TLA7000 Series logic analyzer modules and the TLA5000B Series mainframe, the first transition state indication is always false. On TLA700 Series logic analyzer modules the first transition state always shows a transition indication (even if there are no probes plugged in). One way to avoid

triggering on a false first transition indication is to set the first state to "If Anything Goto Next State". This will consume a state, but will ensure that there is no false triggering on the first transition indication.

Related Topics

[Status Monitor Dialog Box \(see page 366\)](#)

[Troubleshooting Triggering \(see page 338\)](#)

Timer Event Target Value Limit

The maximum value that can be entered as an event value limit is 2,000,000 seconds, which is about 23 days.

Actual ending timer values up to the full 104-day limit can be read from the hardware and displayed in the status monitor.

The 23-day limit is imposed by the user interface in the If/Then dialog.

Data Measurement Types and Definitions: Count

There are four LA data measurement types or categories. Click the following buttons for definitions and algorithms of each measurement.

Count measurements

Pattern Match

$$C_{Pat} = \sum_{n=1}^N X_n$$

The number of occurrences of a specified group value.

Where C_{Pat} is the total number of occurrences of the matched pattern in the gated region.

X_n is the pattern to match.

The total number of occurrences of the specified group value in the gated region is displayed in the Value column.

Count measurements

Edge Count

$$C_{Edge} = \sum_{n=1}^N X(LH)$$

The total number of signal edges (positive and negative).

Where C_{Edge} is the Edge Count.

X(LH) is an edge represented by a transition from Low to High.

X(HL) is an edge represented by a transition from High to Low.

The total number of signal edges in the gated region is displayed in the value column.

Cycle Count

$$C_{Cycle} = \sum LHLH$$

$$C_{Cycle} = \sum HLHL$$

The number of signal cycles in the specified region. One cycle is defined as the time period between rising edge to next rising edge or from a falling edge to the next falling edge.

Where C_{Cycle} is the cycle Count.

Positive Pulse count

$$C_{Pulse}^+ = \sum LHL$$

The number of positive pulses.

Where C_{Pulse}^+ is the Positive Pulse Count.

The total number of positive pulses in the gated region is displayed in the Value column.

Negative Pulse Count

$$C_{Pulse}^- = \sum HLH$$

The number of negative pulses.

Where C_{Pulse}^- is the Negative Pulse Count.

The total number of negative pulses in the gated region is displayed in the Value column.

Count measurements

Violation Count

$$C_{Violation} = \sum V$$

The number of group setup/hold violations or number of glitch violations in the gated region. Setup/Hold violations are only available for synchronous sampling. Glitch violations are only available for asynchronous sampling.

Where $C_{Violation}$ is the Violation Count.

The total number of violations in the gated region is displayed in the Value column.

Violation Rate

$$R_{Violation} = C_{Violation} / N(S)$$

Ratio of the number of group setup/hold or glitch violations (depending on the sampling) per number of seconds.

Where $R_{Violation}$ is the Violation Rate. $C_{Violation}$ is the number of violations in the gated region. $N(S)$ is the number of seconds.

The rate at which violations occur in the gated region expressed as a fraction of total number of violation/seconds is displayed in the Value column.

Data Measurement Types and Definitions: Jitter

There are four LA data measurement types or categories. Click the following buttons for definitions and algorithms of each measurement.

Jitter measurements

Period Jitter

$$\Delta P_n = P_{n+1} - P_0$$

The timing variation in the period measured in seconds.

Where ΔP_n is the Period Jitter.

P_{n+1} is the Period of the (n +1) Cycle.

P_0 is the Period of the First Cycle.

The difference of the Maximum period measured minus the Minimum period measured in the gated region is displayed in the Value column. This is also called peak-to-peak period jitter.

Cycle-to-Cycle Jitter

$$\Delta P_n = P_{n+1} - P_n$$

The timing variation in the period in adjacent cycles measured in seconds.

Where ΔP_n is the Cycle-to-Cycle Jitter.

P_{n+1} is the Period of the (n+1) Cycle.

P_n is the Period of nth Cycle.

The difference of the Maximum cycle-to-cycle jitter minus the Minimum cycle-to-cycle jitter in the gated region is displayed in the Value column. This is also called peak-to-peak cycle-to-cycle jitter.

Data Measurement Types and Definitions: Minimum/Maximum

There are four LA data measurement types or categories. Click the following buttons for definitions and algorithms for each measurement.

Minimum/Maximum measurements

Group Min

$$\text{Min}(X) = \text{Lowest Value of Group}$$

The minimum value the group values. The lowest value of the group in the gate region is displayed in the Value column.

Group Max

$$\text{Max}(X) = \text{Highest Value of Group}$$

The maximum of the group values. The highest value of the group in the gated region is displayed in the Value column.

Data Measurement Types and Definitions: Time

There are four LA data measurement types or categories. Click the following buttons for definitions and algorithms of each measurement.

Time measurements

Frequency

$$F_n = 1/P_n$$

Frequency is the reciprocal of the period of a signal measured in hertz (Hz).

.Where F_n is the Frequency and P_n is the Period.

The average frequency of all cycles in the gated region is displayed in the Value column.

Period

$$P_n = T_{n+1} - T_n$$

Time it takes for a complete cycle in the gated region. Reciprocal of frequency, measured in seconds (s).

Where P_n is the Period and T is the point in time of a Transition (Low to High or High to Low).

The average of all period measurements in the gate region is displayed in the Value column.

Positive Pulse Width

$$W_n^+ = T_n^{HL} - T_n^{LH}$$

The time duration of a positive pulse (low-high-low) measured in seconds (s).

Where W_n^+ is the Positive Pulse Width.

T_n^{HL} is the point in time of the Transition from High to Low.

T_n^{LH} is the point in time of the Transition from Low to High.

The average of all positive pulse width measurements in the gated region is displayed in the Value column.

Negative Pulse Width

$$W_n^- = T_n^{LH} - T_n^{HL}$$

The time duration of a negative pulse (high-low-high) measured in seconds (s).

Where W_n^- is the Negative Pulse Width.

T_n^{LH} is the point in time of the Transition from Low to High.

T_n^{HL} is the point in time of the Transition from High to Low.

The average of all negative pulse width measurements in the gated region is displayed in the Value column.

Time measurements

Positive Duty Cycle

$$D_n^+ = W_n^+ / P_n$$

The ratio of the positive pulse to the total signal period expressed as a percentage (%).

Where D_n is the positive duty cycle.

W_n^+ is the Positive Pulse Width.

P_n is the Period.

The average of all positive duty cycle measurements in the gated region is displayed in the Value column.

Negative Duty Cycle

$$D_n^- = W_n^- / P_n$$

The ratio of the negative pulse to the total signal period expressed as a percentage (%).

Where D_n is the negative duty cycle.

W_n^- is the Negative Pulse Width.

P_n is the Period.

The average of all negative duty cycle measurements in the gated region is displayed in the Value column.

Channel-to-Channel Delay

$$ChD_n = Average \left(\sum_I^N T_n^{Destination} - T_n^{Source} \right)$$

Time delay between the edges of two different signals measured in seconds (s).

Where ChD_n is the Channel to Channel Delay. $T_n^{Destination}$ is the Transition (location of the type of edge selected in the UI using the drag and drop action) of the Destination Channel. This is indicated as the second channel selected during the measurement setup. The average of the delay from the first edge of the type selected as the source to the first edge of the type selected as the destination plus the delay from the second edge of the type selected as the source to the second edge of the type selected as the destination...plus the delay from the nth edge of the type selected as the source to the nth edge of the type selected as the destination for each pair of edges in the gated region is displayed in the Value column.

T_n^{Source} is the Transition of the Source channel. This is indicated as the first channel selected during the measurement setup. The delay from the first edge of the type selected for each edge in the gated region is displayed in the Value column.

Using LA Data Measurements

To use LA data measurements follow these steps:

1. Add a new measurement.
 - a. To select a measurement, right-click on the signal or group header.
 - b. Select Add LA Measurement and then select the measurement you want to add.

OR

- a. Select the Setup page in the Measurements tab.
- b. Click and drag a measurement item from Add Measurements to a signal or group in the data window.

If a measurement is dropped onto any group, the measurement will be applied to all the signals in the group.

2. Define gating.
 - a. Select gating type from the Gate drop-down list on the Setup page.
 - b. Select Display to analyze the samples in the visible display.
 - c. Select Cursors to analyze the samples between Cursor 1 and Cursor 2.
 - d. Select Full Record to analyze all the samples.
3. Select mode for analysis.
 - Select the Accumulate check box to add results to the existing measurement results.

OR

- Clear the Accumulate check box to clear previous results and update with new results.

NOTE. *The results are accumulated when you click Run but not when you click Recalc.*

4. Analyze the data.

Results and statistics are displayed in the Statistics page. To get new data, click Run. To recalculate measurements, click Recalc.

Related Topic

[Adding Dual Channel Measurements \(see page 112\)](#)

Statistic Definitions for Data Measurements

Statistic	Definition
Average	The sum of the results calculated in the gated region divided by the total number calculations. The measurement is expressed in the same units as the data. $\mu(X) = \bar{X} = \frac{1}{N} \left(\sum_{n=1}^N X_n \right)$
Min	Minimum of all the results calculated in the gated region. $\text{Min}(X) = \text{Lowest Value of } X$

Statistic	Definition
Max	Maximum of all the results calculated in the gated region. $Max(X) = \text{Highest Value of } X$
Standard Deviation	A statistical measure of the spread of results. It is a measure of variability expressed in the same units as the data. $StdDev(X) = \sigma_x = \sqrt{\frac{1}{N} \left(\sum_{n=1}^N (X_n - \bar{X})^2 \right)}$
Population	The number of times the measurement was performed in the gated region. $Population(X) = N$

Note: The samples or data considered for calculating the results for all the above measurements and statistics are bound by the gated region. Gating options include display, cursors, and full record.

Adding Dual Channel Measurements

To add a dual channel measurement using drag and drop, do the following:


1. Select the Setup page from the Measurements tab.
2. Click and drag a measurement item from the Available Measurements list to the first channel. While dragging to the first channel there is a tool tip that says on which channel the measurement is to be dropped.
3. Drop the measurement button on the first channel. The tool tip will change to indicate the first channel selected and that a second channel needs to be selected.
4. Click the channel header to select the second channel.

Copying Filters

1. Click the Filters button in the LA Setup window to open the Filter Definitions dialog box.
2. Double click the LA module that contains the filter that you want to copy. The filter list will expand to show all filters attached to the selected LA module.
3. Select the filter and click the Copy button in the dialog box.
4. Select the LA module to which you want to copy the filter.
5. Click the Paste button.

TIP. You can also open the Filter Definitions dialog box from the Listing or Waveform window by selecting Filter Definitions... from the Data menu.

Creating Filters

1. Click the Filters button in the LA Setup window to open the Filter Definitions dialog box.
Logic analyzers with filters attached to them have an expand and collapse button to the left of the name.
2. Select an LA module in the dialog box.
3. Click the New button to attach a new filter to the module.
4. Click the  button to open the Define Filter dialog box where you can define the filter contents.

TIP. You can also open the Filter Definitions dialog box from the Listing or Waveform window by selecting Filter Definitions... from the Data menu.

Opening Filters

1. Click the Filters button in the LA Setup window to open the Filter Definitions dialog box.
2. Locate and double-click the LA module that contains the filter that you want to open.
3. Select the filter.
4. Click the Properties button to view and edit the filter contents in the Define Filter dialog box.

TIP. You can also open the Filter Definitions dialog box from the Listing or Waveform window by selecting Filter Definitions... from the Data menu. Alternately, you can right-click a column label in the Listing window or a waveform label in the Waveform window to display a right-click menu where you can access the filters.

Adding Reference Data to a Listing Window

1. Open a Listing window.
2. Right-click anywhere inside the Listing window header area or data area and select Add Column.
3. In the Add Column dialog box, click the Add Data Source button.
4. Click Browse and search for the appropriate data source.
5. After the source is added, select it in the Add Column dialog box.

6. Select the desired columns and then click Add.
7. Click Close.
8. If necessary, select the new column in the header and drag the column to the desired location.

TIP. To delete a column from the Listing window, select the header and press the Delete key on the keyboard.

Adding an Opcode Mark

Occasionally, you might want to change the way the disassembler interprets a cycle type. Mark Opcodes synchronizes the disassembler. Install a support package to use this feature.


1. In the System window, click the Listing window icon.
2. From the Edit menu, click Mark Opcode.
3. The pointer changes to a cross hair pointer.
4. Move the pointer to the data sample where you want to place an Opcode and click.
5. The Mark Opcode Dialog Box opens.
6. Double-click the Opcode type.

TIP. Move the pointer to the desired data sample and click the right mouse button. Select Mark Opcode.

Showing Qualification Gaps in a Listing Window

1. In the System window, click the Listing window icon.
2. From the Edit menu, click Properties.
3. From the Properties dialog box, click the Listing Window tab.
4. Select Show Qualification Gaps.
5. Click OK.

Gaps are shown in the display with a horizontal gray line above the first data after the gap.


TIP. Click  to open the Properties dialog box.

Showing Glitches in a Listing Window

1. In the System window, click the Listing window icon.
2. From the Edit menu, click Properties.
3. From the Properties dialog box, click the Listing Window tab.
4. Select Show Glitches.
5. Select the glitch color.
6. Click OK.

Glitches are shown in the display by highlighting the character in the sample containing the glitch. In binary data the individual bit containing the glitch is highlighted.


NOTE. *Select Samples + Glitches in the Storage Options box of the Setup window before making an acquisition.*

TIP. Click  to open the Properties dialog box.

To see the individual glitch bit, change the radix of the column to binary.

Going to a Specific Column

1. In the System window, click the Listing window icon.
2. From the Edit menu, click Go To.
3. Click Column.
4. Select the column.
5. Click Go To.

TIP. Click  to open the Go to dialog box.

Creating Symbol Files

1. Open any text processor application. For example: WordPad.

2. Use lines beginning with a number sign (#) to indicate comments.
3. Enter a [Symbol File Header \(see page 117\)](#). Include the #+ characters at the beginning of the line.
4. Enter symbol names and values.

The following is an example of a simple pattern symbol file.

```
# Pattern symbol file
# TSF Format Type Display Radix File Radix
# =====
#+ Version 2.1.0 PATTERN BIN BIN # Symbol Name Pattern Color
# =====
NUL X000 0000 SOH X000 0001 @blue STX X000 0010 @white @red
```

The following is an example of a simple range symbol file.

```
# Range symbol file with function symbols
# TSF Format Type Display Radix File Radix Offset
# =====
#+ Version 2.1.0 RANGE HEX HEX 00000000
#+ Function # Symbol Name Low High
# =====
displayBanner 006035ba 00603675
buildMenus 00603676 006036e5
displayLCDmenu 006036e6 0060372f
```

There are four different types of range symbols. For more information on these range symbols, click the symbol type you are interested in:

[Function Range Symbols \(see page 343\)](#)

[Variable Range Symbols \(see page 344\)](#)

[Source Range Symbols \(see page 345\)](#)

[Color range symbols \(see page 354\)](#)

When you create range symbols avoid defining symbols where the ranges overlap within a given type, such as function or variable because overlapping ranges can produce unexpected results.

Color is an effective means for conveying information that allows you to view your data more efficiently and highlight items of interest. Symbol files provide an easy way to color-code samples in your acquisitions to help distinguish various features of the data. When you set the group radix to Symbolic in the data window property sheet and then pick a symbol file with color information in it, the group values may be displayed in color in a Listing or Waveform window. The foreground color as well as the background color of the data can be set by the symbol file.

For a detailed description of symbol file syntax, see [TLA Symbol File Format \(see page 350\)](#).

Related Topic

[Symbols Dialog Box \(see page 346\)](#)

Symbol File Header

Use header lines to help the logic analyzer identify the contents of a symbol file.

Pattern Symbol File Header

The header line of a pattern symbol table has the following form:

```
#+ <file format version> PATTERN <display radix> <file radix>
```

The file format version contains three fields:

- Major version number
- Minor version number
- Build number (for example, 2.0.0); each field is separated by a period.

The display radix field determines the radix used in the data windows and properties tab controls when there is no symbol defined for a value (valid values for Pattern files are Hex, Oct, or Bin).

The file radix field specifies the radix used for the symbol values in the file; valid values for pattern files are Hex, Oct, or Bin.

Range Symbol File Header

The header line of a range symbol table has the following form:

```
#+ <file format version> RANGE <display radix> <file radix> <offset>
```

The file format version contains three fields:

- Major version number
- Minor version number
- Build number (for example, 2.0.0); each field is separated by a period.

The display radix field determines the radix used in the data windows and properties tab controls when there is no symbol defined for a value (valid values for Pattern files are Hex, Dec, Oct, or Bin).

The file radix field specifies the radix used for the symbol values in the file; valid values for pattern files are Hex, Dec, Oct, or Bin.

The offset is a 32-bit hexadecimal value added to the lower and upper bound of each range symbol read from the file.

Coloring Data

1. From the Systems window, click the Listing window icon.
2. Double-click a column heading to display the Listing Properties dialog box.
3. Change the radix to symbolic.
4. Click the Symbol File button to load a symbol tab with color symbols.

NOTE. You can use this column as a color source for other columns by double-clicking another column in the Listing window. From the Listing Properties dialog box, click Same As in the Color group box.


Related Topics

[Symbol Files \(see page 341\)](#)

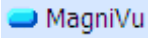
[TLA Symbol File Format \(see page 350\)](#)

Adding a Waveform

1. In the System window, click the Waveform window icon.
2. From the Edit menu, click Add Waveform.
3. Select the Data Source. If the data source you want is not listed, click Add Data Source and select your particular data source.
4. If you are adding a DSO or External Oscilloscope channel, Select the Channel and then click Add. If you are adding a LA channel, select the Group, the Channel, and then click Add.

TIP. Click  on the waveform toolbar to select a waveform to add.

From the front panel or from the waveform toolbar, click  to add external oscilloscope waveforms to the Waveform window.

From the front panel or from the waveform toolbar, click  to add MagniVu waveforms to the Waveform window for those main timebase waveforms that are already displayed in the Waveform window.

Changing Waveform Appearance

1. In the System window, click a Waveform window icon.
2. Double-click the label of the waveform you want to change.
3. If not selected, select the waveform you want to change from the Waveform list.
4. In the dialog box, select the desired waveform Color, Height, Offset, Radix, Readout Options, and Waveform Measurements, and select whether to Use Polarity and Show the Waveform.

NOTE. *Radix and Use Polarity are only available for logic analyzer waveforms. Offset is only available for logic analyzer, DSO, and external waveforms that are included in a waveform overlay. Waveform Measurements are only available for DSO and external waveforms.*

Related Topics

[Selecting the Waveform Color \(see page 119\)](#)

[Changing the Height of a Waveform \(see page 124\)](#)

[Selecting the Radix of Busform Readouts \(see page 85\)](#)

[Determining the Data Source of the Selected Waveform \(see page 84\)](#)

[Controlling Whether a Waveform is Visible \(see page 121\)](#)

[Setting the Overlay Waveform Offset \(see page 209\)](#)

[Setting Up Waveform Measurements \(see page 126\)](#)

Selecting the Waveform Color

1. In the System window, click a Waveform window icon.
2. Double-click the label of the waveform you want to change.
3. If not selected, select the waveform you want to change from the Waveform list.
4. From the Color drop down list, select the desired waveform Color.
5. Alternately, select "Same As" and choose another waveform to use as the color source.
6. Click OK.

Related Topics

[Changing Waveform Appearance \(see page 119\)](#)

[Changing the Height of a Waveform \(see page 124\)](#)

[Selecting the Radix of Busform Readouts \(see page 85\)](#)

[Determining the Data Source of the Selected Waveform \(see page 84\)](#)

[Controlling Whether a Waveform is Visible \(see page 121\)](#)

Displaying Waveform Readouts

1. In the System window, click a Waveform window icon.
2. Double-click the label of the waveform whose options you want to set.
3. Click Options.
4. If you want to show readouts, click Show Readouts.
5. If Show Readouts is checked, set the color of the displayed readouts.

Related Topics

[Setting Waveform Options \(see page 125\)](#)

[Setting the Waveform Style \(see page 83\)](#)

[Setting the Waveform Limits \(see page 84\)](#)

Changing the Waveform Time Scale (Time/Division)

1. In the System window, click a Waveform window icon to open the window.
2. If the Waveform Toolbar is not displayed, click Waveform Toolbar from the View menu.
3. Using the Time/Div control, select the time per division or enter the time/div if the desired value does not appear in the list.

The center of the screen remains fixed when time per division changes. You cannot display less than two horizontal divisions.

Changing Waveform Window Properties

1. In the System window, click a Waveform window icon to open the window.

2. From the View menu, click Properties.
3. Select the Waveform Window tab.
4. In the dialog box, select whether or not to show graticules and violations.
5. Click Show Compare to enable comparison of data.
6. Use the drop-down list to select the color of glitches, graticules, and background.
7. Select the desired Color Scheme.
8. Select whether or not to snap cursors and marks to transition edges.

TIP. To quickly access the Properties dialog box, right-click in the Waveform window and select Properties.

Related Topics

[Making Glitches Visible \(see page 122\)](#)

[Selecting the Glitch Color \(see page 122\)](#)

[Selecting the Graticule Color \(see page 123\)](#)

[Selecting the Background Color \(see page 123\)](#)

[Selecting the Active Data Window Color Scheme \(see page 122\)](#)

Controlling Whether a Waveform is Visible

1. In the System window, click a Waveform window icon.
2. Double-click the label of the waveform whose attributes you want to change.
3. If not selected, select from the Waveform list the waveform you want to change.
4. Click the Show Waveform check box.
5. Click OK.

Related Topics

[Changing Waveform Appearance \(see page 119\)](#)

[Changing the Height of a Waveform \(see page 124\)](#)

[Selecting the Radix of Busform Readouts \(see page 85\)](#)

[Determining the Data Source of the Selected Waveform \(see page 84\)](#)

[Changing the Color of a Waveform \(see page 119\)](#)

Selecting the Violation Color

1. In the System window, click a Waveform window data icon.
2. Double-click a waveform label.
3. Click the Waveform Window tab.
4. Click the violation color from the Show Violations color drop-down list.
5. Click OK.

Related Topics

[Making Violations Visible \(see page 122\)](#)

[Changing Waveform Window Properties \(see page 120\)](#)

Making Violations Visible

1. In the System window, click a Waveform window data icon.
2. Double-click a waveform label.
3. Click the Waveform Window tab.
4. Click the Show Violations check box.
5. Click OK.

When the Show Violations box is checked, violations are displayed on all waveforms.

Related Topics

[Changing Waveform Window Properties \(see page 120\)](#)

[Selecting the Violation Color \(see page 122\)](#)

Selecting the Active Data Window Color Scheme

1. In the System window, click a Waveform window icon.
2. Double-click a waveform label.
3. Click the Waveform Window tab.
4. From the Color Scheme drop down list, select the color scheme for the active window.

Related Topics

[Selecting the Glitch Color \(see page 122\)](#)

[Selecting the Graticule Color \(see page 123\)](#)

[Selecting the Background Color \(see page 123\)](#)

[Changing Waveform Window Properties \(see page 120\)](#)

Selecting the Background Color

1. In the System window, click a Waveform window icon.
2. Double-click a waveform label.
3. Click the Waveform Window tab.
4. From the Background drop down list, select the background color.
5. Click OK.

Related Topics

[Selecting the Glitch Color \(see page 122\)](#)

[Selecting the Graticule Color \(see page 123\)](#)

[Changing Waveform Window Properties \(see page 120\)](#)

[Selecting the Active Data Window Color Scheme \(see page 122\)](#)

Selecting the Graticule Color

1. In the System window, click a Waveform window icon.
2. Double-click a waveform label.
3. Click the Waveform Window tab.
4. From the Show Graticule color list, select the graticule color.
5. Click OK.

Related Topics

[Making Graticules Visible \(see page 124\)](#)

[Changing Waveform Window Properties \(see page 120\)](#)

Making Graticules Visible

1. In the System window, click a Waveform window icon.
2. Double-click a waveform label.
3. Click the Waveform Window tab.
4. Click the Show Graticule check box.
5. Click OK.

When the Show Graticule box is checked, graticules are displayed on all waveforms.

Related Topics

[Changing Waveform Window Properties \(see page 120\)](#)

[Selecting the Graticule Color \(see page 123\)](#)

Changing the Size of a Waveform

Move the cursor over the lower part of the waveform label until it changes from a pointer to a resize cursor. Click the left mouse button and drag to change the size of the waveform.

Alternate Method

1. In the System window, click the Waveform window icon.
2. To open the waveform Properties dialog box, double-click the label of the waveform you want to resize.
3. Click the Waveform tab.
4. Enter the size you want the waveform to be using the Height spin box.
5. Click OK.

This procedure changes the size of the waveform by changing the size of the labeled waveform track. The size of the waveform is also affected by the vertical input range setting.

Jump Over Time Gaps in the Waveform Window

One way to jump over large time gaps in the Waveform window is to create a Listing window of the same data as that of the Waveform window, and then lock the two windows together. As you move the cursor through the Listing window, the cursor in the Waveform window will track and jump over the "dead" or blank space between samples.

Setting Waveform Options

1. In the System window, click a Waveform window icon.
2. Double-click the label of the waveform whose options you want to set.
3. Click the Options button.

From this dialog box you can:

- Set whether to show readouts.
- Set the waveform voltage limits.
- Set the style of logic analyzer waveforms.
- Set the readout color.

Related Topics

[Setting the Waveform Style \(see page 83\)](#)

[Setting the Waveform Limits \(see page 84\)](#)

[Showing Waveform Readouts \(see page 120\)](#)

Viewing an LA Waveform in Magnitude Mode

In the Waveform window, right-click on any LA busform label and select Display Magnitude from the menu.

NOTE. *Magnitude mode is only available for LA busforms. Magnitude mode is not available for DSO or External Oscilloscope waveforms.*

Alternate Method

1. In the System window, click the Waveform window icon.
2. Double-click an LA waveform.
3. Click Options.
4. Click Magnitude.
5. Click Signed data if you want to interpret the acquisition data as signed data.

NOTE. *When you want to interpret the LA waveform as signed data, make sure you select a radix, such as signed decimal, so that the readouts will display the correct information.*

6. If Lock to Setup is not checked, set the maximum and minimum values to display in the waveform.
7. Click OK twice and adjust the time per division to view the waveform.

Scrolling Between Samples in a Waveform Window

1. In the Waveform window, move the pointer to the center of the window.
2. Right-click the mouse and click either Go To Next Sample or Go To Prior Sample.

The Waveform window will scroll to the next or previous sample.

Use this method of scrolling over gaps in the waveform data where there are no data samples due to clock or data qualification.

Selecting the DSO Horizontal Settings

For an external oscilloscope:

1. In the System window, click the External Oscilloscope Setup button, and then select the Setup tab.
2. Use the Depth Limit drop-down list to select the Memory Depth.
3. Use the Horizontal menu on the external oscilloscope to select the remaining horizontal settings.

Setting Up Waveform Measurements

1. In the System window, click the Waveform window icon.
2. Double-click the external waveform label for which you want to set up measurements, and then click the Measurement Setup button.
3. Use the >> button to select up to three measurements from the Available Measurements list.
4. For each measurement type, select a Base-Top Determination method.
5. Select a Reference Level to use for the waveform measurements.
6. From the High-, Low-, and Mid-Reference list boxes, set the desired reference values or click the Set Defaults button to use the default values.
7. Select the Gating method to use for the waveform measurement and then set the related parameters, if applicable.

NOTE. *Acquire waveform data before selecting the Marks gating method and setting related parameters.*

8. Click OK.

Alternate Method

1. In the System window, click the Waveform window icon.
2. From the Waveform window, right-click the waveform label, and then click Add/Delete DSO Measurements.
3. Use the >> button to select up to three measurements from the Available Measurements list.
4. For each measurement type, select a Base-Top Determination method.
5. Select a Reference Level to use for the waveform measurements.
6. From the High-, Low-, and Mid-Reference list boxes, set the desired reference values or click the Set Defaults button to use the default values.
7. Select the Gating method to use for the waveform measurement and then set the related parameters, if applicable.

NOTE. *Acquire waveform data before selecting the Marks gating method and setting related parameters.*

8. Click OK.

Related Topic

[Removing Selected DSO Waveform Measurements \(see page 127\)](#)

Removing Selected Analog Waveform Measurements

1. In the waveform Properties dialog box, select the waveform containing measurements that you want to remove and click the Measurement Setup button.
2. Select the waveform measurement that you want to remove from the Selected Measurements list box and click the Remove button (<<).
3. Click OK.

Related Topic

[Selecting DSO Waveform Measurements \(see page 126\)](#)

Taking Voltage Measurements

1. In the System window, click a waveform icon.
2. Select a waveform.

3. Move Cursor 1 to the location on the waveform that you want to measure.
4. Read the voltage from the C1 readout on the Measurement toolbar.
5. Move Cursor 2 to another location on the waveform that you want to measure.
6. Read the voltage from the C2 readout on the Measurement toolbar.
7. Read the voltage difference between the two waveform locations from the Delta readout on the Measurement toolbar.

Related Topics

[Taking Time Measurements \(see page 129\)](#)

[Taking Delta Time Measurements \(see page 129\)](#)

[Taking Delta Volts Measurements \(see page 128\)](#)

[Taking Logic Analyzer Measurements \(see page 131\)](#)

Taking Delta Volts Measurements

1. In the System window, click a waveform icon.
2. Select a waveform.
3. Move Cursor 1 to the first location on the waveform that you want to measure.
4. Move Cursor 2 to a second location on the waveform that you want to measure.
5. Read the voltage difference between the two waveform locations from the Delta readout.

For LA Busform, the delta shows a sort of bit-wise difference. The delta is formed by treating C1 and C2 values as bit vectors, and setting a 1-bit in the delta at each position where C1 and C2 differ.

For example, in LA Busform if C1 has a group value of 0x8A and C2 has a group value of 0x0C, the delta is 0x86 (the difference is 0x7E).

Related Topics

[Taking Voltage Measurements \(see page 127\)](#)

[Taking Time Measurements \(see page 129\)](#)

[Taking Delta Time Measurements \(see page 129\)](#)

[Taking Logic Analyzer Measurements \(see page 131\)](#)

Taking Time Measurements

1. In the System window, click a waveform window icon.
2. Select a waveform.
3. Move Cursor 1 to the location on the waveform that you want to measure.
4. Read the time from the C1 readout on the Measurement toolbar.
5. Move Cursor 2 to another location on the waveform that you want to measure.
6. Read the time from the C2 readout on the Measurement toolbar.
7. Read the time difference between the two waveform locations from the Delta readout on the Measurement toolbar or from the Delta Time toolbar.

Cursor time is relative to the system trigger.

Related Topics

[Taking Delta Time Measurements \(see page 129\)](#)


[Taking Voltage Measurements \(see page 127\)](#)

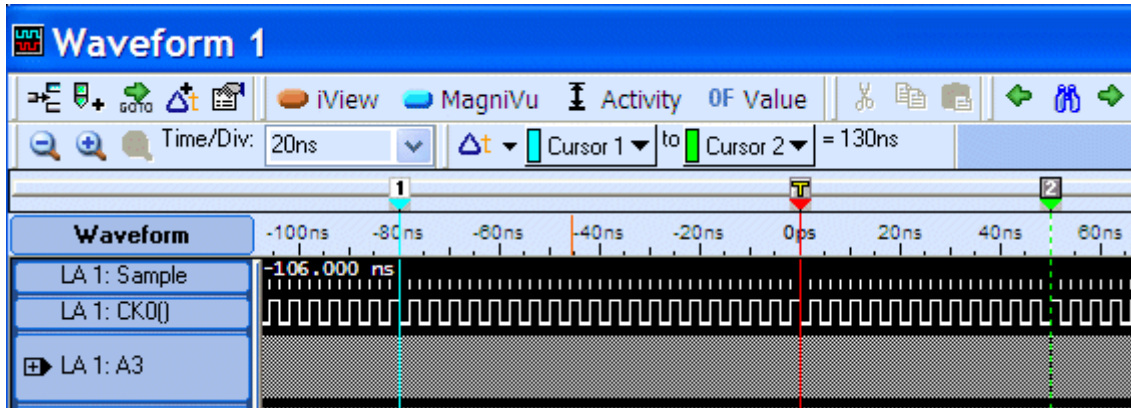
[Taking Delta Volts Measurements \(see page 128\)](#)

[Taking Logic Analyzer Measurements \(see page 131\)](#)

Taking Delta Time Measurements

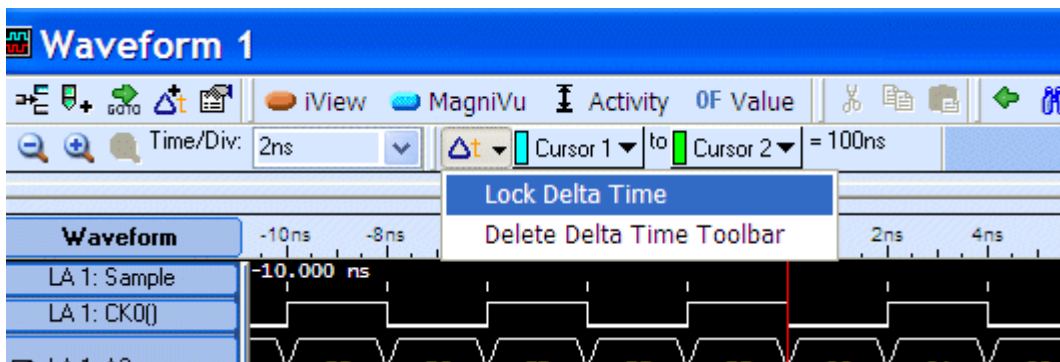
To take a single Delta Time measurement do the following:

1. In the System window, click a waveform window icon.
2. Click the Add Delta Time toolbar button  if the toolbar is not already displayed. Note that you can have as many Delta Time toolbars as you want.
3. Select a waveform.
4. Move Cursor 1 to the location on the waveform where you want the time measurement to start.
5. Move Cursor 2 to another location on the waveform where you want the time measurement to end.
6. Read the time difference between the two locations from the Delta Time toolbar.



To lock the Delta Time do the following:

1. Click the drop-down list on the Delta Time toolbar button.
2. Select Lock Delta Time. This locks the cursors or marks in the Delta Time toolbar with respect to delta time.



Lock Delta time is disabled if either of the following is true:

- The data window is locked
- You have an unmovable measurement point selected for the delta time calculation such as System Trigger

The delta time readout is negative when C2 precedes C1.

For an LA Busform, the delta shows a sort of bit-wise difference. The delta is formed by treating C1 and C2 values as bit vectors, and setting a 1-bit in the delta at each position where C1 and C2 differ.

For example, in LA Busform if C1 has a group value of 0x8A and C2 has a group value of 0x0C, the delta is 0x86 (the difference is 0x7E).

TIP. You can take multiple Delta Time measurements by selecting different measurement points from the Delta Time toolbar. Alternatively, you can add Delta Time toolbars to take additional measurements.

Related Topics

[Taking Time Measurements \(see page 129\)](#)

[Taking Voltage Measurements \(see page 127\)](#)

[Taking Delta Volts Measurements \(see page 128\)](#)

[Taking Logic Analyzer Measurements \(see page 131\)](#)

Taking Logic Analyzer Measurements

1. In the System window, click a waveform icon.
2. Select a logic analyzer group magnitude, group digital, or digital waveform.
3. Move Cursor 1 to the location on the waveform that you want to measure.
4. Read the value of the logic analyzer waveform from the C1 readout on the Measurement toolbar.
5. Move Cursor 2 to another location on the waveform that you want to measure.
6. Read the value of the logic analyzer waveform from the C2 readout on the Measurement toolbar.
7. Read the value of the difference in time between the two waveform locations from the Delta readout on the Measurement toolbar.

Values for logic analyzer digital waveforms are binary 0 or 1.

Values for logic analyzer group magnitude and group digital waveforms are in the selected radix. The delta values are blank if you use a symbolic radix.

Related Topics

[Taking Time Measurements \(see page 129\)](#)

[Taking Delta Time Measurements \(see page 129\)](#)

[Taking Voltage Measurements \(see page 127\)](#)

[Taking Delta Volts Measurements \(see page 128\)](#)


Selecting the Trigger Action and Target for an External Oscilloscope

1. In the System window, click the External Oscilloscope Trigger button.

2. Select the trigger event:
 - Oscilloscope triggers – The external oscilloscope initiates the trigger
 - TLA System trigger – The TLA System trigger initiates the oscilloscope trigger
3. Select the trigger action:
 - Trigger oscilloscope only
 - Trigger TLA too

Adding Reference Data to a Waveform Window

1. In the System window, click the Waveform window icon.
2. From the Edit menu, click Add Waveform.
3. From the Add Waveform dialog box, click the Add Data Source button.
4. Click the Browse button and search for the appropriate source.
5. After the data source is added, select it in the Add Waveform dialog box.
6. Select the desired waveforms.

TIP. Click  to open the Add Waveform dialog box.

Running Extended Diagnostics

1. You may want to save your system before running diagnostics. Extended diagnostics will invalidate your acquired data.
2. Disconnect any probes attached to your instrument.
3. From the System menu, click Calibration and Diagnostics.
4. Click the Extended Diagnostics tab from the Calibration and Diagnostics dialog box.
5. Select the test or set of tests you want to run.
6. Click the option button associated with the Run mode you want to use: One Time, Continuous, or Until Fail.
7. Click Run. If you selected Continuous or Until Fail, you can click on the Abort or Stop buttons to stop the process.

If the diagnostics fail, you can refer to the service manual for the particular product, or contact your local Tektronix service center.

Related Topics

[Extended Diagnostics \(see page 337\)](#)

[Technical Support \(see page 1\)](#)

Defining Reference Data

1. Open a Listing or Waveform data window.
2. Click Define Compare in the TLA toolbar to open the Define Compare dialog box.
3. Click the Enable Data Compare check box.
4. Click the Add Data Source button, then select the reference module from the list or click Browse to search for a saved LA module.
5. Click the Add button to add the data source to the Reference Data Source list.
6. Select a Reference Data Source from the list.
7. Define the parameters in the Compare Data Region.
8. Define the parameters for the Data Source Alignment.
9. Click OK.

Customizing the System Tools menu

You can customize the Tools sub-menu of the System menu by adding, editing, and deleting menu items. You can add frequently used utilities to the Tools menu and run them from within your TLA. To do this, use the Tools page in the Options dialog box (accessed from the System menu).

To add a tool to the Tools menu

As an example, the following procedure demonstrates how to add the Windows WordPad accessory to the Tools menu.

1. From the System menu, click Options and then click the Tools tab.
2. Click the Add button to bring up the Tool Properties Dialog.
3. In the Menu Name text box, type WordPad.
4. In the Command box, browse or type the path and name of the program, for example:
C:\program files\accessories\wordpad.exe
5. In the Arguments text box, type any arguments to be passed to the program.
6. In the Initial Directory box, type the file directory where the command is located.
7. If you want the program to run in a minimized window, check the Run Minimized box.

Once the tool appears on the Tools menu, to run the program, just choose it from the menu or click on its button in the Toolbar.

Enclose quotes around arguments and commands with embedded spaces in the Tool Properties (App Options) dialog box for the commands to work.


Related Topics

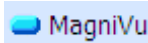
[Editing a Tool on the Tools Menu \(see page 70\)](#)

[Removing a Tool from the Tools Menu \(see page 70\)](#)

Adding a Data Source to an Active Waveform Window

1. In the System window, click a Waveform window icon.
2. From the Edit menu, click Add Waveform.
3. From the Data Source list, select a Data Source or click the Add Data Source button if the source is not listed.
4. Click the Close button.

TIP. Click  to open the dialog box.

To quickly add MagniVu waveforms, click  in the waveform window. Or press the MagniVu button on the front panel. Click the button again to remove the MagniVu data from the window.

Opening a Saved Data Window

1. From the Window menu, click Load Data Window.
2. Click the Browse button to search for the file, or enter a path to the desired file.
3. Enter a Window name or use the default.
4. Once the file is found, click Open.
5. Click OK.

NOTE. *The saved data file must be a saved system file.*

Creating an Overlay Waveform

1. In the System window, click the Waveform window icon.
2. Click the label of a waveform you want to include in an overlay waveform.
3. Drag the waveform label to the second waveform that you want included in the overlay waveform. As you drag the waveform label, a drop marker displays at the left edge of the waveform label. Use this drop marker to determine which waveform you will be selecting to form a waveform overlay.
4. In the Add Overlay Waveform Name dialog box, either enter a name and click OK or simply click OK to use the default overlay waveform name. This name will be prepended to all waveform labels in the overlay waveform.

Related Topics

[Overlay Waveform \(see page 206\)](#)

[Overlay Waveform Properties \(see page 208\)](#)

[Waveform Properties \(see page 88\)](#)

[Changing Waveform Properties \(see page 119\)](#)

Locking Windows

1. In the System window, click a Listing, Waveform, Graph, or Protocol window icon.
2. From the View menu, select Lock Windows.
3. From the Lock Windows dialog box, select which windows to lock together.
4. Click either the No time offsets or Current time offsets option button.
5. If you select No time offsets, select which window is the reference in the Reference Window list.
6. Click OK.

NOTE. *You can lock Listing, Waveform, Graph, and Protocol windows only.*

Aligning Time of Correlated Data Sources

1. In the System window, click a Listing or Waveform window icon.
2. From the Data menu, select Time Alignment.
3. Perform the following steps for each data source:
4. Click OK.

Related Topics

[Adding Reference Data to a Listing Window \(see page 113\)](#)

[Adding Reference Data to a Waveform Window \(see page 132\)](#)

Adding a Mark to a Data Window

From the Edit menu, click Add Mark. The mark appears at the center of the display and has default properties.

NOTE. *You cannot add a mark if you have not acquired data.*

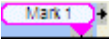
TIP. *Use the right mouse button in the data area to add a mark at a particular sample.*

Related Topic

[Moving Marks \(see page 137\)](#)

Moving Marks

1. In the System window, click the appropriate data window icon.
2. Select the mark handle (in the data area mark bar or the overview mark bar) and drag the mark to the new location.

NOTE. *You cannot move begin, end, or trigger marks. If you drag a mark past the boundary of the data area using a handle in the Data Area Mark bar, the data area scrolls to keep the mark in view. If you drag a mark off-screen using its handle in the Overview Mark bar, the data area scrolls keeping the mark in view. The mark in the Data Area Mark bar indicates that the mark is off screen with an arrow, such as  if you scroll the data window, causing the mark to move off screen.*

Related Topics

[Sending Stacked Marks to the Back \(see page 137\)](#)

[Adding a Mark to a Data Window \(see page 137\)](#)

Sending Stacked Marks to the Back


1. Select a mark you know is on top of another one.
2. From the Edit menu, select Send To Back.

TIP. Click the right mouse button on the mark and select *Send To Back*.

Going to a Specific Mark

1. In the System window, click the appropriate data window icon.
2. From the Data menu, click Go To.
3. Click the Mark option button.
4. Select the mark you want to display.
5. Click the Go To button. The mark appears in the center of the display.

NOTE. When moving between marks in a Source window, the new mark may cause another source file to load in the Source window.

TIP. In the data window, click  to open the name of the dialog box.

Click the right mouse button on a mark and click Go To.

Related Topic

[Adding a Mark to a Data Window \(see page 137\)](#)

Expand Signals

Expand Signals replaces all busforms in the selection set with their component signal waveforms. After the operation is finished, the selection set is modified to contain only the added waveforms.

Related Topic

[Collapse Signals \(see page 139\)](#)

Collapse Signals

Collapse Signals replaces all signal waveforms with their container busform waveforms. Exactly one instance of each busform is added, no matter how many signals are actually present in the selection set. After the operation is finished, the selection set is modified to contain only the added waveforms.

Related Topic

[Expand Signals \(see page 138\)](#)

Removing Data from a Listing or Waveform Window

1. Go to the data window where you want to remove a column or a waveform.
2. Select the column or waveform labels that correspond with the data you want removed.
3. Press the Delete key on the keyboard.

NOTE. *Some columns or waveforms in the display may not be visible. This is generally true if a microprocessor support package is in use. To verify that invisible columns or waveforms exist, open the Properties dialog box and select the Column or Waveform tab. Check whether the Show Column or Waveform option is selected for each column or waveform.*

Removing Waveforms from an Overlay Waveform

1. In the System window, click the Waveform window icon.
2. Click the label of a waveform you want to remove from the overlay waveform.
3. Drag the waveform label to the boundary of the overlay waveform. As you drag the waveform label, a drop marker displays at the left edge of the waveform label. Use this drop marker to position the waveform label outside of the overlay waveform.

Alternate Method

1. In the System window, click the Waveform window icon.
2. Double-click the waveform overlay to display the Waveform properties tab.
3. Click the label of a waveform you want to remove from the Overlay Waveforms list, and then use the << button to remove it from the overlay waveform.

NOTE. You cannot remove waveforms from an overlay waveform using the Waveform properties tab, unless they were added using the Waveform properties tab in the current session. If you attempt to do so, selecting a waveform label from the Overlay Waveforms list prompts the system to disable the << button.

Storage

Overview

Storage refers to storing acquired data in memory. You can select storage modes that filter the data, discard unwanted samples, and store only the data you want.

Storage Qualification

The trigger function block includes storage qualification that looks at sampled data. If the storage conditions are met, a storage qualifier signal enables sampled data to pass into the acquisition memory as qualified data. Any unqualified data samples are excluded.

Storage Options

The Storage Options pane in the Setup window controls storage, using individual storage actions specified in If/Then clauses and the Storage box in the Trigger window toolbar.

NOTE. On TLA7000 Series logic analyzer modules and TLA5000B Series mainframes, the first transition state indication is always false. On TLA700 Series logic analyzer modules, the first transition state always shows a transition indication (even if there are no probes plugged in). One way to avoid triggering on a false first transition indication is to set the first state to "If Anything Goto Next State". This will consume a state, but will ensure that there is no false triggering on the first transition indication.

NOTE. When global storage is set up to use transitional storage, to ensure that a false transition is not recorded, the first state must be "If Anything Then Next State".

Storage Selection for an LA Module

Use the Storage box to select the default [storage \(see page 140\)](#) rules for the module. The default storage setting can be overridden by a storage action in a trigger clause definition.

The Storage box is located in the toolbar of the LA Trigger window.

Related Topics

[Clause Definition Dialog Box \(see page 163\)](#)

[Data Sample Storage Selections \(see page 150\)](#)

Storing DSO Data in Memory

The acquisition memory works like a circular buffer, storing data samples until the entire memory is full. After that, each new data sample overwrites the oldest existing sample.

This process continues until the trigger event is found and the post-trigger delay counter reaches the specified value (based upon the trigger position selection), which stops the acquisition.

During the acquisition you can monitor the progress of the data storage process using the Status Monitor.

Displaying DSO Data

After storing the data you can display the acquired data in the data windows.

In the Waveform window the data is shown as analog waveforms.


In the Listing window the data is shown as a series of voltage values.

Waveform Cursors

Overview

Cursors in a waveform data window are vertical lines that mark time points in the data record. The two cursors are always present, although they may be off-screen. Each cursor has a handle in the Data Area Mark bar and in the Overview Mark bar.


To Use

To select a cursor or other data mark, click its handle in the Data Area Mark Bar or Overview Mark Bar. Selecting a mark changes its color. To edit a data mark, select the mark and then use the Edit menu. You can select a cursor or user mark handle and drag the mark to a new location. You can drag a cursor past the boundary of the data area using a handle in the Data Area Mark bar. The data area scrolls to keep the cursor in view. If you drag a cursor off-screen using its handle in the Overview Mark bar, the data area scrolls to keep the mark in view. The cursor in the Data Area Mark bar indicates that the cursor is off screen with an arrow, such as  if you scroll the data window, causing the cursor to move off screen.

Behavior

When you open a Waveform window, cursor C1 is active. The name of the active cursor is displayed in bold text in the cursor bar. The active cursor is also displayed as a dashed line. Selecting a cursor handle or changing the cursor position using the cursor bar readout makes the cursor the active cursor. Changing the delta time readout changes the position of the inactive cursor.

Cursor time values that fall within an acquisition gap are forced to the closest valid display point. When you place cursors on both sides of an acquisition gap, the displayed delta value is greater than the time between the two data points.

If you drag a cursor past the boundary of the data area using a handle in the Data Area Mark bar, the data area scrolls to keep the cursor in view. If you drag a cursor past the boundary using its handle in the Overview Mark bar, the data area scrolls to keep the cursor in view. The cursor in the Data Area Mark bar indicates that the cursor is off screen with an arrow, such as  if you scroll the data window, causing the cursor to move off screen.

Time per Division

Use the Time/Division dialog box to select the horizontal time per division scaling for the current Waveform window. The list contains standard time per division values in a 1, 2, 5 sequence. You can also enter a non-standard time per division value in the Custom box. This can be any number between the minimum and maximum values shown in the list. The default value is the current setting.

You can change the time per division from a Waveform window or from the Time/Division dialog box. To access the Time/Division dialog box from the View menu, click Time/Div.

Related Topic

[Changing the Time per Division \(see page 120\)](#)

Trigger Tab Page

Overview

The Trigger tab page allows you to quickly and easily set up a trigger program. You can drag and drop trigger programs directly from the Trigger tab page to a data window.

Behavior

The Trigger tab page is only active when a data window is active. Each data window has a unique Trigger tab page. After dragging a trigger program to a data window, a trigger expression is displayed under Trigger Details. The Undo Trigger and Redo Trigger buttons apply only to the current trigger and the previous trigger. The Undo Trigger and Redo Trigger buttons are not active until you use a drag and drop trigger setup.

To Use

Click any Trigger program button and drag it to the waveform or listing window. You can edit the trigger expression using the controls on the Trigger tab page. The controls change depending on the trigger expression.

Click the Undo Trigger button to restore the previous trigger setup. Click the Redo Trigger button to restore the current drag-and-drop trigger setup.

Related Topics

[EasyTrigger Programs \(see page 183\)](#)

[Triggering from the Trigger Tab Page \(see page 92\)](#)

Trigger Window

Overview

The primary function of the Trigger window is for constructing a [trigger program](#). You also use the Trigger window to select how and when to store data; and to set the trigger position, which indicates the percentage of samples that are stored after the trigger occurs.

If you have a TLA7000 Series logic analyzer module or a TLA5000B Series logic analyzer, you can additionally use the Trigger window to defer execution of the trigger state machine until the main memory prefill amount is met. In addition, you can set both the MagniVu storage rate and the amount of pretrigger MagniVu data to store.

Behavior

Before you acquire and display data, set up the LA module using the LA Setup and Trigger windows. These windows determine what data will be acquired. Each module has its own Setup and Trigger window, which are set up individually.

Configure the Setup window before the Trigger window, because some of the Setup window settings affect Trigger window selections.

Trigger programs range from the simple to extremely complex. They are the key to logic analyzer operation. One method for setting up a trigger program is to load a trigger program from the EasyTrigger Program list. The trigger list, which is available from the EasyTrigger tab, contains simplified trigger programs that hide many of the program implementation details, allowing you to focus on acquiring and displaying data. You can then either view or alter the trigger program details from the PowerTrigger tab, as necessary. Once you become familiar with trigger programming, you can also create trigger programs without the help of predefined triggers from the EasyTrigger program list.

Click the following selections for further information:

- [LA Triggers](#)
- [Trigger Program Structure for an LA Module](#)

Trigger Window Structure

The Trigger window contains two tabs that summarize the trigger program to varying degrees.

The EasyTrigger tab contains a trigger program list, program controls, and both a description and graphic example of the selected trigger program. The trigger list is categorized by trigger program function. Once you have selected the trigger program that best fits your data requirements, click the trigger program to display the underlying program controls. The program controls are a simplified representation of the selected trigger program that allows you to select events and set event, timer, and counter values.

The PowerTrigger tab contains both an Overview section and a Trigger Detail section. The Overview shows the general structure of the trigger program. The Trigger Detail summarizes activity within individual program states. You can click the If/Then button in the Trigger Detail to access the Clause Definition dialog box, which is where the actual trigger programming takes place.

You can specify which trigger tab is displayed when you start the TLA application. From the System menu, click Options, and then click the Defaults tab. Click the Trigger Window Style field and then select the desired trigger tab from the list box.

Click the following selections for further information:

- [PowerTrigger Overview](#)
- [PowerTrigger Detail](#)
- [Clause Definition Dialog Box](#)
- [Storage Selection for an LA Module](#)
- [Trigger Position for an LA Module](#)
- [MagniVu Trigger Position](#)

- [MagniVu Storage Rate](#)
- [Force Main Prefill](#)

To Use

To open the LA Trigger window, go to the System window and click the LA module Trigger icon.

To load a trigger program, click the EasyTrigger tab and then select a trigger program. The system responds by loading the selected trigger program, replacing the previous window contents with new program controls, program overview text, and a sample trigger graphic.

Related Topics

[Loading an LA with Saved Trigger Settings \(see page 74\)](#)

[LA Setup Window \(see page 13\)](#)

[Adding a State to a Trigger Program \(see page 93\)](#)

[Trigger Programming Tips for an LA Module \(see page 151\)](#)

[System Trigger and Reference System Trigger Relationship \(see page 165\)](#)


LA Triggers

Overview


The trigger identifies when to stop acquiring data. The trigger also identifies the data of interest. The trigger position determines how much data to store before and after the event of interest. Triggers also provide the main reference points in the acquired data. There are two types of triggers: system and module.

System Trigger (Trigger All)

The system trigger is an overriding command to all modules that causes them to stop looking for a trigger, and to acquire their post-trigger data, regardless of whether they have been armed or have fulfilled their own trigger conditions.

The system trigger is the primary reference point for the entire data acquisition. In data windows, timing and location information is relative to the system trigger. The system trigger is identified in the data windows by the large T mark, .

Module Trigger (Trigger)

The module trigger is local to a module. When a trigger occurs, the module continues to acquire data until the specified amount of post-trigger data is acquired, and then stops. Module triggers are identified in the data windows by the small T mark, .

Related Topic

[System Trigger Placement \(see page 148\)](#)

System Trigger

Overview

The system trigger is a global trigger event that forces all untriggered modules or instruments to arm immediately and trigger. Only one system trigger occurs per acquisition. When displaying data, the logic analyzer uses the system trigger as the primary reference point for the acquisition. The data windows show the system trigger and all module or instrument triggers. Module or instrument triggers can be forced by the system trigger or specified by trigger programs.

The system trigger can be generated from any of several sources. Often, the system trigger is specified in a trigger program. Any module or instrument can specify the system trigger as a trigger action (Trigger All Modules). However, modules or instruments can also specify trigger actions other than system triggers, and the system trigger does not have to originate from a module or instrument. The logic analyzer can also accept a system trigger generated by an external source. External system triggers are input through the SYSTEM TRIG IN connector. Regardless of the source of the system trigger, all modules or instruments must respond.

Use the System Trigger dialog box to specify triggering for the overall system.

System Trigger Source

The following options allow you to change the trigger programs of all modules from one location. A brief functional description follows each System Trigger Source option.

- Modules trigger independently - Trigger All Modules and Wait for System Trigger actions in all trigger programs are changed to Trigger actions.
- Systems triggered by this module - The trigger program of the selected module uses the Trigger All Modules action, while all other programs will wait for the system trigger. Use the associated list box to select which module triggers the system.
- System triggered by the first module that triggers - Trigger actions in all programs are changed to Trigger All Modules. If the system has an external oscilloscope, it is set up to wait for the system trigger.
- Modules trigger using the current custom setup - Enabled by default when your trigger setups do not match one of the previous system trigger options.

NOTE. *When you apply a system trigger option, your choice will only affect those modules that currently have a trigger action. Logic analyzer trigger actions include Trigger, Trigger All Modules, and Wait for System Trigger. DSO and external oscilloscope modules are always affected because all of their actions result in triggering the module.*

System Trigger In/Out Connectors

System Trigger In/Out Connector check boxes reflect the current state of the System Trigger In and System Trigger Out connectors, and can be used to both enable and disable them.

If an external scope is enabled, the options chosen in System Trigger Source can affect the values and sensitivity of the System Trigger In/Out check boxes.

Designated System Trigger

It is not uncommon to have a condition in which the acquisition completed, but an explicit system trigger did not occur. (This means that there was neither an external system trigger, nor a system trigger generated internally by a module trigger program.) To ensure that a time reference exists for the acquisition, the logic analyzer must designate a system trigger. If no system trigger is generated during an acquisition (the modules are internally triggered), the logic analyzer designates the latest-occurring module trigger as the system trigger.

NOTE. *If an acquisition does not complete because one or more modules did not receive a trigger or did not complete the post-trigger acquisition, then you can manually stop the acquisition by clicking the Stop button. Clicking the Stop button effectively generates a system trigger and completes the acquisition.*

Internal and External System Triggers

The system trigger is a latched event and it resets to a false state between acquisitions. The external system trigger input uses real-time gating and is only active (capable of latching system triggers) during the actual acquisition period.

The logic analyzer can also send an internally-generated system trigger out to the target system or to other test equipment through the SYSTEM TRIG OUT connector. All the external signal inputs and outputs operate at TTL levels. The connectors are located at the rear of the portable mainframe and at the front of the benchtop mainframe.

Related Topics

[Setting the System Trigger Source \(see page 148\)](#)

[Signal Properties \(see page 375\)](#)

[External Signal Selection \(see page 367\)](#)

System Trigger Placement

Overview

There is only one system trigger. Because the system trigger is the primary reference point for an acquisition, all acquisitions must have a system trigger identified, even if you did not specifically include one in your trigger program.

Behavior


It is not always obvious how the logic analyzer places the system trigger. The logic analyzer determines the system trigger by one of the following:

- A system trigger occurs when the Stop button is clicked.
- Data windows usually have at most one system trigger plus one module trigger per module.
- If a module initiates the Trigger All Modules action, then the system trigger is also the module trigger for that module.
- If only module triggers are generated, then the latest occurring module trigger is designated the system trigger.

Related Topic

[System Trigger and Reference System Trigger Relationship \(see page 165\)](#)

Setting the System Trigger Source

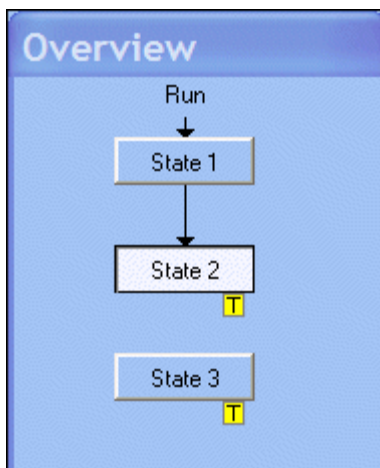
1. From the TLA System toolbar, click the  button. The system responds by displaying the System Trigger dialog box.
2. Select a trigger option from the System Trigger Source group box.
3. If you chose to trigger the system from one module in the previous step, use the associated list box to specify which module is used to trigger the system.
4. If you have an external oscilloscope, the System Trigger In/Out Connector check boxes are disabled and automatically set by the system, based on the system trigger source option that you choose. Otherwise, you can optionally disable these connectors if they are not in use.

NOTE. To save these settings as part of your system setup, perform a [Save System \(see page 78\)](#) operation. This step enables you to reload your saved system settings in a later session.

Related Topic

[System Trigger \(see page 146\)](#)

PowerTrigger Overview



Overview

The Overview portion of the PowerTrigger tab primarily shows how the trigger program flows from one state to the next. The arrow leading to State 1 is marked Run if the module is not armed by another module. The arrow is marked Armed By <name of module> if this module is armed by another module.

Behavior

Arrows indicate the transfer of trigger program control between states. Markers indicate system-level actions that could occur if a clause in that state goes true. Use the trigger overview to check that states are connected. With the exception of State 1, all states must have an arrow leading to them (indicating activation by another state), or they will not execute.

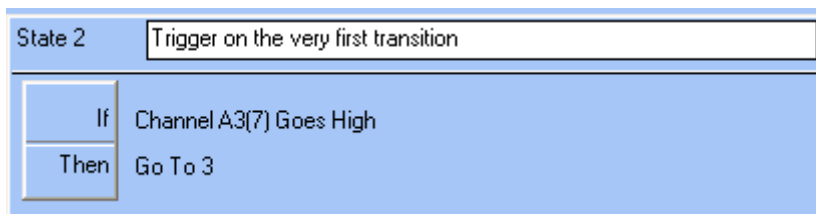
These actions are all defined in the Clause Definition dialog box(es) for each state.

TIP. Double-click the State icon in the Overview to open the top-most clause definition for that state.

PowerTrigger Detail

Overview

The PowerTrigger tab shows a summary of clause settings within the states. To open the Clause Definition dialog box, click the If/Then button. The following dialog box displays:



To Use

You can enter comments in the text box. This feature is controlled by the View menu State Comments selection.

The PowerTrigger tab is part of the LA Trigger window, which you can open by going to the System window and clicking the LA Trigger window icon.

NOTE. *There is a clause summary for each clause defined.*

TIP. *For lengthy trigger programs, click the State icon in the Overview to jump to the corresponding PowerTrigger properties.*

Related Topics

[PowerTrigger Overview \(see page 149\)](#)

[Clause Definition Dialog Box \(see page 163\)](#)

Data Sample Storage Selections

Data sample storage is selected in the LA Trigger window by using the Storage box in the toolbar. The following selections are available for PowerTrigger data sample storage.

- **All.** Store all samples.

You can exclude a sample from being stored by selecting the Don't Store action in the Then statement of the trigger clause. The trigger sample is always stored.

- **None.** Do not store samples.

You can explicitly store a sample by selecting the Store Sample action in the Then statement of the trigger clause.

The following selections generate a special storage clause:

- **Transitional.** Store samples only if one of the specified groups changes state.

To select the groups, click Change Detect in the Trigger Detail.

NOTE. *On TLA7000 Series logic analyzer modules and TLA5000B Series mainframes, the first transition state indication is always false. On TLA700 Series logic analyzer modules, the first transition state always shows a transition indication (even if there are no probes plugged in). One way to avoid triggering on a false first transition indication is to set the first state to "If Anything Goto Next State". This will consume a state, but will ensure that there is no false triggering on the first transition indication.*

- **Conditional.** Store samples only if the storage clause is true.

Program this clause in the same way as a regular trigger clause.

- **Start/Stop.** Storage is controlled by the Start Storing and Stop Storing trigger actions in the Then statement of the trigger clause.

Use the Start Storage/Stop Storage button in the Trigger Detail to select whether storage is initially enabled. For example, if the button is set to Stop Storage, then the initial state is that no samples are stored; storage will begin when the trigger action Start Storing occurs.

Trigger Position for an LA Module

Overview

The Trigger Position selects the amount of pretrigger data to store and determines the position of the trigger in the data record.

Behavior

Once triggered, a module continues to acquire data until it fills a specified amount of memory. The total memory depth that the module fills is set in the Memory Depth field in the Setup window. The Trigger Position field determines the proportion of data that is stored before and after the trigger. For example, if the Trigger Position is set to 10% and the module triggers, then the module continues to acquire data until the remaining 90% of memory is filled.

NOTE. *Under some conditions, the module cannot meet the pre-trigger data storage requirements. For example, if a trigger occurs immediately, the module cannot meet the pretrigger data storage requirement unless it was set to 0%.*

To Use

Set the Trigger position for a LA module in the toolbar of the LA Trigger window using the Trigger Pos control.

Related Topics

[LA Triggers \(see page 145\)](#)


[System Trigger Placement \(see page 148\)](#)

Trigger Programming Tips for an LA Module

- **Use the EasyTrigger program list**

When you create a trigger, start with the trigger program that is closest to your requirements and modify it.

- **Create your own trigger program folder**

Create a folder and save modules that have your own most-used trigger programs. Use the Comment box to help you identify the trigger program later. To access previously saved trigger programs, click the  button from the trigger toolbar, which displays the Load Trigger dialog box. Then click the Browse button and navigate to your trigger program folder.

- **Check the Overview**

After you create a trigger program, review the Overview pane on the PowerTrigger tab to ensure that all states in the program are linked and that state actions, such as program triggers and arming signals, are correctly placed.

- **Start and stop timers**

Remember to explicitly start and stop PowerTrigger timers. They do not start and stop automatically.

- **Use the Status Monitor to debug trigger programs**

The status monitor displays the activity within each state.

- **Trigger state references**

Do not delete PowerTrigger states to which you reference Go To State actions unless you change the reference to another appropriate state or delete the Go To action.

- **Use event text that is always true**

A PowerTrigger clause with an Anything event or another event that is always true, such as Group xyz = XXXX, prevents any subsequent clauses from being evaluated. Such clauses should be considered as default clauses; place them after all other clauses for a given state.

- **Use range recognizers**

When using range recognizers, the individual probe channels must be in hardware order; otherwise the trigger program can cause unexpected results. Range recognizers will only be allowed to encompass signals on the first three modules of a merged set.

- **Use of complex trigger events**

If you are unsure that a complex trigger event will occur, you should first use a broadly defined trigger condition and then subsequently add events to narrow the search. If your trigger program is too narrowly defined or over specified, the trigger program may not acquire the desired data or may never trigger.

- **Use of counters when using asynchronous sampling at fastest sampling rate**

For TLA7000 Series logic analyzer modules and TLA5000B Series logic analyzers, when using asynchronous sampling and running at the fastest rate, a 2 ns delay is required to allow the counter event to propagate through the system before the trigger state machine can respond.

Related Topic

[Range Recognizers and Hardware Order \(see page 153\)](#)

Range Recognizers and Hardware Order

When using range recognizers, the groups and signals must be in hardware order.

Groups must be used from the most-significant probe group to the least-significant group based on the following order:

C3 C2 C1 C0 E3 E2 E1 E0 A3 A2 D3 D2 A1 A0 D1 D0 Q3 Q2 Q1 Q0 CK3 CK2 CK1 CK0

Signals must be from the most-significant channel to the least-significant channel based on the following order:

7 6 5 4 3 2 1 0

The above example assumes a 136-channel LA module. The missing channels in LA modules with fewer than 136 channels are ignored. With merged modules, range recognition extends across the first three modules; the master module contains the most-significant groups, while slave2 contains the least-significant groups.

Related Topic

[Signed Decimal Values \(see page 153\)](#)

Signed Decimal Values

Signed Decimal Values can create confusing interpretations, especially when the listing display uses signed decimal values.

The following is an example of a problem that can be caused when using signed decimal values in the listing display in an If/Then statement:

```
If Group A3 < -128
```

```
Then Trigger
```

A3 is an 8-bit group, so it should never trigger since -128 is the smallest signed value possible; however, -128 is also 80 Hex, so many values will cause the logic analyzer to trigger, all of which will appear as larger values when displayed in the listing with a signed decimal radix.

In this example there is no way to program the range recognizer hardware to behave correctly and not trigger.

Module Arming Using an LA

Overview

Using the arm feature, you can use one module to control when another module begins acquisition.

When module A arms module B, this means that module B does not begin looking for a trigger until it receives an arm signal from module A.

Behavior

Arming is done through PowerTrigger actions specified in the Clause Definition dialog box. A module can arm any other module, and you can set up an arming relationship between more than one set of modules. But only one module can arm another. The same arming action can, however, appear multiple times within the same trigger program.

Arming and internal signaling are mutually exclusive. You cannot simultaneously arm modules and set signals.

For TLA700 Series logic analyzer modules, the trigger state machine is disabled when arming a module and does not store data until the module is armed. For TLA7000 Series logic analyzer modules and TLA5000B Series logic analyzers, the trigger state machine is also disabled when arming a module but this does not restrict data storage. The current setting of the Storage list box, located in the Trigger window, determines whether data is stored.

Related Topics

[Signals Properties \(see page 375\)](#) (External signals)

[Clause Definition Dialog Box \(see page 163\)](#)

Internal Signaling

Overview

Internal signaling is for users with special trigger programming requirements. Internal signaling adds flexibility to trigger programming, but also adds complexity. When using internal signaling take care that the signals are correctly set and cleared, and that the trigger programs for all modules are compatible with regard to signal usage. Also, set the correct internal signal attributes in the Signals tab.

Behavior

All LA modules can set and clear any of the four internal system signals. The DSO can monitor and set but not clear any signal. The logical output of these signals can be used as an event in other modules' trigger programs. You can also connect the internal signals to the External Signal In and External Signal

Out connectors on the mainframe, so that you can use an external signal as a trigger event or send a signal out when a trigger condition is met.

[Internal Signaling](#) is done through PowerTrigger actions specified in the Clause Definition dialog box.

NOTE. *Arming and internal signaling are mutually exclusive. You cannot simultaneously arm modules and set signals.*

Related Topics

[Signals Properties \(see page 375\)](#)

[Clause Definition Dialog Box \(see page 163\)](#)

[Trigger Action Signal Restrictions \(see page 155\)](#)

Internal Signal Selection

Signals 1 and 2 are high-speed, ECL-rate signals, and are usually the preferred choice for both internal and external signal applications. Signals 3 and 4 are functionally equivalent to signals 1 and 2, but operate at the slower TTL rate.

Signals 1 and 2. 162 MHz maximum (internal), 62.5 MHz maximum (at External Signal Out connector)

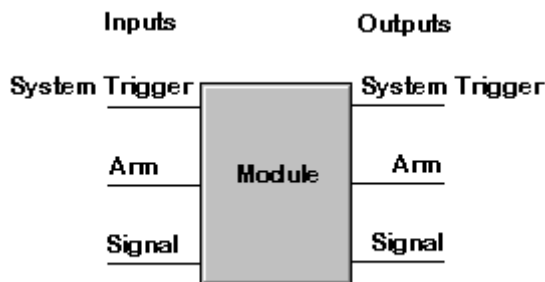
Signals 3 and 4. 12.5 MHz maximum

Trigger Action Signal Restrictions

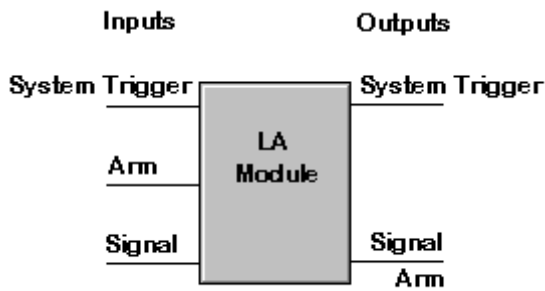
There are some differences between modules in how they handle triggers, arming and internal signals.

LA modules, DSO modules, and external oscilloscopes have different numbers of signal inputs and outputs. As a result, their signaling capabilities are different.

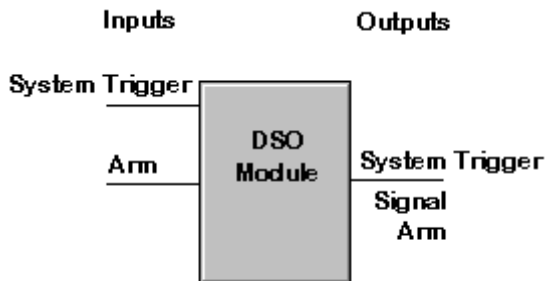
Conceptually, a module has three inputs and three outputs, as shown:



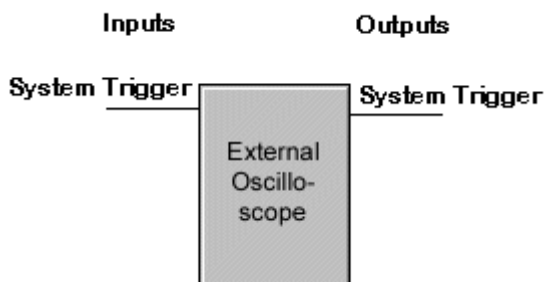
In reality, the LA module has three inputs and two outputs, designated as shown:



The DSO module has two inputs and a single output, as shown:



The External Oscilloscope has one input and either zero or one output, as shown:



This shows why Arming and Signaling are mutually exclusive on both the LA and DSO modules, and why a DSO module does not allow you to set a system trigger and an internal signal at the same time.

Related Topics

[Internal Signal Selection \(see page 367\)](#)

[Module Arming Using an LA Module \(see page 154\)](#)

[Signals Properties \(see page 375\)](#)

Change Detect Dialog Box

Use this dialog box to select the groups that will detect a change in state for Transitional storage. Once a change is detected in any of the selected groups, storage begins.

To open this dialog box, perform the following:

1. In the System window, click the Trigger button for the module.
2. Click the PowerTrigger tab.
3. Select Transitional in the Storage box, which is located in the toolbar.
4. Click the Change Detect button.

TIP. *To detect a change on a single channel, you can define a signal in the Setup window as a group.*

On TLA7000 Series logic analyzer modules and TLA5000B Series mainframes, the first transition state indication is always false. On TLA700 Series logic analyzer modules, the first transition state always shows a transition indication (even if there are no probes plugged in). One way to avoid triggering on a false first transition indication is to set the first state to "If Anything Goto Next State". This will consume a state, but will ensure that there is no false triggering on the first transition indication.

You cannot use transitional storage if your PowerTrigger program is set up with Group Changes or Channel Goes events.

When global storage is set up to use transitional storage, to ensure that a false transition is not recorded, the first state must be "If Anything Then Next State".

Select Signal Dialog Box

Use the Select Signal dialog box to select a specific logic analyzer channel. You can use the following options, available from the Select group box, to specify how the system displays signals in the dialog box.

By Group - displays signal organized by groups. Only groups that are defined in the Setup window can be displayed using this option.

By Probe - displays signals that are alphanumerically organized by individual channels. All signals are displayed when this option is selected, regardless of whether they are defined in the Setup window.

By Name - displays signals by the name assigned to them from the Setup window. Enter names for each signal for names to display when using this selection option.

Related Topics

[Selecting Channels for an LA trigger Channel Event \(see page 95\).](#)

[Routing a Signal from a Logic Analyzer Output \(see page 66\)](#)

Transition Definition Dialog Box

Use this dialog box to define transition events used in the Clause Definition dialog box. Each transition event displays values for valid groups defined in the Setup window. The value of each group is compared against live data.

You can disable the group in each transition event by removing the check mark adjacent to the group name; this causes the transition event to ignore the data for the group. You can also disable or enable all groups by clicking either the Clear All or Select All button. This either removes or adds check marks adjacent to each group name. You can additionally rename the current transition event by clicking the Rename button and entering a new transition name in the Rename group box.

Change the [group radix \(see page 166\)](#) for each group by selecting the desired radix at the bottom of the dialog box.

Conflicts are possible when groups share signals. Conflicts are noted in the boxes. Conflicts do not prevent acquisition.

The Trigger Conflict dialog box displays when a trigger conflict occurs, allowing you to select a trigger replacement option to resolve the conflict.

Value settings closer to the top of the list override later settings.

To Use

Each transition event is identified by a unique name.

To open the Transition Definition dialog box, perform the following:

1. From the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click either the If/Then or the Else If/Then button or the clause you are interested in defining.
4. Click Transition from the drop-down list available from the If section of the Clause Definition dialog box.
5. Click the Define Transition button in the If section of the Clause Definition dialog box to display the Transition Definition dialog box.

Related Topics

[Add Transition Dialog Box \(see page 159\)](#)

[Defining Multiple Transition Recognizers \(see page 94\)](#)

Add Transition Dialog Box

The Add Transition dialog box supplies words that are defined in the Transition Definition dialog box, and are available for defining PowerTrigger Clause Definition If statements.

To Use

Use this dialog box to add a new transition event. Click Use Existing Transition as Template to copy the settings from an existing word event.

To display the Add Transition dialog box, perform the following:

1. From the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click either the If/Then or the Else If/Then button for the clause you are interested in defining.
4. Click Transition from the drop-down list available from the If section of the Clause Definition dialog box.
5. Click the Define Transition button in the If section of the Clause Definition dialog box to display the Transition Definition dialog box.
6. Click the Add button to add a new transition event.

Related Topics

[Defining Multiple Transition Recognizers \(see page 94\)](#)

[Transition Definition Dialog Box \(see page 158\)](#)

Word Definition Dialog Box

Use this dialog box to define word events used in the Clause Definition dialog box. Each word event displays values for valid groups defined in the Setup window.

The value of each group is compared against live data. You can disable the group in each word event by removing the check mark adjacent to the group name; this causes the word event to ignore the data for the group. You can also disable or enable all groups by clicking either the Clear All or Select All button. This either removes or adds check marks adjacent to each group name.

You can change the [group radix \(see page 166\)](#) for each group by selecting the desired radix at the bottom of the dialog box.

Each word event is identified by a unique name.

To Use

To open the Word Definition dialog box, perform the following:

1. From the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click either the If/Then or the Else If/Then button for the clause you are interested in defining.
4. Click Word from the drop-down list available from the If section of the Clause Definition dialog box.
5. Click the Define Word button in the If section of the Clause Definition dialog box to display the Word Definition dialog box.

Conflicts are possible when groups share signals. Conflicts are noted in the boxes. Conflicts do not prevent acquisition.

The Trigger Conflict dialog box displays when a trigger conflict occurs, allowing you to select a trigger replacement option to resolve the conflict.

Value settings closer to the top of the list override later settings.

The Word Definition dialog box is also available from the EasyTrigger tab, depending on the program that you select.

Add Word Dialog Box

Overview

Use this dialog box to add a new word event. Click Use Existing Word as Template to copy the settings from an existing word event.

Behavior

The Add Word dialog box supplies words that are defined in the Word Definition dialog box, and are available for defining PowerTrigger Clause Definition If statements.

To Use

To open the Add Word dialog box, perform the following:

1. From the System window, click the LA Trigger button.
2. Click the PowerTrigger tab.
3. Click either the If/Then or the Else If/Then button for the clause you are interested in defining.
4. Click Word from the drop-down list available from the If section of the Clause Definition dialog box.

5. Click the Define Word button in the If section of the Clause Definition dialog box to display the Word Definition dialog box.
6. Click the Add button to add a new word event.

NOTE. *The Add Word dialog box is also available from the EasyTrigger tab, depending on the program that you select.*

Detecting Violations

One of the logic analyzer’s most useful features is its ability to detect and trigger on both signal glitches and setup and hold violations.

Glitches

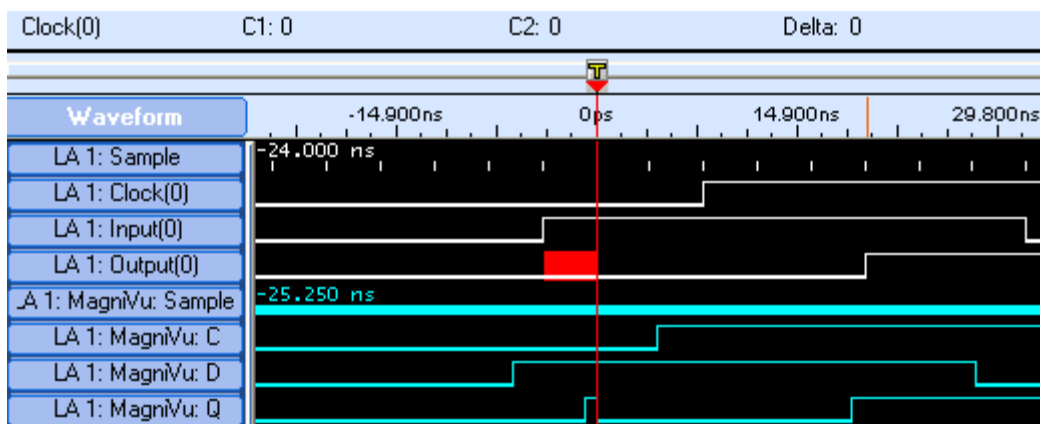
A glitch is a signal that makes a transition through the threshold voltage two or more times between successive acquisition samples. Because glitches are often signal transitions that occur intermittently, they can cause circuit malfunctions that are extremely difficult to diagnose.

Although you could try using a very fast sample rate to ensure that you never miss any glitches, a better solution is to use the glitch-detection feature. The logic analyzer can trigger on a glitch, either alone or in combination with other signal events. This capability is useful for catching intermittent glitches that might not occur very often or appear only when a particular operation is taking place.

You can capture noise spikes and pulse ringing using the glitch capture feature. The following figure shows data captured by triggering on a glitch. In the Waveform window, a glitch captured by the logic analyzer is indicated by a band of color.

Glitch mode affects the storage of glitches in memory, which allows you to display the glitches in the data window. Selecting Glitch mode reduces the available memory depth by half.

Glitch storage is independent of glitch events. Therefore, you can trigger on a glitch even when you are not storing glitches.



Setup and Hold Violations

A setup and hold violation is a data signal that transitions within the setup and hold time period. You could identify setup and hold violations by looking at each clock edge and reviewing all relevant data signals. However, it is faster and reliable to use Setup/Hold triggering to identify violations.

First select the correct sampling mode to use either the glitch detection or the setup and hold violation features. Choose asynchronous sampling to use glitch detection; choose synchronous sampling or custom sampling to use setup and hold detection.

63-Sample Blocks Mode Details

Overview

The 63-Sample Blocks mode provides the ability to acquire data before and after desired points and to limit acquired data to a minimum, enabling you to analyze a very long time window. Circumstances under which Blocks mode is useful include the following:

- Troubleshooting interrupt or subroutine calls and returns
- Troubleshooting DMA data transfers
- Troubleshooting certain types of hardware timing problems when using asynchronous sampling

Behavior

From the Setup menu, the Acquire mode can be set to Samples, Samples + Glitches, Samples + Violations, or 63-Sample Blocks. In Samples mode, only the samples selected by the trigger actions and qualifications are sorted. In 63-Sample Blocks mode, you store 31 samples before and after the target sample, or a block of 63 samples centered on the point where the sample is normally stored.

The ability to use blocks acquisition with either state (synchronous sampling) or timing (asynchronous sampling) allows you to use this feature for both software and hardware timing analysis.

- In 63-Sample Blocks mode only signal data is stored.
- If you store blocks, they will override other forms of data qualification.
- All samples in the block are always stored if post-fill has not matched any of its limits.
- You cannot change block size; it is fixed in the hardware.

63-Sample Blocks Mode Example

The following example troubleshoots the stack operation on a microprocessor system during the start of an interrupt routine that is repeatedly called, such as a video vertical retrace interrupt. In this example, the

group name used for data is DATA. The signal names used for an interrupt and an instruction fetch strobe are INTR and IFETCH, respectively.

This example checks the stack push operation by storing the start and end of every interrupt routine until the analyzer memory is filled. You first set the storage control to None (do not save anything unless explicitly stored in trigger states) and specify the trigger state machine as follows:

STATE 1: IF channel INTR goes low, THEN goto state 2.

STATE 2: IF group DATA = A333 AND channel IFETCH = 1, THEN store sample AND goto state 3.

STATE 3: IF group DATA = 0551 AND channel IFETCH = 1, THEN store sample AND goto state 1.

This trigger setup stores one sample at the instruction (A333) starting the interrupt service routine, and one sample at the instruction (0551) ending the routine.

Since no trigger was set up in the example, set the trigger position to 100% and enable the status screen. This will cause the memory to fill while waiting for a (never occurring) trigger. Then, after the memory is full (state screen shows 0 remaining), you press the RUN/STOP button to force a manual trigger at the end of memory. If MagniVu is displayed, the 1 us display is centered at the end of the regular data display.

In the Samples acquisition mode, only these instructions (A333 and 0551) are stored until memory is filled. Unfortunately, this prevents you from seeing what happened in the stack push and pop at the start and end of the routine. In the 63-Sample Blocks mode, because 31 samples before and 31 samples after each target sample is stored, you can see the stack push and first few instructions at the start of the interrupt service routine and the last few instructions and stack pop at the end of the routine.

Define Glitch Detection

Use this to select which groups should have glitch detection in the clause definition associated with a trigger program. Once a glitch is detected in any of the selected groups, the glitch event condition has occurred.

Use synchronous sampling for the Glitch condition to appear in the If condition list. Otherwise, the Samples + Violation option must be used.

For information on how to open the Glitch Detect dialog box, see [Triggering a LA Module on a Glitch \(see page 96\)](#).

TIP. To detect a glitch on a single channel, define a signal in the Setup window as a group.

Clause Definition Dialog Box

Overview

Use this dialog box to define the If/Then clause (or Else If/Then) of your PowerTrigger program. The clause definition contains the most vital part of the trigger state. (The trigger state is shown in the

Overview pane of the LA PowerTrigger tab). It is in this dialog box that you define the condition for an event to occur before an action can take place. You can have up to 16 clauses per state, depending on the number and types of events monitored within a state.

The dialog box is divided into two parts, If and Then.

If

Use the first part to select the events you are interested in and the condition those events must meet before any actions can take place.

For example, you may be interested in an address group reaching a certain value. The group is the event and the value is the condition. After the condition occurs, the program executes the actions specified in the Then part of the dialog box. For information on trigger events refer to [LA trigger Event Table \(see page 167\)](#).

Then

Use the Then part of the clause to define the action that occurs after the If part is true. For example, after the If condition is met, the action you might want is to trigger the module or trigger all modules. Some actions interact with others and some actions can only be used under certain conditions. For information on actions refer to [LA Trigger Action Table \(see page 168\)](#). Up to eight simultaneous actions can be specified for a clause.

To open Clause Definition dialog box from the System window, click the LA Trigger button. Then click the If/Then button or the Else If/Then button, located on the LA PowerTrigger tab.

Keep the following in mind when using this dialog box:

- Samples + Violations appear with synchronous or custom sampling. Samples + Glitches only appear with asynchronous sampling.
- Events can be ANDed or ORed with each other. To add another event, click Add while an event is selected. Click the first And button to change to the Or expression. You can have up to eight events.
- You can use an Anything or Nothing event as a placeholder. Anything is used when the events are combined with And. Change the And to Or and the event changes to Nothing.
- When using events you should be aware of the number of event resources available for a trigger program. For more information, refer to [LA Trigger Resources \(see page 169\)](#).

Related Topics

[Storage Definition Dialog Box \(see page 92\)](#)

[Signed Decimal Values \(see page 153\)](#)

Define Snapshot Dialog Box

Overview

TLA7000 Series logic analyzer modules and TLA5000B Series logic analyzers support the use of a snapshot to capture a given data sample and compare it with data that the trigger state machine subsequently processes. Based on criteria that you specify, the result of the snapshot comparison can cause a predefined event to occur. You set up the snapshot using the Define Snapshot dialog box, which allows groups to be included or excluded from the snapshot definition. By default, all groups are included.

Behavior

You can exclude a given group by removing the check mark adjacent to the group name. In addition, you can exclude all groups by clicking the Clear All button, which removes check marks adjacent to all group names.

Four types of events operate on the snapshot. At any given time, only one of these events is true:

- Snapshot equals the current sample - A snapshot has been previously loaded and it matches the current data sample from the target system.
- Snapshot does not equal the current sample - A snapshot has been previously loaded and it does not match the current data sample from the target system.
- Snapshot is not loaded - A snapshot has not been loaded or is still in the process of being loaded.
- Snapshot is loaded - A snapshot has been loaded and is ready to compare acquisition data.

NOTE. *Use of a snapshot event requires both an additional dedicated word recognizer and a timer. If either resource is unavailable, a warning message displays and the event type reverts to its previous value.*

Related Topics

[Triggering an LA Module on a Snapshot Event \(see page 99\)](#)

[Trigger Program Structure for an LA Module \(see page 181\)](#)



System Trigger and Reference System Trigger Relationship

Overview

The system trigger is the primary reference point for the entire data acquisition. In data windows, timing and location information is relative to the system trigger. There is only one system trigger per acquisition.

Behavior

If you add reference data to your data window, you can also add the system trigger for this reference data. In this case you have two system triggers in the display: The active system trigger and the reference system trigger. You can tell the two apart by the marks that identify them.

The active system trigger is identified by a large yellow T mark:  The reference system trigger is shown as a gray T mark .

You may not be able to see the reference system trigger in some data windows. For example, you create a reference system (Save System) that contains two LA modules, A and B. The time domain of both modules' acquired data do not overlap, and the system trigger is within module B's time domain.

A data window created from module A will not contain a reference system trigger. If you open the Go To dialog box, a reference system trigger will not be listed. However, because timing and location information are relative to the system trigger, the Timestamp column of module A will be referenced to the system trigger in module B.

Group Radix

Use the group radix to display data in the LA Trigger window and in the Clause Definition dialog box in an easy-to-read format. The following table lists common radices used to view data.

Radix	Valid characters
Binary	0, 1, X
Octal	0-7, \$, X
Decimal	0-9, \$
Signed decimal	0-9, -, \$
Hexadecimal	0-9, A-F, \$, X
Symbolic	Symbol name

Behavior

A Symbol name refers to a set of characters defined in a symbol file.

The X and \$ characters normally appear in the Trigger window. An X represents any character, often known as a "don't care" character.

The X can represent a binary, octal, or hexadecimal value depending on the radix selection. The \$ character appears when you select a radix where the binary combination contains one or more X characters. The following example shows how a binary combination with an X character displays in different radices.

Radix	Display	Value when X=0	Value when X=1
Binary	10101X11	10101011	10101111
Octal	25\$	253	257
Hexadecimal	A\$	AB	AF

Radix	Display	Value when X=0	Value when X=1
Decimal	\$\$\$	171	175
Signed decimal	\$\$\$\$	-85	-81

Tip

To identify the contents of a \$ character in the Trigger window, change the group radix to binary.

LA Trigger Event Table

The following LA trigger events are available in the If section of the PowerTrigger Clause Definition dialog box.

Event	Description	Interactions
Word	Tests the groups for the word values defined in the Word Definition Dialog Box (see page 159) .	Uses one trigger resource across all defined groups.
Group	Tests a specified group for a specific value, a range of values, or a value change.	Only one change detection event (group or signal) is available in a trigger program. Range recognizers use three trigger resources per group event. Word recognizers or change detectors use only one trigger resource.
Channel	Tests the specified signal for a value or a value change.	Only one change detection event (group or signal) is available in a trigger program. Each event uses one trigger resource.
Snapshot	Available only on TLA7000 Series logic analyzer modules and TLA5000B Series logic analyzers; compares the current acquisition sample with a previously loaded sample in the snapshot recognizer.	Only one snapshot recognizer event is available in a trigger program. Uses two event lines and one timer.
Transition	Tests the specified group for the transition values defined in the Transition Definition Dialog Box (see page 158) .	Cannot use the following within the same trigger clause: Group Changes, Unchanged Group, Signal Goes, Signal Doesn't Go, and Transitional Storage capabilities.
Glitch	Detects glitches in groups. You must define the groups in which to detect the glitch (see page 163) .	Only available with asynchronous sampling. Uses one trigger resource.
S & H fault	Tests setup and hold parameters as defined in the Setup & Hold violation dialog box.	Only available as an event with synchronous or custom sampling. Uses one trigger resource.

Event	Description	Interactions
Counter and timer events	Tests the specified counter or timer value.	Counter 1 and 2 events conflict with Timer 1 and 2 events respectively. They also conflict with their respective counter and timer actions.
Signal	Looks for one of the four system signals.	Only one signal event is available in a trigger program. Uses one trigger resource.
Anything	All sampled data makes this event true.	Available with the AND conjunction.
Nothing	All sampled data makes this event false.	Used as a placeholder with the OR conjunction.

NOTE. These event selections are also available from the EasyTrigger tab, depending on the EasyTrigger program that you select.

Related Topics

[Signals Properties \(see page 375\)](#)

[Clause Definition Dialog Box \(see page 163\)](#)

[LA Trigger Resources \(see page 169\)](#)

LA Trigger Action Table

The following LA trigger actions are available in the Then section of the PowerTrigger Clause Definition dialog box.

Action	Description	Interactions
Trigger	Triggers the current module and causes both MagniVu and main memory postfill to begin.	When Trigger is used in the trigger program, Trigger All Modules cannot be used.
Trigger All Modules	Also known as a System trigger. This signal is also available at the System Trigger Out connector.	When Trigger All Modules is used in the trigger program, Trigger cannot be used.
Wait for System Trigger	Used for a module that does not trigger itself or any other module, but waits to be triggered by another module.	Capable of receiving the following three (mutually exclusive) trigger actions: Trigger, Trigger all Modules, or Wait for System Trigger.
Trigger Main	Triggers the module, and causes main memory postfill to begin.	At most, one type of main triggering can be used in the trigger program (Trigger, Trigger Main, or Trigger System)

Action	Description	Interactions
Trigger MagniVu	Causes MagniVu memory postfill to begin.	Trigger MagniVu can be used in combination with one of the main memory trigger actions (Trigger, Trigger Main, Trigger System, or Wait for System Trigger)
Snapshot Current Sample	Reloads the snapshot recognizer.	Snapshot is not available for event recognition while the snapshot is loading its sample.
Go To	Causes a change to a different trigger state.	Use only one in clause definition.
Counter and Timer actions	Starts, stops, resets, clears, increments, or decrements counters or timers.	Counter 1 and 2 actions conflict with Timer 1 and 2 actions respectively. The actions also conflict with their respective events.
Set and Clear Signal	Sets or clears one of the four system signals.	Use only one in a trigger program. Mutually exclusive with Arm Module action.
Arm Module	Sends an Arm signal to another module. The other module begins running its trigger program.	Can only arm one module in a trigger program, but actions can be taken throughout the trigger program. Mutually exclusive with Set and Clear Signal actions.
Store Sample	Stores exactly one sample.	Not available in Start/Stop storage mode.
Start and Stop Storing	Begins or ends storing of samples. Used in conjunction with Start/Stop storage mode in the Trigger window.	Available only in Start/Stop storage mode.
Do Nothing	Used as a placeholder when defining a complicated trigger program.	Does not override other actions specified in a clause.
Don't Store	Does not store this sample.	Not available in Start/Stop storage mode.

Related Topic

[Signals Properties \(see page 375\)](#)

[Clause Definition Dialog Box \(see page 163\)](#)

LA Trigger Resources

Overview

A trigger resource is a hardware constraint. Use trigger resources as tools to build a trigger program. A trigger program can use up to 16 trigger resources.

Behavior

A trigger resource can be used more than once in a trigger program. However some events use more than one trigger resource. The following table lists the trigger events, the event operator, the number of trigger resources, and resource restrictions for each trigger event.

Event	Operator	Trigger resources and restrictions
Word	=, Is Not	1 trigger resource
Group (word recognizer)	=, Is Not	1 trigger resource
(range recognizer)	<, <=, >=, >, Is In, Is Not In	3 trigger resources
(change detector)	Changes, Unchanged	1 trigger resource, one change detector allowed in a trigger program, not available when transitional storage is used
Channel (word recognizer)	=	1 trigger resource
(change detector)	Changes, Unchanged	1 trigger resource, one change detector allowed in a trigger program, not available when transitional storage is used
Snapshot	=, Is Not, Is Loaded, Is Not Loaded	2 trigger events and 1 timer event
Transition	Occurs, Doesn't occur	1 trigger resource for each of the 16 available transition recognizers used.
Glitch		1 trigger resource, only available with asynchronous sampling
S & H fault		1 trigger resource, not available with asynchronous sampling
Counter	>, <=	0 trigger resources, 2 counters or 2 timers Maximum width 51 bits Maximum sampling 250 MHz (TLA700 Series) 500 MHz (TLA7000 Series and TLA5000B) Maximum count 2,251,799,813,685,246
Timer	>, <=	0 trigger resources, 2 counters or 2 timers Maximum width 51 bits Maximum sampling 500 MHz (TLA7000 Series and TLA5200B) Maximum time 2,000,000 s or 23 days Resolution 2 ns for TLA7000 Series and TLA5000B
Signal (Signal In)	Is True, Is False	1 trigger resource, uses one of four system signals
Anything/Nothing		0 trigger resources

EasyTrigger programs are designed to take into account resource limitations so that, in most cases, you do not have to be concerned with resources.

For TLA7000 Series logic analyzer modules and TLA5000B Series logic analyzers, when using asynchronous sampling and running at the fastest sampling rate, a 2 ns delay is required to allow the counter event to propagate through the system before the trigger state machine can respond.

Related Topics

[LA trigger Event Table \(see page 167\)](#)

[LA Trigger Action Table \(see page 168\)](#)

[Clause Definition Dialog Box \(see page 163\)](#)

[Trigger Action Signal Restrictions \(see page 155\)](#)

[Internal Signaling \(see page 154\)](#)

Force Main Prefill

Overview

By default, the Force Main Prefill check box is selected, which defers execution of the trigger state machine until the main memory prefill amount is met. Use the Memory Depth field, located in the LA Setup window, to set the total memory depth that the module fills. Use the Trigger Pos slider or box, located in the LA Trigger window, to set the main prefill amount that must be met before the trigger state machine is activated.

Behavior

When you use Force Main Prefill functionality, you specify which data samples are stored by selecting the desired sample type from the Storage drop-down list. Note that if you select either the None or Stop/Start storage settings, the Force Main Prefill check box and related functionality become unavailable.

System triggers produced by other modules override force prefill functionality and immediately start execution of the trigger state machine. An externally generated system trigger does not purge the force prefill data samples stored in main memory. However, the TLA7000 Series logic analyzer modules or the TLA5000B Series logic analyzer stops acquiring further prefill data samples and begins acquiring postfill data samples until main memory allocated for postfill data samples is filled.

NOTE. To change the default setting for Force Main Prefill, from the System menu select Options, and then click the Defaults tab.

Related Topics

[LA Triggers \(see page 145\)](#)

[LA Trigger Action Table \(see page 168\)](#)

MagniVu

MagniVu functionality is designed to acquire data at a high sample rate so that you can see the timing of signals in much greater detail. This is useful when you are debugging hardware and needing higher resolution data samples for viewing events that change very quickly. Although the TLA7000 Series logic analyzer modules and the TLA5000B Series logic analyzer hardware supports storage at slower rates, data sampling is always performed at the maximum PowerFlex MagniVu sample rate. MagniVu memory depth is set to a fixed 16K.

Related Topic

[LA Trigger Action Table \(see page 168\)](#)

MagniVu Storage Rate

Overview

This is the rate at which MagniVu data is stored in memory. By slowing the MagniVu storage rate, you can stretch MagniVu memory over a greater span of time, providing a lower resolution data sample from a greater time period.

Behavior

The MagniVu storage rate cannot be set slower than the effective main memory rate, according to 1X, 2X or 4X modes. The available rates are:

- 8 GHz <-> 125 ps
- 4 GHz <-> 250 ps
- 2 GHz <-> 500 ps
- 1 GHz <-> 1 ns

The MagniVu storage rates are limited by the fastest storage rate of the main memory under 1X, 2X, and 4X modes. The slowest MagniVu Storage rate is half of the fastest storage rate of the main memory.

1X MagniVu storage selections	2X MagniVu storage selections	4X MagniVu storage selections
125 ps, 250 ps, 500 ps, 1 ns	125 ps, 250 ps, 500 ps	125 ps, 250 ps

NOTE. *MagniVu storage rate adjustments do not affect setup and hold violation or glitch violation.*

MagniVu Trigger Position

Overview

The MagniVu trigger position, available with TLA7000 Series logic analyzer modules and TLA5000B Series logic analyzers, selects the amount of pretrigger MagniVu data to store and determines the position of the trigger in the data record. You can control the MagniVu trigger position independently of main trigger position by a set of controls in the advanced toolbar.

Behavior

Once triggered, a module continues to acquire MagniVu data until it fills a specified amount of memory. MagniVu memory depth is set to a fixed 16K. The Trigger Position field determines the proportion of data that is stored before and after the trigger. For example, if the Trigger Position is set to 15% and the module triggers, then the module continues to acquire data until the remaining 85% of memory is filled.

The MagniVu trigger position controls operate the same as the controls for main memory trigger position. They consist of a slider control, an edit box, and a spin control positioned beneath the main memory trigger controls. They behave like the main memory counterparts, except that the MagniVu trigger position is constrained by the MagniVu storage rate as follows.

Storage rate	Resulting trigger position constraint
125 ps	0% to 58%, in 1% increments
250 ps	0% to 79%, in 1% increments
500 ps	0% to 89%, in 1% increments
1 ns	0% to 94%, in 1% increments

NOTE. *MagniVu data acquisition starts prior to the run line and fills memory before any possible trigger event. Therefore, there is no MagniVu forced prefill control.*

Related Topics

[MagniVu \(see page 172\)](#)

[LA Triggers \(see page 145\)](#)

[LA Trigger Action Table \(see page 168\)](#)

Multiple Samples per Clock Trigger Programs

Some trigger programs that rely on multiple samples per clock require special attention to work properly. In some situations, it may be necessary for a data value to be present on the probe input for two or more

clock cycles for a word recognizer to detect it. In other situations, it may be necessary to use word recognizers logically ORed together across the base and prime signals to detect a value.

When using multiple samples per clock, trigger programs only operate on the acquired data at normal speed. In other words, when using multiple samples per clock, the current trigger machine state is evaluated at the physical sample rate and not the logical sample rate.

To illustrate this point, consider the following two examples. In both examples, a minimum sample period of 2 ns is assumed with asynchronous sampling.

Example 1 shows a situation where there are two samples per clock.

A3 Channels	D3 Channels	Timestamp
01	02	0 ns
03	04	2 ns

The data was acquired with the trigger: **If Group A3 equals 01, then Trigger.**

However, if the "01" data value happened to be stored as the prime signal (D3¹), the instrument would not have triggered, even though the value occurred on the A3 lines and was captured in the acquisition memory.

To reliably trigger on an event that only lasts for 1 ns, use two word recognizers: one to watch the base signal, and the other to watch the prime signal. The following trigger will always catch the 02 pattern using Two Samples per Clock mode: **If Group A3 equals 02 or Group D3 equals 02, then Trigger.**

If the desired event for triggering is present for at least 2 ns, then it follows that the base signals will always capture the event. In this case, the trigger will work correctly in the Two Samples per Clock mode.

Example 2 shows a situation where there are four samples per clock.

A3 Channels	A2 Channels	D3 Channels	D2 Channels	Timestamp
01	02	03	04	0ns
05	06	07	08	2ns

The data was acquired with the trigger: **If Group A3 equals 01, then Trigger.**

However, if the "01" data value happened to be stored as one of the prime signals, the instrument would not have triggered, even though value occurred on the A3 lines and was captured in the acquisition memory.

To reliably trigger on an event that only lasts for 500 ps, use four word recognizers: one to watch the base signal, and the others to watch the prime signal. The following trigger will always catch the 01 pattern using Four Samples per Clock: **If Group A3 equals 01 or Group A2 equals 01 or Group D3 equals 01 or Group D2 equals 01, then Trigger.**

If the desired event for triggering is present for at least 2 ns, then it follows that the base signals will always capture the event. In this case, the trigger will work correctly using Four Samples per Clock.

NOTE. You can use the Trigger on group match or the Trigger on signal match [EasyTrigger Programs](#) (see page 183) located under Match Events on multiple samples per clock. You can set up this type of triggering on a data value that you specify.

Entering a Symbol Plus an Offset in a Trigger

In a group value field that has a symbolic radix you can directly type in a symbol with an offset. For example you can type into that field:

MyFunction+55

The application software will parse the symbol and the number and redisplay the result. The software takes the offset as a HEX number. As in other places where you type in numbers, you can force the radix to be something else. For example:

MyFunction+\d10 'decimal 10

MyFunction+\b1111 'binary 1111

MyFunction+\o33 'octal 33

MyFunction+\haa 'hexadecimal AA

Triggering Problems

When trigger program is not being satisfied, check the following:

- The trigger program is correct.
- Storage qualification is correct.
- The system under test or module does not have an exception or fault. The system or acquisition module might have experienced a hardware or software exception or fault after the trigger condition was satisfied.
- A clock is available. Click Status in the toolbar to open the Status Monitor and verify that a clock is available. If no clock is available, check your clock connections.

If the previous suggestions do not fix the problem, try the following:





- Reload the support package to restore the logic analyzer to a default state.
- Possible loading problems might be remedied by removing one or more of the protective sockets from the probe adapter. These sockets might add enough additional capacitance or delay to your system under test to affect it.

Serial Analyzer Trigger Window

The serial analyzer Trigger window is similar to a Power Trigger window on a logic analyzer module. You use it to construct a [trigger program](#). The trigger program is made up of one or more trigger states. Each state has one or more clauses. Each trigger state contains an area where you can enter a brief description of the state. You can have up to eight trigger states in the trigger window.

Each clause has a trigger event section that specifies a trigger event that you are looking for. The second part of the clause contains a trigger action section describing the appropriate action to take when the event information has been satisfied. You can have up to eight trigger clauses per trigger state.




Some suggestions for using the Trigger window follow:

- Click  to collapse the current trigger state to provide more room on the screen.
- Click  to expand the current trigger state.
- Look for the  or  to expand or collapse information in the current clause. They are indicators that there may be more or less information to display on screen.

TIP. *Configure the Setup window before the Trigger window, because the Setup window settings may affect Trigger window selections.*

Trigger Tool Bar

The trigger tool bar in the serial analyzer Trigger window has controls for common tasks. Icons at the left side of the tool bar provide an easy means to do the following tasks:

- Click  to set the default Trigger window.
- Click  to load a previously saved Trigger window.
- Click  to save a Trigger window.

Use the On Run control to specify the storage control options. Select Start Storing to begin storing data when you click the Run button. Select Don't Store to not store data until the conditions have been met in the trigger program.

Use the Store control to specify the amount of data to acquire. The data volume is expressed in terms of symbols with a maximum of 32 M symbols. Use the Trig Pos control to specify where the trigger will appear in memory (1% accuracy within the data record). Both of these controls perform the same functions as in the serial analyzer Setup window.

Creating a Serial Analyzer Trigger Program

Use the following guidelines to create a serial analyzer trigger program:

1. Open the serial analyzer Trigger window.
2. In State 1, enter a meaningful description in the Description field.
3. In Clause 1, click Add Event, and select a trigger event from the drop-down list.

Depending on the event, additional controls will appear. Select the information that you want from the drop-down lists. For some packets (such as TLPs, DLLPs, Sequences, or Link Events) you can click



to open a dialog box where you can further define the packet.

4. If necessary, click Add Event to add additional events to the current clause. You can logical OR or AND the events.
5. In the action area of the clause, select an action from the drop-down list and then fill out the necessary information to complete the trigger action.
6. If necessary, you can add additional clauses and states to complete your trigger program.


TIP. You can also add states, clauses, events, and actions by right-clicking the mouse.

Serial Analyzer Trigger Events

The following table lists the serial analyzer trigger events.

Event	Description
Anything	Recognizes any data.
TLP	Recognizes the presence or absence of a specific TLP. You can choose the TLP from a list, or define a TLP.
DLLP	Recognizes the presence or absence of a specific DLLP. You can choose the DLLP from a list, or define a DLLP.
Sequence	Recognizes a specific ordered set or symbol sequence. You can choose the Sequence from a list, or define a Sequence.
Link Event	Recognizes link events and link error signals. You can choose the Link Event from a list, or define a Link Event and specify which lanes to monitor.
Timer	Recognizes a specified timer value. You can have a maximum of four global counters or timers.
Counter	Recognizes a specified counter value. You can have a maximum of four global counters or timers.
Signal In	Recognizes a signal from another module.

Defining Packets in Trigger Windows

The serial analyzer lets you define packets for triggering on PCI Express data. Click  in the Event section of the trigger program to open a dialog box where you can define the event.

1. Enter a meaningful name for the packet or select one from the drop-down list.
If you select a packet from the list, a copy is automatically created.
2. Fill in the information as needed to define the packet.
3. To change the radix of a particular field, right-click the mouse in the field and then select a radix from the right-click menu.
4. Click close to save the changes.

NOTE. *The X and \$ characters appear whenever you define a TLP, DLLP, or a symbol sequence Event. An X represents any character, often known as a "don't care" character. The X can represent a binary, octal, or hexadecimal digit depending on the radix selection. The \$ character appears when you select a radix where the binary combination contains one or more X characters. To identify the contents of a \$ character change the radix to binary.*

Trigger Program Structure for an LA Module

A [trigger program](#) is made up of one or more [states](#) (up to 16). Only one state is active at a time.

Each state is made up of one or more [clauses](#). If there are more than four trigger event resources used in a state, you can have up to four clauses per state. If there are four or fewer trigger event resources in a state, you can define up to 16 clauses per state. However, if you use a snapshot recognizer as an event in your clause definition, your use of clauses per state is restricted as follows (note that neither counters nor timers are included as trigger resources).

- If you use a snapshot recognizer and more than two additional events, you can have up to two clauses (and one counter or timer) per state.
- If you use a snapshot recognizer and no more than one additional event, you can have up to eight clauses (and one counter or timer) per state.

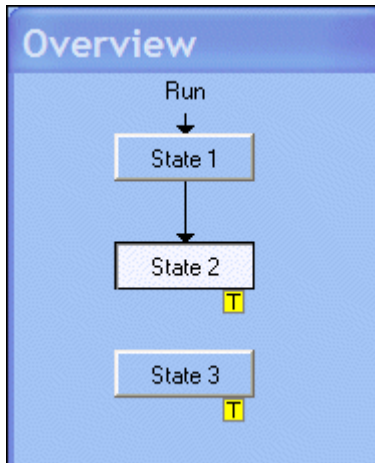
During each clock cycle, all clauses within the active state simultaneously evaluate the acquired data. The priority of evaluation of a clause is assigned from top to bottom. The If statement of a clause can contain a single event, or it can contain multiple event conditions. When the correct [event conditions](#) occur, the logic analyzer passes control to the actions specified in the Then statement. There are several actions to choose from, including triggering the system, or transferring control of the trigger program to another state.

If multiple clauses are true in the same clock cycle, only the clause with the highest priority that goes true will have its actions executed.

NOTE. *Avoid using the If Anything event in the top-most clause of a state with multiple clauses. Successive If-Then clauses will never become active. If you decide to use the If Anything event, it should be in the last clause of a state.*

PowerTrigger Tab

The Overview portion of the LA PowerTrigger tab shows the relationship between states.



Arrows indicate the transfer of trigger program control between states. Marks indicate system-level actions that could occur if a clause in that state goes true. Use the trigger Overview to check that all states are connected. With the exception of State 1, all states must have an arrow leading to them (indicating activation by another state), or they will not execute. These actions are defined in the Clause Definition dialog box.

Trigger Detail

The Trigger Detail, located in the right portion of the LA PowerTrigger tab, shows summary information about the clauses within the states. For lengthy trigger programs, click the State icon in the Overview to jump to the corresponding PowerTrigger details. Click the If/Then icon to open the Clause Definition dialog box, for complete information about the clauses for that state.

TIP. To check the progress of your trigger program during acquisition, click the Status button in the toolbar to display the status monitor.

To track a sequence of events, create trigger programs with multiple states.

To simultaneously monitor different events, create trigger programs with states containing multiple clauses. Each clause can carry out different actions when the events occur within each state.

Related Topics

[Clause Definition Dialog Box \(see page 163\)](#)

[LA Trigger Event Resources \(see page 169\)](#)

Trigger Clauses

Trigger clauses are made up of two parts: An If statement that defines the data events to search for, and the Then statement which specifies the actions taken when the If statement is true.

Clauses contain the most important part of the trigger program; they specify the data and conditions to look for. Clauses also define the relationship between the states by specifying when control is passed from one state to another state.

Event Conditions

Event conditions are a logical combination of trigger events within a single clause.

- If you set up a logical AND statement, all event conditions in the clause must be met before the clause can execute the action.
- If you set up a logical OR statement, any one of the event conditions can be fulfilled before the clause can execute the action.
- You cannot define both AND and OR statements, define one or the other.

Trigger States

Trigger states monitor and act on a sequence of data events. States break up the trigger program into stages.

States make the trigger program easier to read and provide more program flexibility by providing a method to define the program in greater detail. During acquisition, trigger program control passes from one state to another. The contents of the active state are evaluated each clock cycle.

Each state contains up to 16 clauses.

EasyTrigger Program List

Overview

An EasyTrigger program list contains a collection of predefined trigger programs for the LA module. A list of predefined trigger programs is available from the EasyTrigger tab of the Trigger window. These predefined trigger programs are designed to hide program implementation details, allowing you to focus on acquiring and displaying data. However, you can use the PowerTrigger tab to view the selected trigger program's inherent trigger states and related clauses that define the data event, and possible resulting actions.

To Use

You can also use the predefined trigger programs as the basis for designing increasingly complex trigger programs that meet your data requirements. Selecting a predefined trigger program from the EasyTrigger tab and then displaying it using the PowerTrigger tab also allows you to modify the preexisting state and clause definitions. After making trigger program modifications, you can save the resulting trigger program to a specified folder for later reuse by clicking the button, which is available from the Trigger window toolbar.

Symbol	Meaning
A, B, C	Replaces Events
Range A	A Range value

Symbol	Meaning
N	A count (counter)
T	A time (timer)

NOTE. *When you use the PowerTrigger tab to make changes to a predefined trigger program, you can no longer use the EasyTrigger tab to view the modified program. Attempting to do so will display a message warning you that, if you proceed, your unsaved program changes will be lost. Only predefined trigger programs can be viewed from the EasyTrigger tab.*

After you become familiar with defining trigger states and related event and action clauses, you can use the PowerTrigger tab to create new trigger programs that are specifically designed to your data requirements.

Related Topic

[Loading a Trigger from the Trigger Tab Page \(see page 92\)](#)




Listing Window

Overview

Use the Listing window to view and evaluate acquisition data. This window displays the acquired data as a result of what you specified in Setup and Trigger windows. To open a Listing window from the Window menu, click Listing.

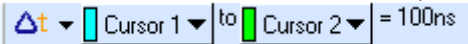

Columns


Data is presented as tabular text in columns. There are several types of columns; logic analyzer groups, individual logic analyzer signals, DSO channels, Disassembly, Sample numbers, and Timestamp values.


- Columns can be dragged and dropped. Click and drag the column label to where you want the column displayed. An arrow shows where the column will be placed.
- Double-click a column label to open the Column Properties tab.
- Add columns by clicking the Add Column toolbar button .
- Remove columns by selecting the column label and clicking the Cut toolbar button. You can paste the removed column to the right of any selected column.
- Expand columns to show the individual signals in a group by clicking the + . Collapse the individual signals by clicking the - .

Marks

Marks are used to make time measurements, locate triggers, and mark specific samples for evaluation.

- Marks you can move include Cursor 1, Cursor 2, and any marks you add. The numbers 1 and 2 on the mark handles denote the cursor marks. Readouts for Cursor 1, Cursor 2, and the time difference (Delta Time) between the two are located on the Delta Time toolbar. . You can add any number of delta time toolbars by clicking the Add Delta Time button at the top of the Listing window. .
- Marks on the left side of the window are data area marks. These marks are in the portion of the acquisition that is currently viewable. Use data area marks to move within the data area.
- You cannot move System Trigger, Module Trigger, or Begin and End marks.
- Marks on the right side are overview marks. These marks show where marks are located relative to the entire acquisition. Use overview marks to move throughout the acquisition. For example, move Cursor 1 to a particular sample as shown in the Cursor 1 readout. Click the Go To toolbar button and select Cursor 1. You can also click the right mouse button on any overview mark, and then click Go To Mark.


- Double-click a mark to open the Marks properties tab. You can assign a unique name to the Mark in the Mark Name field.
- You can add marks by clicking  (the Add Mark toolbar button). Remove a mark by selecting the mark and using the Cut toolbar button. When you add or paste a mark, it appears in the center of the display.
- You can move to any mark using the Go To toolbar button.

System trigger 



The system trigger is the reference point for the acquisition. Timing and location information is relative to the system trigger. Trigger marks cannot be moved.

The system trigger associated with the current data is called the active system trigger. The system trigger associated with saved data is called the reference system trigger. The active system trigger is indicated by a yellow T; the reference system trigger is indicated by a gray T.

The system trigger appears even if no data from the module that generated the system trigger is around it.

Module trigger 

The point at which the module triggered. Trigger marks cannot be moved.

Begin data and end data  and 

The start and end of a module's data record. These data marks cannot be moved.

Cursors 1 and 2



Moveable marks used for visual reference and for data measurements.

User mark 

User-created marks. Use marks to make specific data easier to identify and find.

Lock Delta Time check box

Click this check box to lock the delta time value. When you lock delta time, the marks move across the data simultaneously, maintaining a fixed delta time.

Data Area

- The Data area can be split vertically by dragging the split box from the left end of the horizontal scroll bar. This allows you to compare columns.
- [Qualification Gaps](#) are indicated with a horizontal gray line above the first data after the gap. Turn indicator gaps on or off from the Listing Window Properties tab.
- Glitches are indicated by highlighted text of the complete sample, except for binary radices. Only the glitch bits are highlighted in binary. Select the color of highlighted glitch text from the Listing window properties tab. Select radices from the Column properties tab.

Enable Glitch storage in the Setup window before acquiring data.

Memory [differences](#) and [equalities](#) are indicated by highlighted data. You can enable data comparison and select the color of highlighted data from the Listing window properties tab.

Before you can acquire and compare data, first enable the comparison in the Setup window and define both the signals to compare and a valid data reference.

Qualification Gaps

Positions between data samples where one or more samples have not been stored.

Data Differences

Highlighted data shows where there are differences between the two signals or groups being compared.

Data Equalities

Highlighted data shows where there are no differences between the two signals or groups being compared.

Add Column/Waveform Dialog Box

Use to add a column to the Listing window, or a waveform to the Waveform window:

1. Select the data source and its associated group or signal to add it to the display.
2. Click the Add Data Source button to add a source to the selection box.

To open the Add Column or Add Waveform dialog box from the System window, click the appropriate data window icon. Then go to the Edit menu and click either Add Column or Add Waveform.

TIP. To quickly add columns and waveforms, right-click in the Listing or Waveform window and select either Add Column or Add Waveform.

Lock Windows Dialog Box

Overview

Use the Lock Windows dialog box to lock cursors and the display areas of selected data windows. Locking windows provides an easy method to compare data from two different windows.

Use the Lock Windows dialog box to select how windows are locked together with respect to cursors C1 and C2 and the center screen.

Behavior

To have the cursors and center screen of all locked windows go to the same time stamp position as the selected reference window, click the No time offsets option button. To have each window's cursor and center screen retain its current position, click the Current time offsets option button.

You can lock Listing, Waveform, Protocol, and Graph windows.

TIP. *One way to jump over big time gaps in the Waveform window is to create a Listing window of the same data, and then lock the two windows together. As you move the cursor through the Listing window, the cursor in the Waveform window will track and jump over the "dead" space between samples.*

To Use

To open the Lock Windows dialog box from the System window, click the Listing, Waveform, Protocol, or Graph data window icon. From the System menu, click Lock Windows.

Export Data Dialog Box

Overview

Use the Export Data dialog box to export data from the current Listing window to an ASCII text file or to a binary file. You can export the following listing data:

- All data from the acquisition
- A range of listing data between two selected marks
- A range of listing data between two samples

Behavior

If you choose to export the listing data to a text file, you can click Options to define the format of the data. The data is saved in a file with the .txt extension.

If you choose TLA Data Exchange the text format is predefined as follows. The first line displays vectors and the second line contains the column headings Sample, Address, Data, and Timestamp.

If you choose to export the listing data to a binary file, the data is saved in a file with a .tbf (TLA binary format) extension. Sample numbers and mnemonics groups are not exported; time stamps are listed in picoseconds and are relative to the start of the acquisition. The binary option is available only when the data window has a single data source.

To Use

To open the Export Data dialog box, open a Listing window. From the File menu, click Export Data.

Related Topic

[Export Data Options Dialog Box \(see page 190\)](#)

[TLA Binary File Format \(see page 189\)](#)

TLA Binary File Format

Overview

The TLA Binary File Format is used to export data from Listing windows in binary rather than ASCII format. You can use this format to export DSO data and MagniVu data in Listing windows, in addition to the standard listing data.

LA Module Binary Format

The binary data for an LA module is exported as a stream of bytes with the following characteristics:

- The binary data uses big endian fields (the most significant bit is the left-most data bit).
- Each group field is zero-padded to the nearest byte.
- Sample numbers and mnemonics groups are not exported.
- Time stamp data is seven bytes wide and represents the number of picoseconds since the start of the acquisition.
- The left-most column in the Listing window corresponds the first exported field of a sample.
- The first sample in the Listing window appears at the beginning of the exported file.
- MagniVu data follows the same guidelines as regular listing data.

DSO Module Binary Format

The binary data for a DSO module (in a Listing window) is exported as a stream of bytes with the following characteristics:

- The binary data uses little endian fields (the most significant bit is the right-most data bit).
- Each channel is a 16-bit value.
- Sample numbers, mnemonics groups, and time stamp values are not exported.
- The left-most column in the Listing window corresponds the first exported field of a sample.
- The first sample in the Listing window appears at the beginning of the exported file.
- MagniVu data follows the same guidelines as regular listing data.
- DSO data values are in twos-compliment format.
- The formula used for converting a channel field into a voltage is as follows:

$((\text{vertical range in volts} / 64512) \times \text{channel field}) + \text{vertical offset in volts}$

Export Data Options Dialog Box

Overview

Use the Export Data Options dialog box to customize listing data for export. The settings are retained after you close the dialog box.

To Use

To open the Export Data Options dialog box, open a Listing window. From the File menu, click Export Data, and then click Options.

1. Choose a field delimiter to separate the listing data columns.
2. Click the Include Unit Character check box to export data with measurement units. If you export data without unit characters, time stamp values are in picoseconds and analog values are in volts.
3. Click the Fixed Data Radix check box to export data in the specified radix.
4. Click the Use Enhanced Column Headers check box to export group width and radix information in column headers.

Compare Definition Dialog Box

Before you can compare data, use the Define Compare dialog box to specify the source of the reference data, the number of samples to compare, and how to align the data. To enable the Compare Definition dialog box controls, click the Enable Data Compare check box.

A summary of the samples being compared is displayed at the bottom of the dialog box. This area shows the samples that are being compared in the current acquisition using the current compare range and alignment settings.

To Use

1. From the toolbar, open the Setup menu and select your module.
2. In the Setup window, [define a group \(see page 22\)](#).
3. Create a Listing window.
4. In the toolbar, click the Define Compare button.

Behavior

You can compare data in the following ways:

- **Reference Data Source.** Select a reference data source from any of the installed modules or from a saved setup.
- **Compare Data Region.** Compare all data between the system under test and the reference data. You can also compare a range of data with respect to the beginning of memory or around the trigger location.
- **Data Source Alignment.** Align and compare the data with respect to the beginning of memory or to the trigger location. You can also specify the number of samples to offset the data.
- **Compare limitations.** Compare only the main LA data. You cannot compare glitch data, disassembler group data, MagniVu data, External Oscilloscope or DSO data.

Repetitive Properties Dialog Box

Overview

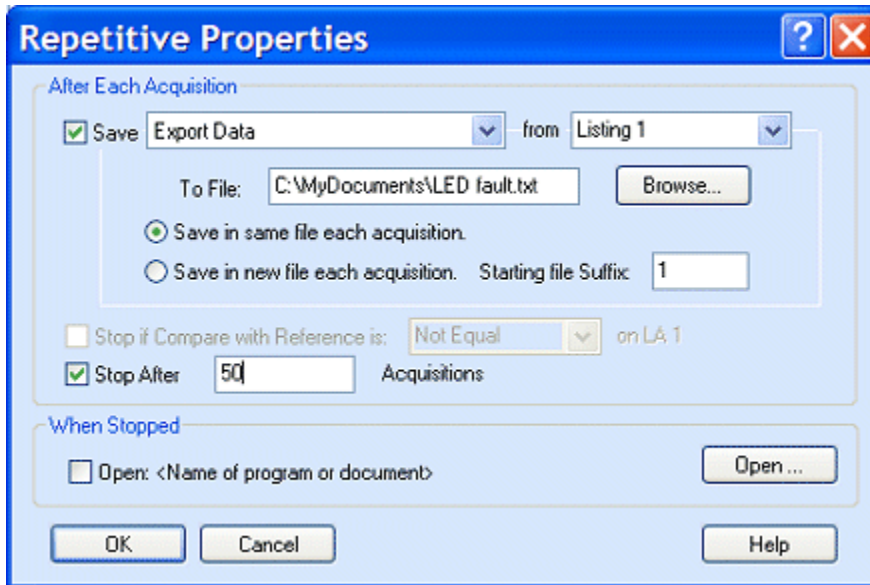
Use the Repetitive acquisition features of the logic analyzer to automate repetitive and time-consuming tasks. For example, you can specify the number of times that you want the logic analyzer to acquire data. With each acquisition, you can save the data to a file for analysis. You can also set up the logic analyzer to open a file or execute a program when all of the acquisitions have been completed.

Behavior

Click the Save check box to save the setups or export data to a file. You can overwrite the file with each acquisition or you can save the data in separate files by entering a starting file suffix number.

During compare operations, you can acquire data until the compare conditions are met. Then click OK to close the dialog box and save your settings.

The following is an example of such a setup where the acquired data is exported to a file for each acquisition. When a data-mismatch occurs, the logic analyzer stops acquiring data and exports the data to a file. The faulty data can now be analyzed by another application.



NOTE. Click the Enable Data Compare check box, located in the Compare Definition dialog box, to stop the acquisitions when the compare conditions are met.

After completing the repetitive operation, you can have the logic analyzer open a document or execute an operation as defined in the Open dialog box. Click Open to access the Open dialog box.

TIP. To improve the response time of the logic analyzer during repetitive operations, you can delete any unnecessary data windows. This allows the logic analyzer to spend time on processing data rather than redrawing data in data windows.

To open the Repetitive Properties dialog box from the System menu, click Repetitive Properties.

Related Topics

[Compare Definition Dialog Box \(see page 190\)](#)

[Export Data Dialog Box \(see page 188\)](#)

[Open Dialog Box \(see page 193\)](#)

Open Dialog Box

Overview

Use the Open dialog box to open a document or an application after fulfilling the requirements of a repetitive operation.

To Use

Enter the path and name of the document or application that you want to open. If you do not know the path name of the document or the application, click the Browse button to navigate to the document or application.

- Use the Arguments list to enter any command line arguments that are required by the application or document.
- Use the Initial Directory box to specify the default working directory for the application or document.
- Click the Test Open button to verify the selections for the Open dialog box.

To access the Open dialog box from the System menu, click Repetitive Properties, and then click the Open button.

Define Search Dialog Box

Overview

Use the Define Search dialog box to search for specific data within the current data window. Search definitions can search any number of data sources at the same time. By default, searches begin from the active cursor, but you can specify any cursor or mark to begin the search. All data sources begin searching at the same point and continue until the number of occurrences specified is found. Neither hidden data nor suppressed data can be found by the search function. Suppressed data can be identified by gaps in the sequence numbers.

Behavior

- Module sample numbers and stored time stamps can be searched.
- All module signals and groups can be searched; each has unique conditions to aid you in searching. Types are LA signals, LA groups, DSO channels, and disassembly.
- You can search for data differences and data equalities during compare operations.
- You can add an unlimited number of events.
- After adding an event, click the AND/OR button to alternate between logic expressions.

- The Radix for groups can be set independent from those used in Listing or Waveform windows.
- The polarity state set in the module setup applies to the search values.
- Searches on numeric disassembly groups with partial invalid bits (indicated by dashes) will match across the entire unqualified value. Set invalid bits to "don't care" values (X) for a proper match.
- Searches on numeric disassembly groups that contain all invalid bits always fail.

To Use

To open the Search Definition dialog box, open a data window and click  in the toolbar.


Click either the  Search Back or the  Search Fwd button to begin the search.

Click the Search Options... button to set the action that you want the logic analyzer to take when the search is successful.

NOTE. You can type the following meta-characters in the value box for text searches.

Character	Matches
	Zero or more characters
?	Any single character
< or >	One or more delimiters. A delimiter is any character that is not: a - z, A - Z, 0 - 9, _ (underscore) or - (dash).
%	Opcode mark. This character can only be used in a disassembly mnemonics column search.
space	One or more spaces
\	Makes the next character literal.

Right-click in the Search area to activate a short cut edit menu. The short cut edit menu allows you to add or delete data sources and events, to enable or disable searches and to cut, copy, or paste. Where you right-click in the Search area determines which menu selections are active.

Clicking in the Search area will activate a  icon. Holding your cursor over the arrow activates the short cut edit menu.

Related Topics

[Column Properties Tab \(see page 196\)](#)

[Waveform Properties \(see page 88\)](#)

[Compare Definition Dialog Box \(see page 190\)](#)

[Search Definitions \(see page 195\)](#)

Search Definitions Dialog Box

Overview

Use the Search Definitions dialog box to create new search definitions or load saved searches.

Behavior

Clicking the New button adds a new search to the selected data window. Clicking the Load button opens the Load Search dialog box where you can select and load searches. Selecting a search, and then clicking the properties button, opens the Define Search dialog box where you set the criteria for the search.

Related Topic

[Define Search Dialog Box \(see page 193\)](#)

Time Alignment Dialog Box

Use to align the time of data sources. The data source's alignment is offset by the adjusted time in all data windows. LA and associated [MagniVu \(see page 368\)](#) data sources are always offset together.

Saved data and current data are time-correlated by aligning their system triggers. You can manually adjust this alignment using the Time Alignment dialog box.

To open this dialog box, open a data window. From the Data menu, click Time Alignment.

Go To Dialog Box

Use to scroll the data window to a new position by selecting any current mark, column, or waveform.

To open the Go To dialog box, open a data window and then click the Go To toolbar button.

Related Topics

[Adding a Mark to a Data Window \(see page 137\)](#)

[Adding a Waveform \(see page 118\)](#)

Mark Opcode Dialog Box

Use this dialog box to place an opcode mark into a disassembly analysis. To do so, select an opcode from the list and click OK. To remove an existing opcode mark, click Undo Mark.

NOTE. *Load a support package to use this feature.*

About Data Properties

Provides information about the data source used for the active data window.

Column Properties

Use the Column properties tab to format a column in the Listing window.

- Click the Column drop-down list to view properties of another column. Note that changing columns applies changes that you made to the previous column.
- Color selection changes the color of text in the column.
- Column data can be displayed with user defined foreground and background colors by setting up the column to use TSF symbol file with color.
- The Show Column check box can be used to temporarily remove a column from the display, but still retain its properties.
- Column selection of Sample or Timestamp provides controls for reference points (for example, you may want to reference all time stamp values to the module trigger point rather than the beginning of data).
- Select the radix if the selected column is a data column. If you select Symbolic, controls appear to aid you in finding the symbol file.

NOTE. *Radix and Polarity selections for disassembly mnemonics cannot be changed. Polarity cannot be changed for any disassembly column.*

To open the Column properties tab, open a Listing window and double-click the column label.

Marks Properties

Use to format any mark in the current data window. You can also enter comments for any mark you have added.

To open the Marks properties tab, open a data window and double-click the mark.

NOTE. *Changing the selected mark applies any changes that you made to the previous mark.*

Listing Window Properties

Use to format the current Listing window.

- [Qualification gaps](#) in the Listing window are indicated by a gray horizontal bar above the first data after the gap.
- Data Font Size allows you to select the size of the text within the columns and the column labels.
- Click Acq != Ref to display data [differences](#) during data compare operations.
- Click Acq = Ref to display data [equalities](#) during data compare operations.


NOTE. *To display differences and equalities, define a valid data reference and enable the comparison.*

- Glitches are indicated by highlighted text of the complete sample for all radices except binary; in binary radix the actual bits containing the glitches are highlighted.

NOTE. *Select Glitches in the Storage box of the Setup window before acquiring data.*

- Background allows you to select the color for the data area background.
- Selecting a different color scheme resets the colors shown in the Column tab, Marks tab, and Listing Window tab to their default settings for that color scheme. Color schemes are defined in the Colors tab of the Options dialog box.

NOTE. *Default colors are set in the Defaults tab of the Application Options dialog box, available from the System menu.*

To display the Listing Window property tab from a Listing window, click  in the toolbar, and then click the Listing Window tab.

Related Topics

[Defaults Tab \(see page 378\)](#)

[Color Tab \(see page 377\)](#)

Disassembly Properties

Use to select the display format of disassembly groups.


Load a support package for this page to be active.

- If you have more than one disassembler loaded, you can select from all modules that have a disassembler currently loaded.
- You can select which disassembled data to display: [Hardware](#), [Software](#), [Control Flow](#), or [Subroutine](#). Other display options may be available depending on your support package.
- You can highlight lines appropriate for the current selection in the Show list box.

NOTE. *The highlight controls are disabled if the Mnemonics column is colored the same as another column.*

- You can select whether to continue disassembly analysis across storage [qualification gaps](#) in the acquisition data.

Other selections are support package dependent. Refer to your instruction manual for specific selections and choices.

To display the Disassembly Properties tab from a Listing window, click  in the toolbar, and then click the Disassembly tab.

Hardware Disassembly

Shows bus cycles in the order they were sampled. The disassembler displays certain cycle type labels in parentheses.

Refer to the support package manual for a list, and definition of, the labels.

Software Disassembly

Shows only executed instructions.

The displayed information resembles assembly language.

Control Flow Disassembly

Shows only calls, exceptions, returns, and branches; which are types of instruction that changes the flow of control.

Subroutine Disassembly

Shows only the first fetch or subroutine call and return instructions.

Error Message Dialog Box

This dialog box displays error messages, usually in relation to disassembly support and symbol table loading. The following are attributes of the Error Message dialog box:

- Holds up to 100 messages.
- The oldest messages are discarded when the message area fills.
- The most recent message is at the bottom.

NOTE. *Closing the Error Message dialog box discards all messages; minimizing does not.*

Define Suppression Dialog Box

The Define Suppression dialog box defines what module samples will be shown. This dialog box defines how many samples will be shown for only the data sources in this display.

Show Samples Dialog Box

The Show Samples Dialog Box shows preciously suppressed samples to be shown starting with the current mouse position.

Show by Samples

Making this selection indicates that the selected number of samples will be shown starting from my current mouse position within the view.

Show by Mark

Marking this selection causes all samples between the current mouse position and the selected mark to be shown.

Apply to Mark

Normally the data source pointed to by the mouse pointer is the only source that will be acted on. If this check box is selected the action will be applied to all data sources in this display.

Suppress Samples Dialog Box

The Suppress Samples dialog box suppresses samples from the view.

Suppress the <selection> <number> Samples

Making this selection indicates that the selected number of samples will be suppressed starting from my current mouse position within the view.

Suppress to Mark <selection>

Marking this selection causes all samples between the current mouse position and the selected mark to be suppressed.

Apply to all Data Sources in this Data Window

Normally the data source pointed to by the mouse pointer is the only source that will be acted on. If this check box is selected the action will be applied to all data sources in this display.

Sample Suppression Dialog Box

Use Sample Suppression to control how many samples are initially made available for display each time you acquire new data. Sample suppression does not affect the number of samples stored during acquisition.

You can control sample suppression in the following ways:

- Show All acquired samples. Makes all acquired samples visible immediately following acquisition.
- Show N samples around the trigger. Shows one block of samples with the trigger centered in the block of samples. All other samples are suppressed.
- Reuse existing sample suppression relative to. Retains the setting currently being used to show data in the Listing or Waveform window and applies it to newly acquired data. Suppression can be kept relative to the trigger point or beginning of memory.

Changes made to the dialog box do not take effect until new data is acquired. Following acquisition, use the Show or Suppress controls, in the Listing or Waveform windows, to view or suppress additional data samples.

To access this dialog box, click the Suppress . . . button in the toolbar in the LA Setup window.

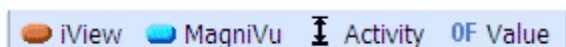
Waveform Window

Use the Waveform window to display and evaluate acquisition data. You may display both analog waveforms and digital waveforms. Each window contains a data area, a waveform label, marks, and several toolbars that allow you to measure and manipulate your waveforms:

Waveforms

The following waveforms can appear in a waveform display. Click the buttons for more information on each type of waveform.

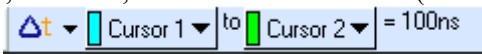

- [Single-Signal LA Waveforms](#)
- [Multi-Signal Busform LA Waveforms](#)
- [Multi-Signal Magnitude LA Waveforms](#)
- [DSO Waveforms](#)
- [Sample Waveforms](#)
- [Overlay Waveform](#)
- [Time Gaps](#)
- Waveforms can be dragged and dropped. Click the waveform label and drag it to where you want to display the waveform. An arrow shows where the waveform will be placed.
- Double-click a waveform label to open the Waveform dialog box.
- Add waveforms using the Add Waveform toolbar button.
- Remove waveforms by selecting the waveform label and clicking the Cut toolbar button. You can paste the removed waveform below any selected waveform.



- To quickly display the waveform from an external oscilloscope, click the iView button.
- To display MagniVu waveforms click the MagniVu button. To turn the iView or MagniVu display off, click the button again.
- To display the probe activity in the Waveform window, click the Activity button. Click the button again to turn the probe activity display off.
- To display the value of each group at Cursor 1, click the Value button. Click the button again to turn the value display off.

Marks

Marks are used to make time measurements, locate triggers, and mark specific waveform events for evaluation.

- Marks you can move include Cursor 1, Cursor 2, and any marks you add. The numbers 1 and 2 on the mark handles denote the cursor marks. Readouts for Cursor 1, Cursor 2, and the time difference (Delta Time) between the two are located on the Delta Time toolbar.  You can add any number of delta time toolbars by clicking the Add Delta Time button at the top of the Waveform window. 
- Marks on the top of the window are data area marks. Use moveable data area marks to move within the data area. You cannot move System Trigger, Module Trigger, or Begin and End marks.
- Marks on the bottom of the window are overview marks. These marks show where marks are located within the entire acquisition. Use moveable overview marks to move throughout the acquisition. For example, move Cursor 1 to a particular sample as shown in the Cursor 1 readout. Click the Go To toolbar button and select Cursor 1. You can also use the right mouse button on any overview mark and click Go To Mark.
- Double-click a mark to open the Properties dialog box and display the Marks tab. You can assign a unique name to the Mark in the Mark Name field.
- You can add marks using the Add Mark toolbar button. Remove an added mark by selecting it and using the Cut toolbar button. When you paste a mark, it appears in the center of the display.
- You can go to any mark using the Go To toolbar button.

System trigger



The system trigger is the reference point for the acquisition. Timing and location information is relative to the system trigger. Trigger marks cannot be moved.

The system trigger associated with the current data is called the active system trigger. The system trigger associated with saved data is called the reference system trigger. The active system trigger is indicated by a yellow T; the reference system trigger is indicated by a gray T.

The system trigger appears even if no data from the module that generated the system trigger is around it.

Module trigger

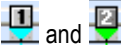



The point at which the module triggered. Trigger marks cannot be moved.

Begin data and end data



The start and end of a module's data record. These data marks cannot be moved.

Cursors 1 and 2 	Moveable marks used for visual reference and for data measurements.
User mark 	User-created marks. Use marks to make specific data easier to identify and find.

Data Area

- The Data area can be split by dragging the split box from the top of the vertical scroll bar. This allows you to compare waveforms.
- Several situations can cause long periods of time ([Time Gaps \(see page 205\)](#)) to elapse between acquired samples.
- Acquired glitches display directly on the affected waveform. Glitches display as a vertical bar behind the sample data in the glitch color. Glitches are drawn as if they were behind the waveform. You can disable the display of glitches and select their color in the Waveform window.
- Memory [differences](#) and [equalities](#) in logic analyzer waveforms are indicated by different colored text for each trace. Enable data comparison and select the color of highlighted data from the Waveform window.

Highlighted data shows where there are no differences between the two signals or groups being compared.

NOTE. Define the signals to compare, define a valid data reference, and enable the comparison in the LA Setup window before acquiring data and viewing differences and equalities.

Related Topics

[Waveform Properties \(see page 88\)](#)

[Waveform Labels \(see page 216\)](#)

[Marks Tab \(see page 197\)](#)

[Symbol Tables \(see page 341\)](#)

[Selecting Objects \(see page 206\)](#)

Time Gaps

Several situations can cause long periods of time to elapse between acquired samples:

- [Qualification gaps](#) in a single LA module record.
- Acquisition gaps between the last sample in one data source and the first sample in another data source.

As with any data samples, a horizontal vector is drawn from the last sample before a qualification gap to the first sample after the gap. If the values of these two samples are different, the transition is drawn at the sample after the qualification gap.

Qualification gaps within a module are compressed. If there are acquisition gaps when no modules are acquiring (including reference waveforms), this time is eliminated.

Cursor time values that fall within an acquisition gap are forced to the closest valid display point. If you place cursors on both sides of an acquisition gap, the displayed delta value is greater than the time between the two data points.

Selecting Objects

Selected Objects

You can select waveform labels, navigation marks, and data window regions in a Waveform or Listing window. Only one of these elements can be selected at one time; however, several waveforms can be selected as a single element.

For example, you can select three waveforms or a cursor, but you cannot select a waveform and a cursor at the same time.

Active Objects

There is a distinction between selected and active. While you can only select one element (or set of waveforms), there are element types for which one item is active at all times.

For example, there is always one active cursor (C1 or C2) and one active waveform. The active waveform is the top waveform of a selected set of waveforms. Measurements pertain to this waveform.

Overlay Waveform

Overview

Overlay waveforms provide you with the ability to visually compare two or more waveforms by dragging one waveform over the other. The overlay waveform can contain LA single-signal, DSO, external, and clock waveforms.

LA waveforms have a horizontal resolution of one pixel per display point. A displayed point has three possible states: high, low, or transition. When the sample period and time per division are set so that one pixel is exactly one sample, every displayed sample is either high or low and may show a transition vector. When the Time/Div is increased, the analyzer stretches out the waveform by padding between the samples. At slower Time/Divisions, the data is compressed.

Vertical lines or vertical vectors are drawn at high-to-low and low-to-high transitions.

Acquired glitches are displayed directly behind the affected data points. Glitches are displayed as a vertical line in the glitch color. Glitches are drawn as if they were behind the waveform. You can disable the display of glitches using the Waveform Window tab, which is located in the Waveform Properties dialog box.

Behavior

For DSO and external waveforms, each horizontal, pixel-wide time slice has either a point or a vertical line to represent the vertical value of the acquired data sample. If the time per division is fast enough that each display point represents one or fewer acquired points, the vertical value is represented by a point. If the time per division is slower and multiple acquired data samples need to be represented within a pixel width, the vertical value is represented by a vertical line whose length and offset are determined by the acquired data values.

When the time per division is decreased, waveforms are horizontally expanded, using (sine x)/x interpolation to create display points between acquired sample points. A vertical value is calculated for every horizontal time point between the acquired data points.

To Use

You can select whether to display range readouts for the selected waveform, to display all DSO and external waveforms range readouts, or to not display range readouts at all. You can also select the readout color. For each waveform, two readouts display the maximum and minimum vertical input voltage settings. The readouts may contain symbols indicating AC or Ground coupling, 50 Ohms input impedance, and the bandwidth limit. The ground line is indicated as a horizontal dotted line. The trigger threshold is indicated with a T and an up arrow if the threshold is above the maximum voltage or a down arrow if the threshold is below the minimum voltage.

Waveform size is dependent on the signal size and the input voltage range set in the DSO setup window. The entire input voltage range is used to set the default minimum and maximum values of the vertical range. Set the vertical range values from the Waveform Properties dialog box by clicking the Waveform tab, and then clicking the Options button.

To display no more than one data sample per display point, add a sample waveform from the same data source and decrease the time per division until you can see individual clock tick marks.

Related Topics

[Creating an Overlay Waveform \(see page 135\)](#)

[Removing Waveforms from an Overlay Waveform \(see page 139\)](#)

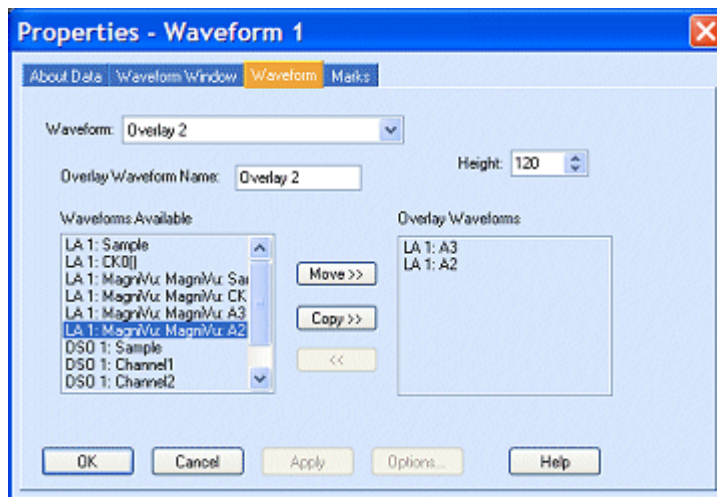
[Waveform Properties \(see page 88\)](#)

[Sample Waveforms \(see page 215\)](#)

Overlay Waveform Properties

Overview

The Waveform tab displays overlay waveform properties when you double-click the label of an overlay waveform in the Waveform list box. The following graphic displays the resulting content of the Waveform tab.



Behavior

The following are properties associated with overlay waveforms:

- **Overlay Waveform Name:** permits you to rename overlay waveforms.
- **Waveforms Available** lists all of the individual waveforms and overlay waveforms. You can move and copy waveforms in this list to the Overlay Waveforms list using the Move and Copy buttons. You can also remove buttons from the Overlay Waveforms list and return them to the Waveforms Available list using the << button.
- **Overlay Waveforms** lists the waveforms currently in the selected overlay waveform group. You can move and copy waveforms from the Waveforms Available list to this list using the Move and Copy buttons, respectively. You can also remove waveforms from the Overlay Waveforms list and return them to the Waveforms Available list using the << button.

To Use

To change waveform properties, from the System window click a Waveform window icon, double-click the label of a waveform, and then click the Waveform tab.

Related Topics

[Waveform Properties \(see page 88\)](#)

[Individual Waveform Properties \(see page 209\)](#)

[Individual Waveform Properties within an Overlay Waveform \(see page 211\)](#)

[Waveform Window Properties \(see page 89\)](#)

[Color Settings \(see page 377\)](#)

[Waveform Options \(see page 216\)](#)

[Measurement Setup \(see page 396\)](#)

Setting the Overlay Waveform Offset

1. In the System window, click the Waveform window icon.
2. Double-click the desired waveform label to display the Waveform Properties dialog box, and then click the Waveform tab.
3. Set the desired pixel value in the Offset list box and click the Apply button.

NOTE. *Offset can only be specified for waveforms that are members of an overlay waveform.*

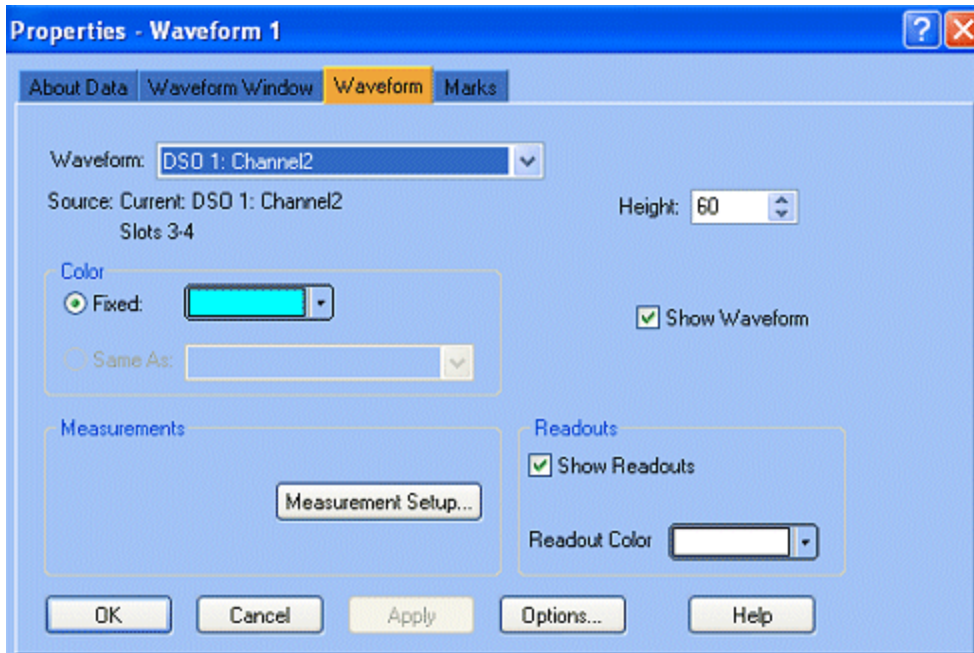
Alternate Method

1. In the System window, click the Waveform window icon.
2. Highlight the desired waveform label.
3. Use the vertical scroll bar to set the waveform offset.

Individual Waveform Properties

Overview

The Waveform tab displays individual waveform properties when you double-click the label of a single waveform in the Waveform list box. The following graphic displays the resulting content of the Waveform tab.



Behavior

The following are properties associated with individual waveforms:

- **Source.** displays the data source of the selected waveform. The source file (or current system), module name, and signal information are displayed. Depending upon the way the signals were originally added, the information will be a group-based signal name (for example, Address (31-0)), a single signal name, or a probe-based name (A2-4, A3(7-0)). If the module is a secondary module of a merged pair, an S is prepended to probe-based names. If the module is part of a three-way merge, S and S2 are prepended to probe-based names. If the module is part of a five-way merge, S, S2, S3 and S4 are prepended to probe-based names. No text is displayed for multiple waveform selections.
- **Color.** permits the selection of either a fixed waveform color or a waveform color that is the same as another waveform, which is designed to promote comparison of waveforms. If you are selecting a fixed color, you can select a color from the list. Select the color scheme using the Waveform window tab. Select the colors of each color scheme by clicking Options from the System menu and then clicking the Color tab.
- **Radix.** controls the radix for LA group data displayed as busform or cursor value readouts. Radix is only enabled when a single LA group waveform is selected. Set the default radix by clicking Options from the System menu and then clicking the Defaults tab.
- **Show Waveform.** temporarily hides a waveform without deleting it. This check box determines if the waveform is visible in the data window. If a waveform is not visible, you cannot search for it.
- **Use Polarity.** enables the display of waveform signal polarity as defined in the Setup window. This check box is only enabled for LA waveforms.

- **Readouts.** determines whether waveform readouts are displayed and their color. Readout options only display for logic analyzer, DSO and external waveforms.
- **Measurements.** displays the user defined measurement selections for DSO waveforms. Click the Measurement Setup button to modify the list of selected measurements.

To Use

To change waveform properties from the System window, click a Waveform window icon, double-click the label of a waveform, and then click the Waveform tab.

Related Topics

[Waveform Properties \(see page 88\)](#)

[Overlay Waveform Properties \(see page 208\)](#)

[Individual Waveform Properties within an Overlay Waveform \(see page 211\)](#)

[Waveform Window Properties \(see page 89\)](#)

[Color Settings \(see page 377\)](#)

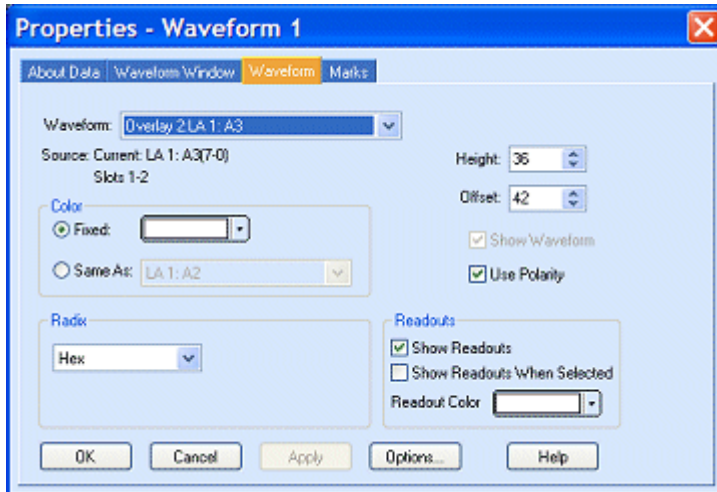
[Waveform Options \(see page 216\)](#)

[Measurement Setup \(see page 396\)](#)

Individual Waveform Properties within an Overlay Waveform

Overview

The Waveform tab displays the properties of an individual waveform within an overlay waveform, when you double-click the label of a waveform, from within an overlay waveform, in the Waveform list box. The following graphic displays the resulting content of the Waveform tab.



Behavior

The following are properties associated with individual waveforms within an overlay waveform:

- **Offset.** allows you to visually compare two or more waveforms by specifying their vertical position in the Waveform window.
- **Source.** displays the data source of the selected waveform. The source file (or current system), module name, and signal information are displayed. Depending upon the way the signals were originally added, the information will be a group-based signal name (for example, Address (31-0)), a single signal name, or a probe-based name (A2-4, A3(7-0)). If the module is a secondary module of a merged pair, an S is prepended to probe-based names. If the module is part of a three-way merge, S and S2 are prepended to probe-based names. No text is displayed for multiple waveform selections.
- **Color.** permits the selection of either a fixed waveform color or a waveform color that is the same as another waveform, which is designed to promote comparison of waveforms. If you are selecting a fixed color, you can select a color from the list. Select the color scheme using the Waveform window tab. Select the colors of each color scheme by clicking Options from the System menu and then clicking the Color tab.
- **Radix.** controls the radix for LA group data displayed as busform or cursor value readouts. Radix is only enabled when a single LA group waveform is selected. Set the default radix by clicking Options from the System menu and then clicking the Defaults tab.
- **Show Waveform.** check box determines if the waveform is visible in the data window. If a waveform is not visible, you cannot search for it. Use this box to temporarily hide a waveform without deleting it.
- **Use Polarity.** check box enables the display of waveform signal polarity as defined in the Setup window. This check box is only enabled for LA waveforms.
- **Readouts.** determines whether waveform readouts are displayed, as well as their color. When selected, the Show readouts when selected check box allows waveform readouts to display only when the

associated waveform label is highlighted. Note that Readout Options only displays for logic analyzer, DSO and external waveforms.

- **Measurements.** displays the user defined measurement selections for DSO waveforms. Click the Measurement Setup button to modify the list of selected measurements.

To Use

To change waveform properties, from the System window click a Waveform window icon, double-click the label of a waveform, and then click the Waveform tab.

Related Topics

[Waveform Properties \(see page 88\)](#)

[Individual Waveform Properties \(see page 209\)](#)

[Overlay Waveform Properties \(see page 208\)](#)

[Waveform Window Properties \(see page 89\)](#)

[Color Settings \(see page 377\)](#)

[Waveform Options \(see page 216\)](#)

[Measurement setup \(see page 396\)](#)

Multi-Signal Busform LA Waveforms

Busform waveforms can contain group value readouts. If the width between transitions is sufficient, a value is displayed. If the width is not sufficient, no readout is shown in that segment. The readout is displayed in the selected radix.

Acquired glitches are displayed directly behind the affected waveform. Glitches are displayed as vertical lines in the glitch color. Glitches are drawn as if they were behind the waveform and any readouts. You can disable the display of glitches using the Waveform Window tab, which is located in the Waveform Properties dialog box.

TIP. *To display no more than one data sample per display point, add a sample waveform from the same data source and decrease the time per division until you can see individual clock tick marks.*

Waveform height is set from the Waveform tab, which is located in the Waveform Properties dialog box.

Waveforms can be displayed with user defined foreground and background colors by setting up the busform to use a TSF symbol table with color.

Related Topic

[Sample Waveforms \(see page 215\)](#)

Multi-Signal Magnitude LA Waveforms

Overview

In Magnitude mode, the vertical placement of a data point in waveforms represents the numeric value of the group. By default, the possible range of values (determined by the group width) is scaled to the selected waveform size.

Behavior

When the time per division is decreased, waveforms are horizontally expanded, using linear interpolation to create display points between the acquired sample points. A vertical value is calculated for every horizontal time point between the acquired data points.

When the time per division is increased, waveforms are horizontally compressed and decimated. The minimum and maximum values are used as the endpoints of the vertical line that represents the data values for that time point. The cursor readout displays the value of the top endpoint.

If you compress the waveform, the minimum and maximum values at each display point are displayed to create a vertical line at each horizontal data point.

To Use

You can change the waveform minimum and maximum values by accessing the Waveform tab, located in the Waveform Properties dialog box, and clicking the Options button. You can also select the radix used in cursor measurements.

TIP. *To display no more than one data sample per display point, add a sample waveform from the same data source and decrease the time per division until you can see individual clock tick marks.*

Waveform height is set in the Waveform tab of the Waveform Properties dialog box.

Waveforms can be displayed with user defined foreground and background colors by setting up the busform to use a TSF symbol table with color.

Related Topic

[Sample Waveforms \(see page 215\)](#)

Sample Waveforms

You can select sample waveforms in the Add Waveform dialog box. Each module has its own sample waveform. A sample waveform consists of a row of short vertical marks on the time axis at each display point that represents an actual acquired sample for the module.

When the display is expanded, points without a sample are empty. When the display is compressed, the sample waveform looks like a thick horizontal line.

Two time stamp values are displayed at the edges of the sample waveform.

If the sample waveform is the active waveform, the Measurement Bar readouts are empty.

The waveform size is fixed.

Single-Signal LA Waveforms

Single-signal LA waveforms have a horizontal resolution of one pixel per displayed point. A displayed point has three possible states: high, low, or transition. When the sample period and time per division are set so that one pixel is exactly one sample, every displayed sample is either high or low and may show a transition vector. When the Time/Div is decreased, the analyzer stretches out the waveform by padding between the samples. At slower Time/Divisions, the data is compressed.

For example, if the time per division is set so that two acquired samples must be represented in each horizontal pixel width and both of the acquired samples in a pixel are high, the displayed sample is high. If both samples are low, the displayed sample is low. If the two acquired samples are a high and a low, in either order, the display will be a vertical line representing both samples. At even slower time per division settings, more acquired samples are represented by each displayed point.

Vertical lines or vertical vectors are drawn at high-to-low and low-to-high transitions.

Acquired glitches are displayed directly behind the affected data points. Glitches are displayed as a vertical line in the glitch color. Glitches are drawn as if they were behind the waveform. You can disable the display of glitches using the Waveform Window tab, which is located in the Waveform Properties dialog box.

TIP. *To display no more than one data sample per display point, add a sample waveform from the same data source and decrease the time per division until you can see individual clock tick marks.*

Waveform height is set from the Waveform tab, which is located in the Waveform Properties dialog box.

Waveforms can be displayed with user defined foreground and background colors by selecting "Color Same As" on a LA busform waveform, and then setting up the busform to use a TSF symbol table with color.

Related Topic

[Sample Waveforms \(see page 215\)](#)

Waveform Labels

Overview

Waveform labels serve as handles to mark the waveform location and name of each waveform. The label allows you to adjust the position and other properties of waveforms.

The waveform name is the logic analyzer group or signal name, the oscilloscope channel name, or the name for a special waveform type. The name is prepended with the module name when there are multiple modules, the waveform is included as part of a waveform overlay, and with the file name of the data source file if the waveform is a reference waveform. If the name will not fit in the label, characters on the left, which are the most likely to be redundant, are clipped until the name fits in the label.

To Use

You can select and drag one or more waveform labels. When you drag a label, a drop marker indicates where the label will be dropped. If there are waveforms beyond the top or bottom edge of the data area, dragging a label past that boundary will scroll the display.

You can change the width of the label area by dragging the border between the label area and the data area. You can change the height of a label by clicking and dragging its bottom edge. Double-clicking a waveform label opens the Properties dialog box and displays the Waveform tab, which contains the settings for that waveform.

Waveform Options

Waveform Options allows you to set waveform characteristics that are specific to a waveform type.

Waveform Limits sets the minimum and maximum values that will be displayed in the selected waveform.

Behavior

- Max Value sets the maximum value displayed in the waveform.
- Min Value sets the minimum value displayed in the waveform.
- For DSO waveforms, Lock to Setup determines if the Max and Min values are reset when the DSO setup parameters are changed. When checked, the values are reset to the maximum and minimum values possible with the current range and offset settings. Checked is the default.
- For LA group waveforms, Lock to Setup determines the minimum and maximum values based on the number of signals in the group. When not checked, you can specify a subset range of the group min and max.

LA Waveform Style selects the style of displayed LA data. These controls are disabled unless a single LA group waveform is selected.

- Busform displays LA group data as a bus form.
- Magnitude displays LA group data as a graph display (Magnitude mode).
- Signed data selects whether acquisition data is interpreted as signed or unsigned. Signed data is only active for magnitude style LA waveforms.
- Draw Interpolation inserts vectors between data points, thereby approximating a magnitude waveform.

To Use

To change waveform options from the System window, click a waveform data window icon, double-click a waveform label, click the Waveform tab, and then click the Options button.

Related Topics

[Setting Waveform Options \(see page 125\)](#)

[Setting the Waveform Style \(see page 83\)](#)

[Setting the Waveform Limits \(see page 84\)](#)

Histogram Window

Overview

Use the Histogram window to set up, capture, and display performance analysis data from an LA module group, counter, or timer. Each Histogram window displays address ranges and the corresponding counts, percentages, and histogram bars based on the defined ranges and selected data source.

Before you can acquire and display histogram data, first set up the LA module using the LA Setup and LA Trigger windows. Use the Setup window to set up the groups, sampling, and memory requirements. Use the Trigger window to set up a trigger program, counters, and timers. You should configure the Setup window before the Trigger window, because some of the Setup window settings affect Trigger window selections.

Use the New Data Window wizard to create a Histogram window for the first time.

Statistics

The Histogram statistics provide an overview of the data.

- The number of displayed lines depends on the data source. Statistics for a group use only one line. Statistics for counters or timers use two lines.
- The Percentages Based On list shows the number of data samples depending on whether you select [All Samples](#) or [Matched Samples](#).
- If the data source is a counter or timer, the second line lists the minimum, maximum, and average values of the counter or timer.

Header Control

Use to resize columns, sort ranges, and specify the magnification value for the histogram bars.

- Resize columns by dragging and dropping the column separators in the Histogram Header Control.

NOTE. *Dragging a column separator behind another column separator may hide a column.*

- When ranges are displayed as symbols, you can sort the ranges by ascending or descending alphabetical order by clicking the Range heading.
- When ranges are displayed as bounds, you can sort the ranges by increasing or decreasing lower bounds by clicking on the Range heading.
- Click the Count heading or the Percentage heading (%) to sort between increasing and decreasing counts.
- Change the magnification of the histogram bars by clicking on the bar heading and selecting a new value from the displayed list.

Data Area

Displays ranges, counts, percentages, and histogram bars.

- You can resize the data area by turning the toolbar and the statistics on or off.
- You can split the Histogram window horizontally by dragging the split box from the top of the vertical scroll bar.
- All ranges are displayed in columns. When ranges are used as a timer, the bounds include a time unit suffix.
- A clip indicator (<) in a text column indicates that only part of the row can be displayed. To view the entire row, you can resize the column by dragging the column header.
- Counts, percentages, and histogram bars are updated as new data becomes available.
- You can sort ranges, counts, and percentages by clicking on the column headers.
- Changing the ranges in the Histogram Properties updates the new ranges with zero counts.

Related Topics

[Defining Histogram Ranges \(see page 225\)](#)

[Data Source Property Page \(see page 221\)](#)

[Histogram Range Properties \(see page 222\)](#)

[Histogram Window Properties \(see page 220\)](#)

All Samples

The total number of samples analyzed that matched a define range and all samples that fell outside of the defined ranges.

Matched Samples

The total number of samples analyzed that matched a defined range. These samples exclude any samples outside of the defined ranges.

Histogram Window Properties

Overview


Use the Histogram Window properties tab to change and view the properties of the Histogram window.

Behavior

- Range Labels control the labels in the Range column in the data area. When the ranges are defined from a symbol file, select Symbols or Bounds. The Symbols selection is unavailable when the ranges are not defined in a symbol file. There is no limit to the number of symbolic ranges for histogram data.
- Data Fonts Size selects the size of the data in the Histogram data area. You can change the data font size as necessary to view the data.
- Radix determines the format of range labels when Range Labels is set to Bounds. The Radix is only present when the data source is a group or a counter.
- Background selects the color of the data area background.
- Foreground selects the color of the text in the data area.
- Bar color selects the color of the histogram bars.
- Selecting a different color scheme resets the colors shown. Color schemes are defined in the Colors page of the Options property sheet.
- Data is accumulated until you clear the data.
- You can analyze live data or saved data.
- You can use up to 32 bits of data in an address group.
- You can enable or disable group polarity.

NOTE. *Default colors are set in the Defaults page of the Applications Options property sheet, available from the System menu.*

To Use

To display the Histogram Window properties, open a Histogram window, right-click the window and select Properties, or click  (the Property toolbar button), and then click the Histogram Window tab.

Data Source Properties

Overview

Use the Data Source properties tab to display information about the current [data source](#) or to select a new data source for the Histogram window.

Behavior

If you change the groups in the Setup window or if you change counters or timers in the Trigger window, the histogram data source may become invalid. The data analysis will stop until you define a new data source or until you restore the current data source.

The Use Polarity box controls whether or not polarities are applied to the group data.

Changing the histogram data source generates new range definitions; existing counts are lost.

To Use

To display the Data Source properties, open a Histogram window, click  (the Property toolbar button), and then click the Data Source tab.

Select the LA module from the drop-down list. (If the data source is not included in the drop-down list, click Add Data Source to add data from a saved setup.

Click either the [Group](#) or Counter/Timer buttons to use for the histogram data. The name of the data is displayed in the drop-down list next to the buttons. The button selections are active when valid data is available based on the groups and trigger setups.

Histogram Data Source

A histogram data source consists of histogram data from an LA module. The data can be from a valid group or from a counter or a timer defined in the Trigger window. DSO modules and MagniVu data are not allowed as data sources for Histogram windows.

Histogram Group

A group can consist of a single data channel or up to 32 data channels. Groups are defined in the Setup window.

Ranges Properties

Overview


Use the Ranges properties tab to define the data ranges for the Histogram window. To define ranges, specify a data source for the Histogram window.

Behavior

- The Default Ranges button creates default histogram ranges based on the data source.
- The Define Ranges Using list specifies how the ranges will be defined: [Linear Generation](#), [Log Generation](#), or [Symbolic](#).
- The Symbol File button opens the Load Symbol File dialog box to load a range symbol file when using symbolic range generation. Use the Symbol File list to select one of the loaded symbol files.

- The Number of Ranges specifies the number of ranges available with Linear Generation or Log Generation. When used with Linear Generation, the number of ranges can be changed. The Bound1 and Bound2 controls determine the maximum number of ranges.
- The Radix controls the format of the Bound1 and Bound2 controls. In Log Generation, the Radix also controls how ranges are generated. Changing the radix also changes the display radix in the Histogram Window properties tab. The Radix control is not present when the data source is a timer.
- The Bound1 and Bound2 controls limit the overall boundaries or size of the ranges. The limits depend on the data source. When used with a symbol file the bounds are limited by the smallest and largest symbol file boundaries. When used with linear or log generation, the lowest bound is 0 and the highest bound is limited by the data source size.
- The Range Size shows the size of a single range when used with Linear Generation.
- Use Time Units to format the timer values.

To Use

To display the Ranges properties, open a Histogram window, click  (the Property toolbar button), and then click the Data Ranges tab.

Related Topics

[Defining Histogram Ranges \(see page 225\)](#)

[Load Symbols Options Dialog Box \(see page 348\)](#)

Linear Generation

The histogram ranges are evenly distributed from the highest range boundary to the lowest range boundary.

Log Generation

The histogram ranges are distributed over a logarithmic scale.

Symbolic Range Generation

The histogram ranges are defined in a range symbol file. The highest and lowest ranges depend on the maximum and minimum boundaries for the ranges defined in the symbol file.

Histogram Scale Dialog Box

Use this dialog box to change the scale (magnification) of the histogram bars for the current Histogram window. You can change the scale factor from the Histogram window or from this dialog box.

To open the Histogram Scale dialog box from the Histogram window, go to the View menu and click Scale.

Histogram Add Counts Dialog Box

Use this dialog box to add data from an LA module or from a saved setup. Select the data from a module in the displayed list, or click Add Data Source to add data from a saved setup. Select a group, a counter, or a timer from the group list.

To open the Histogram Add Counts dialog box from the Histogram window, go to the Data menu and then click Add Counts.

Export Histogram Dialog Box

Use this dialog box to export Histogram window information to a file. Specify the export file path and the rows of the histogram information. The exported data will be saved in the specified format. The dialog box uses any export options specified in the Export Histogram Options Dialog Box.

To open the Export Histogram dialog box from the Histogram window, go to the File menu and then click Export Histogram.

Related Topics

[Exporting Histogram Data \(see page 72\)](#)

[Export Histogram Options Dialog Box \(see page 224\)](#)

Export Histogram Options Dialog Box

Use this dialog box to customize the export histogram operation. You can specify file formatting options and how to export the bounds. You can select the label button if you want to export the bounds as a string ("00-FF") or click the numbers button to export the bounds as two delimited numbers. You can also include headings with the exported data.

To open the Export Histogram Options dialog box from the Histogram window, click Export Histogram from the File menu, and then click the Options button.

Measuring Histogram Data

Overview

There are two basic ways of using the Histogram window to analyze data. One is to provide a graphic overview of the address activity of software execution. The other is to use counters or timers to measure specific events.

If you want to use counters or timers to measure events, set up the counters or timers in the Trigger window. You can then select the data source in the New Data Window wizard when you create a Histogram window or select the source from the Data Source properties tab for an existing Histogram window.

Viewing Address Activity

To provide an overview of the address activity of a software routine, define a set of ranges (numeric, logarithmic, or symbolic). Then you can set up the Trigger window to look for data within the defined ranges. The data is processed and displayed in the Histogram window for the current acquisition.

If you set up the logic analyzer to repetitively acquire data, the sampling errors will decrease over time as you acquire more data. The resultant data provides a stable display in the Histogram window where you can view the overall activity of the software.

You can use this method of analyzing data to determine which sections of code are being accessed or not accessed. You can also determine the time spent in a routine relative to other areas of code.

Measuring Counter or Timer Events

To measure an event, define a starting point, a target event, a counter or a timer, and a stopping point in the Trigger window. Use the Data Source properties tab to select the counter or timer as the data source for your analysis. When you acquire data, the Histogram window displays the minimum, maximum, and average value of the counter or timer data.

Related Topics

[Defining Histogram Ranges \(see page 225\)](#)


[Selecting the Histogram Data Source \(see page 226\)](#)

Defining Histogram Ranges

1. In the System window, click the Histogram data window icon.
2. From the Edit menu, click Properties.
3. Click the Ranges tab in the Properties dialog box.

4. From the Define Ranges Using list, click Linear Generation, Log Generation, or Symbols.
5. If using Symbols, select a symbol file.
6. Select a radix.
7. Specify the bounds.
8. Click OK.


TIP. Define the data source before you define the ranges.

Click  to open the Properties dialog box.

Click Default Ranges to return to the default range settings.

Selecting the Histogram Data Source

1. In the System window, click the Histogram data window icon.
2. From the Edit menu, click Properties.
3. Click the Data Source tab in the Properties dialog box.
4. Select the new data source from the list or click Add Data Source to add data from a saved data source.
5. Select the group or a Counter/Timer source if defined in the Trigger window.
6. Click OK.

TIP. Click  to open the Properties dialog box.

Set up the groups in the Setup window and the counters or timers in the LA Trigger window before you select a histogram data source.

Source Window

Overview

The Source window displays source code that you wrote, correlated with the executed (disassembled) code acquired by the logic analyzer and displayed in the Listing window. You can monitor actual code execution and correlate it directly with your source code statements. The logic analyzer links the Source and Listing windows, and provides additional tools to help you view the code and data.

Behavior

The Listing window displays an execution trace of the object code derived from the source files. The relationship between the acquisition data and the specific source is determined by examining the debug information stored in the executable file that was created by code creation tools such as compilers, linkers and loaders. The specific file being displayed, the cursor positions, and user-defined marks in the file are automatically updated as you use the controls in the Source window and the associated Listing window.

The source statements in each of the source files displayed in the Source window have associated address ranges that define the memory locations occupied by the microprocessor instructions that implement the statement. The address ranges determine the mapping from samples in the Listing window to statements in the Source window. The address ranges must be absolute. The Source window does not support code that is dynamically relocatable. The address group that you use depends on whether you use a disassembler. The disassemblers typically set up address groups for the specific support package.

Because Source windows interact with associated Listing windows, you may want to display both windows at the same time. Moving the cursors in the Source window moves the cursors in the associated Listing window, and moving the cursors in the Listing window moves the cursors in the Source window.

Source Window Structure

The Source window consists of three main components; the Source toolbar, the Source control bar and the Source data area.

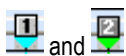
- The Source toolbar provides access for tools to change the appearance of the data area and to search for data. A Scan Listing field specifies the search direction in the Listing window when you move the cursors in the Source window.
- The Source control bar provides means for stepping through data, stepping between marks, and scrolling through data. These controls directly affect the active cursor in the associated Listing window and indirectly affect the active cursor in the Source window.
- The source data area displays the contents of the source file. This area is also used to display diagnostic messages if the Source window is unable to display a source file. When the window displays a source file, the header displays the path of the file.

Marks and Cursors

Marks and cursors identify specific statements in source files. Marks consist of horizontal lines and may include highlighted blocks of text signifying individual source statements.

- Marks you can move are: Cursor 1, Cursor 2, and any marks you add. The numbers 1 and 2 on the mark handles denote the cursor marks. Readouts for Cursor 1 and Cursor 2 are located at the top of the window. The numbers in the readouts indicate the line number position of the cursor in the source file.
- Only one of the two cursors is active at a time. An active cursor appears as a solid horizontal line; an inactive cursor appears as a dashed line.
- Marks on the left side of the window are data area marks. These marks are in the part of the source file that is currently visible in the window. Use data area marks to move within the data area.
- Marks on the right side are overview marks. These marks show where marks are located within the entire source file. Use overview marks to move throughout the source file.
- Marks are only visible in the currently displayed source file. Marks and cursors in other source files are not shown in the overview area.
- Double clicking a mark opens the Marks properties tab. You can assign a unique name to the Mark in the Mark Name field.
- You can add marks using the Add Mark toolbar button. Remove a mark by selecting it and using the Cut toolbar button. When you add or paste a mark it appears in the center of the display.
- You can add a mark at a specific source statement by right-clicking the source statement and then selecting Add Mark Here from the menu. If the line in the source window contains multiple source statements, you can add marks to the individual statements.
- You can move to any mark, including those in other source files, by using the Go To toolbar button.
- Marks are saved in the Source window with system setups so they can be made available for use when you load a saved system.

Cursors 1 and 2



Moveable marks used for visual reference and for data measurements.

User mark



User-created marks. Use marks to make specific data easier to identify and find.

Related Topics

[Listing Window \(see page 185\)](#)

[Moving Through Source Files \(see page 229\)](#)

[Symbol Files \(see page 341\)](#)

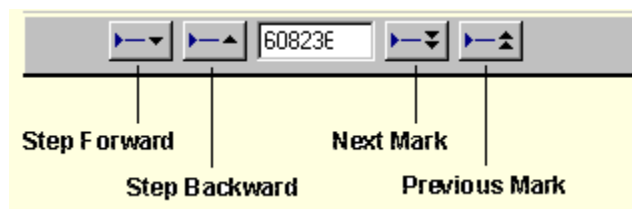
[Symbols Dialog Box \(see page 346\)](#)

Moving Through Source Files

Overview

You can move through source files from the Source window or from the Listing window. Moving a cursor in one window will cause the corresponding cursor to move to a corresponding location in the other window.

Use the Step Forward or the Step Backward buttons to step through source statements in execution order in the Source window. You can also use the Next Mark or the Previous Mark buttons to jump to the next or previous executed source statement that has a user-defined mark.



Use the cursor controls in either window to move the cursors. You can also move the cursors in either window by clicking and dragging the cursor handles or by adjusting the cursor list box controls.

For more information on moving the cursors, click one of the buttons below.

- [Source-Relative Cursor Positioning](#)
- [Listing-Relative Cursor Positioning](#)

Behavior

There are situations when you move the cursor in the Listing window to a location where there is no corresponding source statement. The cursor in the Source window remains at the current location and changes the color of the Source window to signify an [uncorrelated state](#) exists between the Source and Listing windows. When this happens click the Step Forward or the Step Backward buttons to search for an executable source line in the Listing window; this correlates the active cursors in both windows.

Clicking the Step Forward or Step Backward buttons forces the Listing window to search for a data sample that corresponds to the next or previous line of executed source code. The line of executed code may be in a different source file. If this happens, the Source window will display the required file or prompt you for a path name. The ability to locate source files depends on the Search Path List and Suffix List defined in the Source Files properties tab.

Related Topics

[Step Forward and Step Backward Buttons \(see page 231\)](#)

[Next Mark and Previous Mark Buttons \(see page 231\)](#)

[Active Cursor Readout \(see page 232\)](#)

[Source Files Properties \(see page 237\)](#)

Uncorrelated State

The Source window enters an uncorrelated state when the active Source window cursor is moved to a statement that was not executed and captured in the acquisition memory. An uncorrelated state also occurs when the active Listing window cursor is moved to a sample with an address for which there is no corresponding symbol information in the loaded symbol file.

When the Source window becomes uncorrelated, the window displays text in a different color: the uncorrelated color. You can change the color of the uncorrelated state in the Source Window properties tab.

To return the Source window to a correlated state, click the Step Forward or the Step Backward button in the Source window.

Scan Listing Mode

Overview

The Scan Listing box is used to specify which direction to search in a Listing window when you move the cursor in the Source window.

Behavior

The following options for the Scan Listing field are available:

Later – The Listing window scans samples executed after the sample identified by the active cursor.

Earlier – The Listing window scans samples executed before the sample identified by the active cursor.

Ask – When you change the position of the active cursor in the Source window, the Scan Listing dialog appears and prompts you to specify the scan direction.

Auto – The Source window automatically selects the scan direction in the Listing window. The scan direction is based on the relative position of the new and old cursor.

Moving the cursor between statements in the Source window does not necessarily move the cursor in the same direction in the Listing window. This can happen, for example, if the statement in the new location was executed both before and after the statement in the old location, such as when the statement is in a loop.

Step Forward and Step Backward Buttons

Overview

Use the Step Forward and Step Backward Buttons to trace the order of execution of source statements. Click the Step Forward button to move to the next executed source statement. Click the Step Backward button to move to the previous executed source statement.

Behavior

When you click the Step Forward or Step Backward buttons, the Source window tells the Listing window to search forward or backward beginning at the active Listing cursor for the next or previous executed source statement. When a match is found, the Listing window positions the active Listing cursor to the matching sample. The Source window converts the address from that sample to a file name and line number, and then updates the active Source cursor position to the corresponding source statement. The statement may be located in a source file that is different than the original displayed file. In this case, the new file is displayed and the cursor is placed on the proper statement.

A single line in the Source window may consist of more than one statement on a single line, for example:

```
for ( i=0; i < NUM_STATES; i++ )
```

The ability of the Source window to discriminate between multiple statements on the same line depends on the amount of information provided by the code-generation tools. If there is enough information to identify multiple statements per line, the Source window cursors will include character highlighting to identify individual statements on a line.

Related Topics

[Moving Through Source Files \(see page 229\)](#)

[Next Mark and Previous Mark Buttons \(see page 231\)](#)

Next Mark and Previous Mark Buttons

Overview

Use the Next Mark and Previous Mark buttons to move the active cursor in the Source window to the next or previous user-defined mark position (if any marks have been defined). Next and previous refer to the execution sequence and not to the position of the mark in the window.

Behavior

The Next Mark and Previous Mark buttons operate similarly to the Step Forward and Step Backward buttons. However, rather than stepping through every executed statement, you can define marks in the

Source window as breakpoints and then step between marks in execution order to move through the source code. You can use the Next Mark and the Previous Mark buttons to get around loops or acquired code that you are not interested in viewing.

When you click the Next Mark or Previous Mark buttons, the Source window tells the Listing window to search for addresses corresponding to the marked source statements. When a corresponding sample is found, the Listing window positions its active cursor to the matching sample. The Source window converts the address to a file name and line number, updates the active Source cursor position, and then displays the corresponding source statement.

If the code generation tools provide column information for symbol files, you can mark individual statements of multiple statement lines in the source code. You can then use the Next Mark and Previous Mark buttons to step between the marks and highlight the individual statements.

The Next Mark and Previous Mark buttons are not active if there are no marks defined in the Source window.

Related Topics

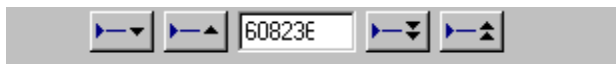
[Moving Through Source Files \(see page 229\)](#)

[Step Forward and Step Backward Buttons \(see page 231\)](#)

Active Cursor Readout

Overview

The Source window Active cursor readout displays the address of the source statement at the active cursor. The address shown is the low address bound of the source statement as indicated by the loaded symbol file, not the value of the address group in the listing window. The radix of the address is always hexadecimal. If you position the cursor on a nonexecutable statement, such as a comment or white space, the address of the next executable statement is displayed.



Behavior

You can also use the Active cursor readout to jump to a new location in the source file. Enter an address in this field and the active cursor moves to the corresponding statement. If you enter an address that does not correspond directly to the address of a source statement, the next statement is used and the address adjusted.

You can copy and paste the readout value into other locations such as in the event portion of the Clause Definition dialog in the LA Trigger window.

Source Files Edit Dialog Box

Overview

The Source window uses a single file name to locate the source files. Use the Source Files Edit dialog box to specify a list of paths to the locations of source files in the file system. You can also specify a list of source file suffix entries in the Source Files properties tab.

Behavior

The appearance of the dialog box depends on which option (Search Path List or Suffix List) is selected in the Source Files properties tab. If the Search Path List is selected, the title of the dialog box displays Modify Search Path List; if the Suffix List is selected, the title of the dialog box displays Modify Suffix List. The selection box is blank if you clicked Add to access the dialog. If you clicked Edit to access the dialog, the selection contains the highlighted entry from the Source Files properties tab.

To Use

When you enter the dialog box to add or edit the search path, you can enter a path in the Selection field. Often the source files will be located on a networked drive. You can click Browse to specify a search path to the networked drive or to any other directory in the file system.

When specifying the search path, you can use an asterisk (*) as the final component of a search path directory.

For example, to specify the following path:

C:\MySources*

The MySources directory and all directories (one-level deep) within MySources will be searched. However, you cannot use the asterisk as a general-purpose wild card character.

For example, the use of an asterisk in the following is invalid:

C:\MySources**

or

C:\MySources\abc*

NOTE. *When you specify a search path or file suffix, make sure that the source file you are looking for has a unique file name or file suffix. When searching for source files it is possible to load the wrong file if there is another file with the same name and suffix in the directories specified by the search path. The Source window loads the first file that matches the search criteria.*


To open the Source Files Edit dialog box from a Source window, click the Properties toolbar button, click Source Files, and then click either Add or Edit.

Related Topic

[Source Files Properties \(see page 237\)](#)

Source Go To Dialog Box

Use the Source Go To dialog box to move in the source code. Double-click a cursor or mark from the displayed list to go to the mark in the Source window.

To open this dialog box, open a Source window and click .

Scan Listing Direction Dialog Box

Use the Scan Listing Direction dialog box to specify the direction to search for samples in the associated Listing window. This dialog box displays when the Scan Listing field in the Source window is set to Ask and you move the cursor in the Source window by dragging the cursor, adjusting the cursor line number, or doing a search.

Click << Earlier to search for samples from the present active Listing cursor position towards the beginning of data. Click Later >> to search forward from the active Listing cursor.

Related Topic

[Scan Listing Mode \(see page 230\)](#)

Source Window Search Dialog Box

Overview


Use the Source Window Search dialog box to search for text patterns in the current file or in all source files identified by the loaded symbol file.

Behavior

When searching for data in multiple files, the search operation follows the order of files as listed in the Source Window properties tab in the Displayed Source File list. When you search for data in all files and one or more of the files cannot be found, the Source File Locator dialog box displays asking you to specify the file location.

If the search is not completed in a set amount of time, the Search Progress dialog box displays the status of the search operation. You can allow the search to continue or you can click the Abort button to halt the search.

To Use

To open the Source Window Search dialog box, open a Source window and click .

Enter an ASCII value in the pattern field and click the search direction. The search string can include leading and embedded white space. Trailing white space is ignored.

Related Topic

[Source Window Properties \(see page 236\)](#)

Source File Locator Dialog Box

The Source File Locator dialog box appears when a source file cannot be located. This may happen under the following conditions:

- When you first attempt to create a new Source window
- When you move the active cursor in either the Source window or the Listing window
- When you do a text search
- When you select a different source file while in the Source Window properties tab

This dialog box appears when the Source window searches for source files and is unable to find the files as specified in the Search Path List and Suffix Lists specified in the Source Files properties tab.

Specify the path of the source file or click Browse to select a path.

In some cases, there may not be an available source file to load. If this happens, you can provide an empty placeholder file to prevent the Source File Locator dialog box from appearing.

TIP. When specifying the search path use an asterisk (*) as the final component of a search path directory. For example, if you specify the following path:

C:\MySources*

The MySources directory and all directories (one-level deep) within MySources will be searched. However, you cannot use the asterisk as a general-purpose wild card character, and the wild card character may only be used as the last character in the search string. For example, the use of an asterisk is invalid in the following:

C:\MySources**

or

C:\MySources\abc*

NOTE. *The wild card character may only be used in the source path, and only as the last character in the search string.*

Related Topics

[Source Files Properties \(see page 237\)](#)

[Source Files Edit Dialog Box \(see page 233\)](#)

Source Window Properties

Overview

Use the Source Window properties tab to change the appearance and content of the Source window.


Behavior

- Display Line Numbers turns line numbers on or off. Turn the line numbers off if you need more room to display the source file.
- Font Size selects the size of the displayed text and the line numbers.
- Tab Spacing specifies the spacing between tab columns.

NOTE. *You may need to adjust the tab spacing if the highlighting is incorrect. Use the same tab spacing values as in the editor that you used to create the source files. If you are unsure of the actual setting, you may need to experiment with tab settings until the highlighting is correct.*

- Foreground Color specifies the color of displayed text and line numbers.
- Background Color specifies the color for the data area background.
- Uncorrelated Color specifies the color of the text in the Source window when the active cursor position in the Source window does not correlate to the active cursor position in the Listing window.
- Color Scheme sets the colors shown in the Source window. The color scheme can be user-defined colors, as specified in the Color tab. Access the Color tab by selecting Options from the System menu.
- Displayed Source File lists the source file currently displayed in the Source window. To view the entire path name of the source file, click the About Data tab. The path name also appears in the source code column header in the Source window.

To Use

To open the Source Window properties tab, open a Source window and click  in the toolbar.

TIP. You can also double-click the header in the data area to open the Source Window properties tab.

Source Files Properties

Overview

Use the Source Files properties tab to define the location of source files in the file system.

The Source Files properties tab lists the directories to search for the source files in the file system. It also lists the file suffixes to try attaching to the file names provided by the compiler (used for compilers that do not include the source file suffix).

Behavior

The Source window uses the file path and file suffix in combination to locate the correct source file. It searches for the file name in each directory indicated by the file path list in the order specified. If a file name does not have a suffix, then suffixes from the suffix list are tried until a match is found or all file path and suffix combinations are exhausted.

- Select Search Path List to specify source file locations; select Suffix List to specify possible file suffixes.
- Click Add to add items.
- Click Edit to edit the selected items.
- Click Delete to delete a selected entry from the list.
- Check the box at the bottom of the Source Files properties tab to save the file paths and suffixes as defaults.

When specifying the search path, you can use an asterisk (*) as the final component of a search path directory.

For example, if you specify the following path:

```
C:\MySources\*
```

The MySources directory and all directories (one-level deep) within MySources will be searched.

However, you cannot use the asterisk as a general-purpose wild card character.

For example, the use of an asterisk is invalid in the following:

```
C:\MySources\*\*
```

or

```
C:\MySources\abc*
```

Specify the complete absolute (not relative) path name for the file in the Search Path List. Otherwise the Source window will not locate the source file. Make sure that the source file you are looking for has a unique file name. When searching for source files it is possible to load the wrong file if there is another file

with the same name in the directories specified by the search path. The Source window always loads the first file that matches the search criteria.

The format of the file path information can take the form of a file path, a file path list file, or both. Each entry is prefaced with either **Path:** or **Path File:** to indicate which type of information it contains. For example:

Path: C:\Program Files\TLA700\Samples\Source Files

Path File: C:\My Documents\filePathList.txt


Note that the file path list can contain numerous file paths, which are searched according to the order in which the paths appear in the file. In addition, both file paths and file path lists can be listed in the Source Files tab of the Properties dialog box. In this case, the paths will be searched according to the position of the entry in the Source Files tab, and according to the order in which the paths appear in the file path list file.

NOTE. *When using file path list files to define the location of source files in the file system, either DOS or UNIX end of line controls are acceptable.*

Because some compilers do not include a source file suffix, you may also need to specify the correct file suffixes in the file suffix list (examples of correct file suffixes are: .c .cpp , and .s). The Source window always loads the first file that matches the criteria in the file suffix list.

The logic analyzer searches for files using both lists from top to bottom. Therefore, you should place the most likely search path and file suffix at the beginning of the lists.

To Use

To view the Source Files properties, open a Source window, click  in the toolbar, and click the Source Files tab.

NOTE. *The wild card character may only be used in the source path, and only as the last character in the search string.*

Related Topics

[Source Files Edit Dialog Box \(see page 233\)](#)

[System Source Files Properties \(see page 374\)](#)

Source Window Data Properties

Overview

Use the Data properties tab to specify Source window attributes. You can specify the Listing window, data source group (Listing window column), and symbol file.


Behavior

When working with disassembled data, the quality of a disassembler's fetch prediction and queue tracking will determine the quality of the source correlation. You should consider using Mark Opcode to fix any disassembly problems across gaps.

TIP. Load your microprocessor support package and set the Listing window disassembly mode to Software before you create the Source window. The software disassembly mode filters out non-instruction cycles to make it easier to view and trace data in the Listing window.

To Use

- Select a Listing window from the list of all currently available Listing windows containing data.
- Select a data source from the selections based on the Listing window. The data source can be data from an installed module or from a saved data file.
- Select a column (data group) from the selected data source. In most cases, you will want to select a microprocessor address column. Timestamp columns, Sample columns, and groups without assigned data signals are not available as data groups for this purpose.
- Select a symbol file that contains the source code symbols from a list of all loaded symbol files. Valid symbol files include those that correspond to the data group column (when used with a symbolic radix). The symbol file can be the executable file that has been compiled and linked for debug or a TSF format file.
- If the symbol file has not yet been loaded, click Symbol File to browse and load a symbol file.


To view the Source window Data properties, open a Source window, click  in the toolbar, and click the Data tab.

Related Topic

[Tektronix Symbol File Format \(TSF\) \(see page 350\)](#)

Source Window Marks Properties

Use the Marks properties tab to format marks in the Source window. You can also enter comments for any marks that you have added. Changing a mark in the Source window does not affect the marks in the Listing window.

To view the Source window Marks properties, open a Source window, click  in the toolbar, and click the Marks tab.

TIP. You can also open this property page by double-clicking a mark in a Source window.

Source-Relative Cursor Positioning

Overview

Source-relative cursor positioning refers to updating the Listing window cursor position as a result of changing the position of the active cursor in the Source window.

Behavior

When you move the Source window active cursor to a new statement, the Source window determines the address range of the source statement based on information from the loaded symbol file. The Source window uses the Scan Listing mode setting to determine the search direction in the Listing window.

The Listing window searches the acquisition data for a matching address. When the matching address is found, the active cursor in the Listing window is moved to the matching sample. If the Source window cursor is moved to a nonexecutable statement, such as a comment, the address of the next executable statement is used.

To Use

You can move the Source window cursor directly in the Source window using the following methods:

- Clicking or dragging the cursor handles
- Using the cursor list box controls
- Entering an address in the Active Cursor Readout
- Right-clicking the mouse and selecting Move Cursor 1 (or 2) Here
- Doing a search

Listing-Relative Cursor Positioning

Overview

Listing-relative cursor positioning refers to updating the Source window cursor position as a result of changing the position of the active cursor in the Listing window.

Behavior



When you move the Listing window cursor to a new sample, the active Source window cursor attempts to move to a corresponding source statement in the Source window. The Source window uses information from the loaded symbol file to convert the address of the Listing cursor position to a source file name and statement (line number) location. The Source window then updates the active Source window cursor position and displays the corresponding source statement.

To Use

You can move the Listing window cursor using the following methods:

- Clicking the Next Mark or Previous Mark buttons in the Source window
- Clicking the Step Forward or Step Backward buttons in the Source window
- Directly by using the controls in the Listing window
- Indirectly by locking cursors to another Listing or Waveform window

Tracing the Execution of Source Code

1. In the System window, click the Source data window icon.
2. Click  (Step Forward) or  (Step Backward) to move the cursor to the next or previous executed statement.
3. The cursors in the Source window, and in the associated Listing window, will advance to the next or previous executed source code statement.

Getting Out of a Loop in a Source Window

This procedure assumes that you have a Source window and a Listing window open.

1. In the Source window, locate the code for the loop.

2. Position the cursor on an executable statement outside of the loop.
3. Right-click the mouse and click either Move Cursor 1 Here or Move Cursor 2 Here (depending on which cursor is active).

Setting a Trigger from Source Window Data

This procedure assumes that you have a Source window and a Listing window open.

1. In the Source window, position a cursor on the statement of interest so that the address displays in the Active Cursor Readout.
2. Select the address in the Active Cursor Readout by double-clicking the left mouse button.
3. Copy the address to the clipboard by selecting Copy Text from the Edit menu, or by typing the CTRL+C key combination.
4. Open the Trigger window for the LA module.
5. Click the If-Then button to open the Clause Definition dialog box.
6. Click Group and paste the address into the group value field by selecting Paste Text from the File menu or by typing the CTRL+V key combination. The radix for the group value must be set to Hex.
7. Click OK.

When you start the acquisition, the logic analyzer will be set to trigger on the group address.

Packet Structure Description Language

Overview

Packet Structure Description Language or PSDL is used for describing and creating protocols. It is based on the Abstract Syntax Notation One (ASN.1). A complete description of the syntax and notation used can be found in the International Telecommunication Union X.680 document. For a high level overview on PSDL syntax and notation, select one of the related topics.

Use PSDL to define packet structures in the Protocol Editor pane of the Protocol Designer Window. The PSDL files have the .spz protocol file suffixes. For more information on PSDL, click on one of the links below.

Syntax

Overview

You can use PSDL to edit standard protocols or create your own. The symbols and conventions used in the PSDL are described in the related topics. PSDL uses Abstract Syntax Notation One (ASN.1).

An ASN.1 item consists of a sequence of characters listed in the ASN.1 Character table.

ASN.1 Characters		
A to Z	a to z	0 to 9
:	=	{
}	<	.
@	()
[]	-
'	"	
&	^	*
;	!	

General rules for ASN.1 items are as follows:

- PSDL is case sensitive.
- Most items appear on a single line and do not contain white space.
- Line length is unrestricted.
- Items in production sequences may appear on more than one line.
- Items in production sequences may be separated by white space, empty lines, or comments.
- Items must be separated from each other by white space, newline, or comment if the following item begins with a character that is allowed in the preceding item.

For information on the following syntactical items, click one of the related topics.

Related Topics

[ASN.1 Types \(see page 252\)](#)

[Length and Value Expressions \(see page 261\)](#)

[Packet Structure \(see page 245\)](#)

[PSDL Additions to ASN.1 \(see page 258\)](#)

[PSDL Types \(see page 258\)](#)

[Tags \(see page 262\)](#)

[Miscellaneous Commands \(see page 262\)](#)

[Type Definitions \(see page 250\)](#)

[Value Definitions \(see page 261\)](#)

Packet Structure Definition

PacketDef: DescrList PacketTypeDef [TypeValueDefList \(see page 250\)](#)

DescrList: [empty](#)

| Descr DescrList

Descr: **DESCRIPTION** "::<=" [cstring](#)

BIT_ORDER "::<=" BitDecodeOrder

| **DECODING_RULES** "::<=" EncRules

EncRules: **BER**

| **BASIC_PER_ALIGNED**

| **BASIC_PER_UNALIGNED**

PacketTypeDef: **PACKET** "::<=" [typereference](#)

Operators

The assignment operator (::<=) defines a type reference (assigning a name to a type definition):

TypeRefName ::= Type

Type can be another type reference or a basic type (such as BYTE_HEX, BIT_BIN, or PSDL_SEQUENCE).

The semantic of a type reference is that it can be replaced by its definition. For example, given

TypeRef1 ::= BIT_DEC { LENGTH(4) }

TypeRef2 ::= BYTE_DEC { LENGTH(2) }

TypeRef3 ::= PSDL_SEQUENCE {

field1 TypeRef1,

```

field2    TypeRef1,
field3    TypeRef2
}
Packet ::= TypeRef3

```

The definition of the Packet is equivalent to:

```

Packet ::= PSDL_SEQUENCE {
field1    BIT_DEC { LENGTH(4) },
field2    BIT_DEC { LENGTH(4) },
field3    BYTE_DEC { LENGTH(2) }
}

```

NOTE. *In the above statements, field1, field2, field3 are the field names of three components that constitute the Packet Structure.*

=====

The field name is required if the value of a field is to be used in the LENGTH expressions of other fields, for example:

```

PSDL_SEQUENCE {
length    BYTE_DEC{ LENGTH(2) }           "Length Field",
data      BYTE_HEX { LENGTH(length - 2) } "Data Field"
}

```

It is a good idea to always specify an internal name. It can be omitted, because the standard ASN.1 syntax allows it to be omitted.

=====

The expression syntax of LENGTH expressions is:

```

Expr:      "(" Expr ")"
| Expr "+" Expr
| Expr "-" Expr
| Expr "*" Expr
| Expr "/" Expr
| Expr "%" Expr
| number
| identifier

```

```
| PACKET_LENGTH
| REST_LENGTH
| ALIGN "(" Expr ")"
```

The "identifier" refers to a field name, `PACKET_LENGTH` is the total length of the frame, `REST_LENGTH` is the length of the not yet decoded frame, and `BYTE_HEX { LENGTH(ALIGN(4)) }` tells the decoder to make the field long enough so that it ends on a four byte boundary (for bit strings, the alignment is counted in bits).

The `VALUE` expression tells the encoder how to compute the value of a field. For example,

```
PSDL_SEQUENCE {
dat_len  BYTE_DEC { LENGTH(2) } VALUE(LENGTH(data)),
data     BYTE_HEX { LENGTH(dat_len) } USER_DATA
}
```

This specifies that the value of 'dat_len' is the length of the data field. Because you are in a decoding only environment, you will never have to deal with `VALUE` expressions.

Control Statements

The syntax for a PSDL tag is as follows:

```
PsdlTag: '[' Offset ',' TagVal ']' or
 '[' ABSOLUTE Offset ',' TagVal ']'
Offset: decimal_value
TagVal: binary_value or hex_value
```

The units for the offset are bits for `binary_value` and bytes for `hex_value`.

The offset is relative to the point in the packet data stream that has to be decoded next, unless `ABSOLUTE` is specified, which means relative to the start of the frame. Offsets can be negative, therefore it is possible to match bits before the point in the packet data stream that will be decoded next. The following example shows `PSDL_CHOICE` statements:

```
IpContents ::= PSDL_CHOICE {
[50 ABSOLUTE, '0000000000000000'B] user_data BYTE_HEX { LENGTH(total_length -
(ihl * 4)) } USER_DATA "Data",
[50 ABSOLUTE, 'x'B] frag_data BYTE_HEX { LENGTH(total_length - (ihl * 4)) }
"Fragment Data"
}
```

In the above structure, if bits 50 through 63 (counting from 0) relative to the start of the frame are all zeros, then `user_data` is selected, otherwise `frag_data` is selected.

The `PSDL_CHOICE` statement can be used like an If-Else statement. You cannot refer to fields directly in `PSDL_CHOICES`. In other words, you cannot use field names here.

You can avoid a given number of bytes or bits in a packet being decoded by skipping the bytes or bits in the packet data stream.

To skip bits or bytes, use the key word `NODISPLAY`. `NODISPLAY` means that the field is not shown in the monitor window. It can be used as follows:

```
BIT_BIN { NODISPLAY LENGTH(3) }
```

This skips three bits from the point in the packet data stream that was already decoded, and continues decoding.

```
BYTE_HEX { NODISPLAY LENGTH(1) }
```

This skips one byte from the point in the packet data stream that was already decoded and continues decoding.

Loop Statements

You can decode fields with loop statements using the `UNTIL ('xxxx'H)` statement. You can have up to 4 bytes within the `UNTIL` statement.

```
Phrase ::= BYTE_ASCII { UNTIL('0D0A'H) }
```

This statement tells the compiler to start decoding the packet data from hexadecimal bytes to ASCII characters until it sees the data `'0x0D'` immediately followed by `'0x0A'`.

Use the `UNTIL` statement in `PSDL` octet/bit strings instead of the `LENGTH` expression. The `UNTIL` statement specifies that the field extends up to the point where the `UNTIL` pattern matches. The following example shows a field that extends up to (and includes) the next CR/LF.

```
BYTE_ASCII { UNTIL('0D0A'H) }
```

The `UNTIL` statement can also be used for `PSDL_SET`s to specify where the set ends. The following example shows a set of `ElemType` fields that end at the first null byte (not part of the set).

```
PSDL_SET UNTIL('00'H) { [0, 'xx'H] set_elem ElemType }
```

This can be mixed with a `LENGTH` expression. For example:

```
PSDL_SET OF LENGTH(10) UNTIL('00'H) { ... }
```

This means the set ends either after 10 octets or if a null byte is encountered, whichever happens first.

The `UNTIL` statement can also be used with a `PSDL_SET` statement as

```
HeadSet ::= PSDL_SET UNTIL('0D0A'H)
{
```

```
[0, 'xx'H] param_line ParamLine
}

ParamLine::=BYTE_ASCII { UNTIL('0D0A'H) }
```

The above statement sets a loop for the 'ParamLine' field type, which continues until the packet data equals '0x0D' immediately followed by '0x0A'.

Sample PSDL Program

The following sample program and its explanation are intended to give you a high-level overview of a protocol definition file in PSDL.

For the purposes of this example, assume that the Packet A structure below has two header fields, Address and Length, followed by the Information field.

Address Length Information
 1 byte 2 bytes variable length (rest of frame)

The protocol definition file is shown below.

```
=====
DESCRIPTION::= "Packet A"
DECODING_RULES::=BER
PACKET::= PacketA
PacketA::= PSDL_SEQUENCE {

address Address "Pkt_Address",
length Length "Pkt_Length ",
info Information " Pkt_Information "
}

Address::= BYTE_HEX{LENGTH(1)}
Length::= BYTE_HEX{LENGTH(2)}
Information::= BYTE_HEX {LENGTH(REST_LENGTH)}
=====
```

Packet Description

The DESCRIPTION is assigned to the packet description string that specifies the protocol name whose definition is written and other details such as: RFC details, Standard details, or Version details. The

description string appears with the protocol filename in various "Protocol Designer" dialog boxes (Open Protocol dialog, Save Protocol dialog). Refer to **Packet Structure Definition** below for information on specifying the description.

Decoding Rules

This is an optional field. The `DECODING_RULES` is assigned to the type of decoding rules. Refer to **Packet Structure Definition** below for information on specifying the description.

Packet Structure Definition

Consider the following line of code:

```
PACKET ::= PacketA
```

This statement specifies which type in the type definition list defines the packet structure. If this statement is omitted, the first type definition is considered to be the packet type (in this example the `PACKET` statement can be omitted).

Consider the next few lines of code:

```
PacketA ::= PSDL_SEQUENCE {
address Address "Pkt_Address",
Length Length "Pkt_Length",
info Information "Pkt_Information"
}
```

In the preceding code, the `PacketA` structure is defined as having the following three fields: "address", "length", "info". The address field is of type "Address" with field label "Pkt_Address". Similarly, the length field is of type Length with field label "Pkt_Length" and the info field is of type "Information" with the field label "Pkt_Information".

For more information, refer to [Protocol Stack, SPZ Files, Packet Structure Overview \(see page 266\)](#).

Related Topic

[Defining a Protocol \(see page 268\)](#)

Type Definitions

```
TypeValueDefList:
TypeValueDef
| TypeDefList TypeValueDef
TypeValueDef:   TypeDef
```

```
| valueDef \(see page 261\)
TypeDef: typereference "::=" ASPHType
| DefaultTag
DefaultTag: EXPLICIT TAGS
| IMPLICIT TAGS
| AUTOMATIC TAGS
ASPHType: ASTag \(see page 262\) ASType \(see page 252\) Open UserData \(see page 262\)
| PhType Open UserData \(see page 262\)
CompTypeList: CompType
| CompTypeList "," CompType
CompType: NamedType
| NamedType OPTIONAL
| NamedType DEFAULT value
| COMPONENTS OF subtype
AltTypeList: AltType
| AltTypeList "," AltType
AltType: NamedType
NamedType: PhTag OptIdentifier SubType AttributeList
OptIdentifier: identifier
| empty
SubType: ASPHType
OptSizeConstraint:
Constraint
| SizeConstraint
| empty
```

Type References

Name

The name used for type references is **typereference**.

Rules

- A type reference consists of one or more letters, numbers, and hyphens.
- The first character must be upper case and a hyphen cannot be used as the last character.

Identifiers

Name

The name used for identifiers is **identifier**.

Rules

- An identifier consists of any number of letters, digits, or hyphens.
- The initial character is upper case.
- A hyphen cannot be the last character nor be followed by another hyphen.

Empty Item

Name

The name used for empty items is **empty**.

Rules

- Contains no characters

ASN.1 Types

AsType: AsSequence

| AsSequenceOf

| AsChoice

| AsSet

| AsSetOf


```
| AsAnyDefbyCase
| AsEllipsis
| AsTerminal
OptConstraintList
| typereference
OptConstraintList
AsSequence:  SEQUENCE "{" CompTypeList}"
AsSequenceOf: SEQUENCE OptSizeConstraint OF
OptIdentifier SubType AttributeList
AsSet:  SET "{" CompTypeList}"
AsSetOf: SET OptSizeConstraint OF
OptIdentifier SubType AttributeList
AsChoice: CHOICE MsgChoiceInd "{" AltTypeList}"
OptConstraintList:
empty
| OptConstraintList Constraint
AsTerminal:  BOOLEAN
| OCTET STRING
| REAL
| AsInt
| AsEnum
| ASBitStr
| NullType
| ANY DefinedBy
| OBJECT IDENTIFIER
| NumericString
| PrintableString
| TeletexString
| VideotexString
| VisibleString
| IA5String
```

```

| GraphicString
| GeneralString
| BMPString
| UniversalString
| UTF8String
AsInt:    INTEGER "{" NumberList "}"
| INTEGER
AsEnum:   ENUMERATED "{" EnumNumberList "}"
AsBitStr: BIT STRING "{" NamedBitList "}"
| BIT STRING
EnumNumberList: NamedEnumNumber
| EnumNumberList "," NamedEnumNumber
NamedEnumNumber: identifier EnumNumber
| "..."
EnumNumber:    "(" IntVal ")"
| empty
NumberList:   NamedNumber
| NumberList "," NamedNumber
NamedNumber:  identifier NamedNumberVal
NamedNumberVal: "(" SignedNumber ")"
| empty
NamedBitList: NamedBit
| NamedBitList "," NamedBit
NamedBit:     identifier "("number ")"
DefinedBy:    DEFINED BY identifier
| empty
AsEllipsis:  "..."
NullType:    NULL
Type:        BuiltinType
| ConstrainedType
BuiltinType: NullType

```

```

ConstrainedType: Type Constraint
Constraint: "(" ConstraintSpec ExceptionSpec ")"
ConstraintSpec: SubtypeConstraint
ExceptionSpec: "!" ExceptionIdentification
| empty
ExceptionIdentification:
SignedNumber
SubtypeConstraint:
ElementSetSpecs
ElementSetSpecs: RootElementSetSpec
| RootElementSetSpec "," "... "
| "... " "," AdditionalElementSetSpec
| RootElementSetSpec "," "... " ","
AdditionalElementSetSpec
RootElementSetSpec:
ElementSetSpec
AdditionalElementSetSpec:
ElementSetSpec
ElementSetSpec:
Unions
Unions: Intersections
| UElms UnionMark Intersections
UElms: Unions
Intersections: IntersectionElements
| IElems IntersectionMark IntersectionElements
IElems: Intersections
IntersectionElements:
Elements
Elms: Elements
UnionMark: "|"
| UNION

```

```

IntersectionMark:  "^"
| INTERSECTION
Elements:  SubtypeElements
| "(" ElementSetSpec ")"
SubtypeElements:  SingleValue
| ValueRange
| PermittedAlphabet
| SizeConstraint
Includes: | empty
ValueRange:  LowerEndpoint ".." UpperEndpoint
LowerEndpoint:  LowerEndValue
| LowerEndValue "<"
UpperEndpoint:  UpperEndValue
| "<" UpperEndValue
LowerEndValue:  Value
| MIN
UpperEndValue:  Value
| MAX
SizeConstraint:  SIZE Constraint
PermittedAlphabet:
FROM Constraint
SingleTypeConstraint:
Constraint
ValueConstraint:  Constraint
| empty
PresenceConstraint:
| empty
SingleValue:  Value
Value:  BuiltinValue
| ReferencedValue
BuiltinValue:  IntegerValue

```

```
| BooleanValue
| CharacterStringValue
IntegerValue: SignedNumber
| identifier
BooleanValue: TRUE
| FALSE
CharacterStringValue:
RestrictedCharacterStringValue
RestrictedCharacterStringValue:
cstring
SignedNumber: number
| "-" number
ReferencedValue:
DefinedValue
```

Character String Item

Name

The name used for character string items is **cstring**.

Rules

- A cstring consists of any number of graphic symbols and spacing characters.
- A cstring may span more than one line of text.
- White spaces that appear immediately before or after the end of the cstring have no significance.
- Control characters cannot be represented in a cstring.

Number Item**Name**

The name used for number items is **number**.

Rules

- A number consists of one or more digits.
- The first digit cannot be zero unless the number item is a single digit.

PSDL Additions to ASN.1

```

AsAnyDefbyCase: ANY DEFINED BY identifier CASE "{" OptCaseList "}"
OptCaseList: CaseList
| empty
CaseList: CaseElement
| CaseList CaseElement
CaseElement: IdentOrNum EN TRY SubType
| AsEllipsis
IdentOrNum: identifier
| number

```

PSDL Types

```

PhType: PhSequence
| PhChoice
| PhSetLenCnt
| PhSetUntil
| PhTerminal OptConstraintList
PhSequence: PSDL_SEQUENCE "{" CompTypeList "}"
PhChoice: PSDL_CHOICE MsgChoiceInd "{" AltTypeList "}"
PhSetLenCnt: PSDL_SET OF LengthCount "(" Expr ")" OptUntil
MsgChoiceInd "{" CompTypeList "}"

```

```
OptUntil: empty
| UNTIL "(" EndTag ")"
LengthCount: LENGTH
| COUNT
PhSetUntil: PSDL_SET UNTIL "(" EndTag ")" "{" CompTypeList}"
PhTerminal: MAC_ADR
| IP_ADR
| IPV6_ADR
| SPC
| SLS
| CIC
| BCD_STRING "{" Decod LengthSpec}"
| BYTE_HEX "{" Offset Decod LengthUntilSpec CommSpec}"
| BYTE_BIN "{" Offset Decod LengthUntilSpec CommSpec}"
| BYTE_DEC "{" Reverse Offset Decod LengthUntilSpec CommSpec}"
| BYTE_SDEC "{" Reverse Offset Decod LengthUntilSpec CommSpec}"
| BIT_HEX "{" Decod LengthSpec CommSpec}"
| BIT_BIN "{" Decod LengthSpec CommSpec}"
| BIT_DEC "{" Reverse Decod LengthSpec CommSpec}"
| SESSION_LI "{" Decod}"
| ASN1_TAG "{" Decod}"
| ASN1_LI "{" Decod CommSpec}"
| BYTE_ASCII "{" Offset Decod LengthUntilSpec CommSpec}"
| BYTE_EBCDIC "{" Offset Decod LengthUntilSpec CommSpec}"
| BREAK_LAYER
Offset: empty
| RELATIVE TO END
Decod: empty
| NONDISPLAY
| NONDEC
| BitDecodeOrder
```

```

Reverse: empty
| REVERSE
LengthUntilSpec: LengthSpec
| UNTIL "(" EndTag ")"
LengthSpec: LENGTH "(" Expr ")"
Val: hstring
| bstring
| number
| SignedNumber
| cstring
CommSpec: empty
| COMMENT "(" cstring ")"
| COMMENT "(" CommList "," cstring ")"
| COMMENT "(" CommList ")"
CommList: Comment
| CommList "," Comment
Comment: "[" Val "]" cstring

```

Hexadecimal String Item

Name

The name used for hexadecimal string items is **hstring**.

Rules

- An hstring consists of any number of the following characters: A B C D E F 0 1 2 3 4 5 6 7 8 9, preceded by a single quote and followed by 'H.
- White space and newlines are allowed, but within an hstring have no significance.

Binary String Item**Name**

The name used for binary string items is **bstring**.

Rules

- A bstring consists of any number of zeros or ones, preceded by a single quote and followed by 'B.
- White space and newlines are allowed within a bstring, but have no significance.

Value Definitions

```

ValueDef:  identifier ValueNotation
ValueNotation:  INTEGER "::<=" number
| OCTET STRING "::<=" hstring
| visibleString "::<=" cstring
| OBJECT IDENTIFIER "::<=" "{" ObjIdCompList "}"
ObjIdCompList:  ObjIdComp
| ObjIdCompList ObjIdComp
ObjIdComp:  identifier Rest
| number
Rest:  empty
| "(" NumOrId ")"
NumOrId:  number
| identifier

```

Length and Value Expressions

```

Expr:  "(" Expr ")"
| Expr "+" Expr
| Expr "-"
| Expr "*" Expr
| Expr "/" Expr

```

```

| Expr "%" Expr
| number
| identifier must not contain "-"
| PACKET_LENGTH
| REST_LENGTH
| ALIGN "(" Expr ")"

```

Tags

```

PhTag: "[" TagOffset TagVal "]"
| "[" TagOffset ABSOLUTE TagVal "]"
| empty
TagOffset: IntVal
TagVal: hstring
| bstring
ASTag: empty
| "[" Class number "]" ImplExp1
Class: empty
| UNIVERSAL
| APPLICATION
| PRIVATE
ImplExp1: empty
| IMPLICIT
| EXPLICIT
EndTag: hstring
| bstring

```

Miscellaneous Commands

```

AttributeList: empty
| Attribute AttributeList

```

```

Attribute: ShortName
| ShortName "/" LongName
| PROTECTED
| MULTIPLE
ShortName: cstring
LongName: cstring
IntVal: Sign UnsignedIntVal
UnsignedIntVal: number
BitDecodeOrder: MSBF
| LSBF
MsgChoiceInd: empty
| MSG
Sign: "+"
| "-"
| empty
Open: empty
| OPEN
UserData: empty
| USER_DATA
| USER_DATA "(" identifier ")"

```

Reserved Words (Ab to Con)

This topic is a list of reserved words for PSDL. This list represents reserved character sequences. White space does not occur in these sequences.

ABSENT	ABSOLUTE	ALIGN
ANY	APPLICATION	ASN1_LI
ASN1_TAG	AUTOMATIC	BASIC_PER_ALIGNED
BASIC_PER_UNALIGNED	BCD_STRING	BER
BIT	BIT_BIN	BIT_DEC
BIT_HEX	BIT_LENGTH	BIT_ORDER
BMPString	BOOLEAN	BREAK_LAYER
BY	BYTE_ASCII	BYTE_BIN

BYTE_DEC	BYTE_EBCDIC	BYTE_HEX
BYTE_LENGTH	BYTE_SDEC	CANONICAL_PER_ALIGNED
CANONICAL_PER_UNALIGNED	CASE	CER
CHOICE	CIC	COMMENT
COMPONENT	COMPONENTS	CONDITIONAL

Reserved Words (Cou to Max)

This topic is a list of reserved words for PSDL. This list represents reserved character sequences. White space does not occur in these sequences.

COUNT	DECODING_RULES	DEFAULT
DEFINED	DER	DESCRIPTION
DISP_GROUP	ENCODING_RULES	END
ENTRY	ENUMERATED	EXCEPT
EXPLICIT	FALSE	FRAME_BEGIN
FRAME_END	FROM	GENERIC_TYPE
GROUP	GeneralString	GraphicString
IA5String	IDENTIFIER	IGNORE
IMPLICIT	INCLUDES	INTEGER
INTERSECTION	IP_ADR	IPV6_ADR
ISO646String	LENGTH	LSBF
MAC_ADR	MANDATORY	MAX

Reserved Words (Min to Ta)

This topic is a list of reserved words for PSDL. This list represents reserved character sequences. White space does not occur in these sequences.

MIN	MSBF	MSG
MULTIPLE	NODISPLAY	DONDEC
NUL	NumericString	OBJECT
OCTET	OF	OPEN
OPTIONAL	OVERLAY	PACKET
PACKET_LENGTH	POS	PRESENT
PRIVATE	PROTECTED	PSDL_CHOICE
PSDL_SEQUENCE	PSDL_SET	PrintableString
RANGE	REAL	RELATIVE

REST_LENGTH	REVERSE	SEQUENCE
SESSION_LI	SET	SIZE
SLS	SPC	STATION
STRING	T61String	TAGS

Reserved Words (To to Wi)

This topic is a list of reserved words for PSDL. This list represents reserved character sequences. White space does not occur in these sequences.

TO	TRUE	TeletexString
UNION	UNIVERSAL	UniversatString
UNTIL	USER_DATA	USER_PART_ID
UTF8String	VALUE	VideotexString
VisibleString	WIDTH	

Lexical Items

Overview

In addition to reserved words there are other lexical items used in PSDL. The conventions and syntax regarding their usage vary based on the item. Click the name for details on each item.

Lexical item	Name
Type references	typereference
Identifier	identifier
Value references	valuereference
Module references	modulereference
Empty item	empty
Number item	number
Character string item	cstring
Binary string item	bstring
Hexadecimal string item	hstring

Module References

Name

The name used for module reference items is **modulereference**.

Rules

- A module reference consists of one or more letters, numbers, and hyphens.
- The first character must be upper case and a hyphen cannot be used as the last character.
- A module reference is distinguished from a type reference by the context in which it is used

Value References

Name

The name used for value reference items is **valuereference**.

Rules

- A value reference consists of any number of letters, digits, or hyphens and is distinguished from an identifier by the context in which it is used.

Protocol Stack, SPZ Files, Packet Structure Overview

The PSDL programming is similar to ASN.1 programming that defines a protocol definition. The protocol file has the .spz file extension.

Prerequisites

To write protocol files, you need to be familiar with the following concepts:

- The protocol stack definition to which the packet structure maps; the stack definition should follow the packet structure.
- The packet structure that encapsulates the protocols of higher layers
- Knowledge of all fields of the protocols mentioned in the stack and the relation between protocols

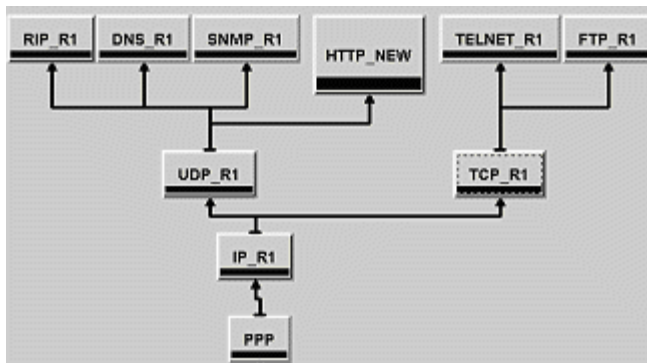
Protocol Stacks

A protocol stack is made up of protocols at different layers. The most common four layers in a network stack are:

- Link Layer

- Network Layer
- Transport Layer
- Application Layer

Stacks are defined using the protocol layers. Each layer (box) is a protocol. There can be many protocols in a given layer. For example, transport layer protocols are TCP and UDP, and link layer protocols are PPP and Ethernet.



Differences Between Link Layer Packets and Packets at Higher Layers

A link layer packet structure maps an entire protocol stack. The link layer packet **header** contains the network layer packet headers. The network layer packets contain the transport layer packet header and the application header and data.

Other packets (above Link layer) map a part of the stack. For example, a PPP packet is a link layer protocol that maps the entire stack, and a TCP packet is a transport layer protocol that maps a part of the stack file (TCP->Telnet, TCP->FTP).

What is Defining a Protocol?

Defining a protocol means writing the protocol packet header definition using PSDL. Each protocol file in the stack has the .spz extension. For example, a PPP protocol file definition consists of the following PPP header fields: Flag, Address, Control, and Protocol. Similarly, an Ethernet protocol file definition consists of the following header fields: SFD, Destination, Source, and Type/Length. A TCP protocol file definition consists of TCP header fields.

Defining a Protocol Structure

Use a top-down approach to define the packet structure. Note all of the possible packet formats that map the stack. For example, consider the stack in the last illustration.

The possible packet formats of PPP frames are shown below (the square brackets refer to a packet header that contains another packet):

- PPP [IP [UDP [HTTP]]]. In this example, PPP encapsulates an IP header, which in turn encapsulates a UDP header. The UDP header encapsulates HTTP.
- PPP [IP [UDP [SNMP]]]
- PPP[IP [TCP [FTP]]]
- PPP[IP [TCP [TELNET]]]
- PPP[IP [UDP [DNS]]]
- PPP[IP [UDP [RIP]]]

Start defining the protocol by writing the .spz file in the Protocol Editor.

Related Topic

[Defining a Protocol \(see page 268\)](#)

Defining a Protocol

Use the following guidelines to define a protocol:

Describe the protocol

Normally, this will be a single string assigned to the DESCRIPTION keyword using the following syntax:

```
DESCRIPTION ::= <string>
```

For example:

```
DESCRIPTION ::= "IP - Internet Protocol, RFC791"
```

Specify the Decoding Rules

In the next few lines, specify the decoding rule using the DECODING_RULES keyword.

```
DECODING_RULES ::= <rule name>
```

The rule name can be one of the following: BER, DER, CER, BASIC_PER_ALIGNED, BASIC_PER_UNALIGNED, CANONICAL_PER_ALIGNED, or CANONICAL_PER_UNALIGNED.

For example:

```
DECODING_RULES ::= BER
```

Consider the following stack definition:

```
Protocol4
```

```
Protocol3
```


Protocol2

MyProtocol

The bottom-most layer is MyProtocol. A MyProtocol packet encapsulates higher layer protocols in the data field (the packet header consisting of Protocol2, Protocol3, and Protocol4).

You can start writing a protocol definition for MyProtocol using PSDL.

```
DESCRIPTION ::= "IP - Internet Protocol, RFC791"
```

```
DECODING_RULES ::= BER
```

Hand the Length of the Protocol Packet to the Protocol Layer

To hand the length of the MyProtocol packet to the MyProtocol layer, use the following syntax:

```
PACKET ::= PSDL_SEQUENCE OF LENGTH ("PACKET_LENGTH") {"PacketName"}
```

Specify the PacketName. It can be the protocol name with a Packet suffix (for example, HTTP_Packet, or IP_Packet). The following example uses MyProtocol_Packet as the PacketName:

```
PACKET ::= PSDL_SEQUENCE OF LENGTH ( PACKET_LENGTH ) { MyProtocol_Packet }
```

Further Define the Packet Using PSDL Structures

Continue defining the packet using the following syntax:

```
PacketName ::= PSDL_SEQUENCE
```

```
"{"
```

```
<field name> <type name> <field label>
```

```
"}"
```

The <field name> variable is a user-defined string starting in lower case letters. This variable is optional and is used to improve the readability of the specification.

The <type name> variable is a user-defined string for the field type and begins with an upper-case letter.

The <field label> variable is an optional user-defined string (normally the packet field name) that will be displayed in the field column of the decode view.

For example:

```
MyProtocol_Packet ::= PSDL_SEQUENCE
```

```
{
```

```
field1 Field1 "Field1",
```

```
length Length "Length",
```

```
address Address "Address",
```

```
type Type "Type",
```

```

payload Payload "Payload"
}

```

In the example above, MyProtocol_Packet is a structure with five components. The first component is called field1 (starts with a lower-case letter) and denotes the data type field Field1 (starts with an upper-case letter). The second component is called length and denotes the data type field Field1. Here the packet decoder takes the data bytes from the packet at the beginning of the packet and relates them to the defined MyProtocol structure. It takes the first byte at the beginning of the packet, relates it to Field1, the next byte to Length, the next two bytes to Address, and the following nibble to Type. It then assigns the remaining items to Payload.

Specify the Type and Size of Each Field

Specify the type and size of each field as shown below:

```

Field1 ::= BYTE_HEX { LENGTH(1) } Length ::= BYTE_HEX { LENGTH(1) } Address ::=
BYTE_HEX { LENGTH(2) } Type ::= BYTE_BIN { LENGTH(4) } Payload ::= BYTE_HEX { LENGTH
(REST_LENGTH) } USER_DATA

```

In the above example, Field1 is a byte-oriented data type (BYTE_*) with the size of one byte. Similarly, Length and Address are byte-oriented data types with the sizes of one byte and two bytes. Type is a bit-oriented data type (BIT_*) with a size of four bits. The Payload data type is a byte-oriented hex string within the MyProtocol layer. When you tag the Data field with USER_DATA, this indicates that the contents of the field have to be handed to the next protocol layer.

The line BYTE_HEX { LENGTH (REST_LENGTH) } USER_DATA means hand the rest of the frame to the next protocol layer (for example, Protocol2).

Note that PACKET_LENGTH and REST_LENGTH can only be used in length expressions such as:

```

TypeB1a ::= BYTE_HEX {LENGTH(REST_LENGTH) }

```

Related Topics

[Sample PSDL Program \(see page 249\)](#)

[Create New Protocol Files \(see page 293\)](#)

Protocol Window

Overview

The Protocol window supports protocol decoding across multiple levels of a stack. Use the Protocol window to view and evaluate acquisition data. The Protocol window has three panes: Packet View, Decode View, and Hex View. Each pane displays the acquisition data at different resolution levels. To open a Protocol window from the Window menu, click New Data Window and then select Protocol from the list.

Acquisition data is typically analyzed one block of samples at a time. This makes the analysis faster and uses less memory. The range of samples currently analyzed is displayed in a readout in the Packet View title bar. To change the position and size of the block of samples to analyze, go to the [Protocol Framing Options properties tab](#) (see page 277).


NOTE. Load either a protocol support package into your system or a user-defined framer before you can add a Protocol window. The Protocol window requires the framing setup from the protocol support package or user-defined framer to display the protocol.

Packet View

The Packet View displays all the packets of the current acquisition along with the Time stamp information. Erroneous packets are displayed in red.

Marks

Marks in the Packet View pane help you to navigate through the data displayed in that pane. There are two cursor marks, a System Trigger mark, and an unlimited number of user-assigned marks. Marks are used to measure time, and mark specific samples for evaluation.

- Add user marks by clicking the  button.
- Delete user marks by right-clicking the mark handle, and then selecting Delete from the right-click menu.
- Marks you can move are Cursor 1, Cursor 2, and any marks you add. The numbers 1 and 2 on the mark handles denote the cursor marks. Readouts for Cursor 1, Cursor 2, and the time difference (Delta Time) between the two are located at the top of the data area.
- Marks on the left side of the window are data area marks. These marks are in the portion of the acquisition that is currently viewable. Use data area marks to move within the data area.
- The System Trigger mark is fixed, you cannot move it.
- Marks on the right side are overview marks. These marks show where marks are located relative to the entire acquisition. Use overview marks to move throughout the acquisition. For example, move Cursor

1 to a particular sample as shown in the Cursor 1 readout. Click the Go To toolbar button and select Cursor 1. You can also click the right mouse button on any overview mark, and then click Go To Mark.

- You can move to any mark using the Go To toolbar button.

Decode View

The Decode View decodes the protocol parameters of the packet selected in Packet View and displays them line by line. Each protocol parameter size is displayed as well as any comments contained in the description file associated with the protocol. The protocol stack that is currently in use determines which protocol parameters are shown.

Hex View:

You can choose one of the following formats to display the packet selected in Packet View in the Hex View pane:

- Hex
- ASCII
- HEX+ASCII
- BINARY

If you select an individual protocol parameter in Decode View, the corresponding values are highlighted in Hex View.

Related Topic

[Protocol Design \(see page 285\)](#)

Compilation Result Pane

Overview

The Compilation Result pane is where the results of compiling are displayed.

Behavior

Double-clicking an error message in the Compilation Result pane highlights the line with the error in the Protocol Editor pane.

Related Topic

[Protocol Editor Pane \(see page 287\)](#)

Frames and Framers

The following information provides high-level answers to questions regarding frames and framers.

What is Framing?

Frames are logical groupings of information sent as data layers. Frames are basically data and control information that flow between link layer devices and physical layer devices. The cells or packets received from network layers are transformed into frames by link layer devices and passed on to physical layer devices. This entire process is known as framing.

What is a Generic Framer?

A generic framer is an application that processes acquired data from the logic analyzer (data and control information) from any bus that carries packetized data. It works on any type of bus that connects the link layer and physical layer devices.

What is a Bus-Specific Framer?

A bus-specific framer is an application that processes acquired data from the logic analyzer (data and control information) from a particular bus that only carries packets. Tektronix supplies framers for the SPI-3 TX and RX, SPI-4.2 LVDS, and LVTTTL TX and RX buses.

What is the Difference Between Generic Framers and Bus-Specific Framers?

Bus-specific framers require their corresponding support packages. The packets created by the bus-specific framers are exact packets. Generic framers add more information to the packets that need to be decoded using protocol definition files. As a result, you may not get exact packets if you use a generic framer.

What are the Limitations of the Generic Framer?

The generic framer has the following limitations:

- Scrambling is not supported.
- Fragmentation is not supported.
- Encoded data will be framed "as is."
- Transmit and receive busses cannot be linked.
- Only multiples of 8-bit bus widths are supported.
- In some cases, additional information is added to the packets depending on the specified field conditions. You can further decode the packet using protocol definition (Protocol Designer).

When Should You Use the Generic Framer?

When setups do not have associated bus-specific framers, the Define Bus check box in the New Protocol Window wizard is automatically selected. If a specific framer is available, the Define Bus box is cleared. However, you can enable the check box even if a specific framer is available.

Use the generic framer when the information (cells, packets, and control words) flowing on a bus is captured and framed by the logic analyzer. The Protocol window displays the decoded framed packets.

Protocol Filter Setup Dialog Box

Overview

The Protocol Filter Setup dialog box allows you to selectively filter specific packets from the acquired data. You can choose to filter all packets based on the protocols and their fields.

Behavior


The filter options are active when you select the Enable Filter check box. This dialog box contains all the Filter Options: Fields, Operation, Value, and Format. You can select an item from the Fields, identify the Operation, and specify the Value and the Format.

The Protocol drop-down list shows all the protocols contained in the stack. When you select a protocol, its fields are listed in the Field box. When you select an item from the Field box, its value (if any), is displayed in the Predefined Values.

Select the Enable Fields check box to filter the packets based on the specific fields. If you clear the Enable Fields check box, you can filter the data based on the selected protocol.

You can specify the values in any of the following formats: Decimal, Hex, ASCII or Binary.

To Use

Click  (the Filter Setup toolbar button) to access the Filter Setup dialog box.

Use this dialog box to define operations and value for the filter. By default, the filter is not enabled. To enable the filter, select the Enable Filter check box.

Related Topic


[Filtering the Protocol Display \(see page 284\)](#)

Protocol Properties: About Data Tab

Overview

The Protocol Properties: About Data tab provides information about the data source used for the active data window.

To Use

From the Protocol window, click the  button and then click the About Data tab. This is a read-only tab.

Protocol Window Properties Tab


Overview

The Protocol Window properties tab allows you to view or change the default settings of Protocol window.

Behavior

You can change the background color settings of all three panes and enable or disable the grid lines in the Packet View pane.

To Use

From the Protocol window, click the  button and then click the Protocol Window tab. Click the Background drop-down list to select a background color. Use the Grid Lines check box to enable or disable the grid lines in the Packet View and Hex View panes.

Protocol Column Properties

Overview

The Column properties tab allows you to view or change the column settings in the Packet View pane of the Protocol window.

Behavior


The drop-down Column list shows all columns from the current acquisition. Select the column that you want to view or change from the drop-down list. Use the width box to change the width of the selected column; you can enter a value or use the arrows to change the width.

To hide a column, clear the Show Column selection. The column properties are retained when you want to display the column again.

You can specify a fixed column color by selecting Fixed in the Color section and then selecting one of the fixed colors. You can also use the same color as an existing column by selecting Same As in the Color section and then selecting the column from the drop-down list.

When you select the time stamp column, you can use the Time Stamp Reference selection to set the reference point for calculating time stamps. The Time Stamp Reference selection is only available for time stamp column.

To Use

From the Protocol window, click one of the columns and then click  button. The selected column appears in the Column drop-down list. If necessary select the column that you want to update and use the other settings in the dialog box to change the column width or color.

Protocol Color Properties Tab


Overview

The Protocol Color Properties tab allows you to view or change the color settings of the protocols in the Packet and Decode View pane of the Protocol window.

Behavior

The Protocol Color Properties tab lists all the protocols present in the current acquisition and the available color options. Select a color for each protocol. The selected color is used for display in the Decode View of the Protocol window. A preview is displayed in the Protocol Properties Color tab. This tab allows you to set default colors for all the protocols.

To Use

From the Protocol window, click the  button, and then click the Color tab. Select the protocol from the Protocol list. Use the Color for Protocol drop-down list to select the color for the protocol. Click the Set all to default button to change the color to black for all protocols.

Protocol Marks Properties Tab


Overview

The Protocol Marks properties tab allows you to view or change the color settings of marks placed in the Packet View pane. You can also associate a comment for each mark. The Protocol Marks properties tab lists all the marks and cursors placed in the Packet View. Selecting Show Hairline allows you to enable or disable the hairline properties.

Behavior

Marks in the Packet View pane help you to navigate through the data displayed in that pane. There are two cursor marks, a System Trigger mark, and an unlimited number of user-assigned marks. Marks are used to measure time and mark specific samples for evaluation.

To Use

From the Protocol window, click the  button to display the Protocol Properties dialog box, and then click the Marks tab. Use the Marks drop-down list to select a mark. Use the color drop-down list to select the color for the mark. Enter any comments into the comment text box. Use the Show Hairline check box to enable or disable the hairline properties of the mark.

Protocol Framing Options Properties Tab

Overview

The Protocol Framing Options Properties Tab allows you to set the range of samples in the acquisition memory to analyze and frame. This tab also allows you to set bus-specific settings, such as the port address, bus width, and other settings for the packet data bytes that have to be framed. You can enter framing options for the framer support provided for the specific communication bus, such as the port address.


Behavior

Analyzing data can be memory intensive. It is best to analyze acquisition data in blocks of approximately 100,000. Use the Start and Number of Samples settings to specify the ranges of samples to analyze. You can experiment to find the optimum block size for your situation. The samples are analyzed when you exit the property sheet.

If the bus definition for acquiring data is user-defined, there are no framing options on the Framing Options tab.

If you select Frame Across Gaps, the acquisition will not be considered for framing.

To Use

From the Protocol window, click the  button to display the Protocol Properties dialog box, and then click the Framing Options tab. Enter the port address in the Port Address text box and other options pertained to the bus-specific framers. Use the Enable Gaps check box to enable and disable gaps.

Protocol Search Dialog Box

Overview


The Protocol Search dialog box allows you to search in the Packet View pane of the Protocol window.

Behavior

This dialog box contains all the Search Options. You can select a protocol from the Protocols drop-down list. The Search Options are: Fields, Operation, Value, and Format. You can select an item to search for from the Fields, identify the Operation, and specify the Value and the Format.

You can specify the Value in Decimal, Hex, ASCII, or Binary formats. You can clear the Enable Fields check box and search on the selected protocol. The search is completed once for each packet of data. If the search is successful, the packet is highlighted in the Packet View pane and the corresponding packet information displays in the Decode View pane.

To Use

From the Protocol window, click the  button to open the Define Search dialog box.

Use this dialog box to define operations and specify the value for the search. Select either <<Back or Forward>> buttons to search before or after the current cursor position.

Export Protocol Data Dialog Box

Overview

The Export Data dialog box allows you to export data to a text file in a location that you choose. You can choose options that customize what will be exported.

Behavior

Use the Export Data dialog box to export data from the current Protocol window to an ASCII text file. You can export the following Protocol data:

- All packets from the acquisition

- A range of packets between two selected marks
- A range of packets between two packet indices

If you choose to export the protocol data to a text file, you can click Options to define the format of the data. The data is saved in a file with the .txt extension.

To Use

Place the cursor in the Protocol window, and click Data> Export to display the Export Data dialog box.

New Protocol Display Window

Overview

Use this dialog box to create a new Protocol Display window. To access the dialog box from the Window menu, select New Data Window....

Behavior

Select Data From if you want to use a data source from the current system. If the data source you want is not listed, click the Add Data Source button to add a data source from the TLA system or logic analyzer file to the current system. You can also click the Browse button to search for a desired TLA system file.

Select No data to create an empty Protocol Display window.

Checking the Define Bus check box activates the Select Bus dialog box. Use this dialog box when you want to select a different bus from the data source. You can also create a new bus or modify an existing bus to meet your needs.

NOTE. *If your data source does not have a framer associated with it, the Define Bus check box is automatically selected. Choose an existing bus or define a new one.*

To Use

Specify the data source for your new data window or select No data: Check the Define Bus check box to define your own bus or clear the check box if you want to use the bus from the data source. Click Next, and then enter a name for the new Protocol window. Click Finish to create the new window.

Related Topic

[Creating a New Protocol Display Window \(see page 283\)](#)

Select Bus Page

Overview

The Select Bus page allows you to select a Bus from a list of Available Bus Definitions. You can also create a new bus or edit an existing bus.

Behavior

Clicking either the New...or Edit... button will activate the Define Bus Dialog Box. Clicking Next attaches the selected bus to the data source.

To Use

To attach an available bus to your data source, select the bus definition from the list and click Next. If you want to define a new bus, click New.... To modify an existing bus, select that bus from the list of available busses, and then click Edit....

Define Bus Dialog Box

Overview

Use the Define Bus dialog box to modify an existing bus or create a new one. You can select groups from drop-down lists and then define the data patterns by entering data in the adjacent boxes.

Behavior

The Bus Name text box contains information on a bus that you can edit. For a new bus enter a unique name in the text box. By default, MyBus displays in the Bus Name text box for a new bus; the text description is optional. The Field Definition area describes the details of the bus. Use the drop-down lists to define the field, group name, operation, value, and radix. The Field drop-down list shows all of the groups available in the Setup window; the group that you select will become the data group of the bus. Use the Group Name drop-down list to select the group and pattern for the selections for the field selections.

If the packet length is fixed, enter the packet length in bytes in the Packet Length box at the bottom of the dialog box.

You can enter Binary or Hexadecimal values for Data Valid, Error Condition, Start of Packet, or End of Packet. You can also use don't care values.

Fields

- The Data field is a required field.

- The Start of Packet (SOP) is an optional selection. Use it to define the SOP condition of the bus. Enter the data pattern in the Value box. An SOP or EOP condition is expected for packets.
- The End of Packet (EOP) is an optional selection. Use it to define the EOP condition of the bus. Enter the data pattern in the Value box. When designing the data groups in the Setup window, you need to maintain the bit and byte order such that the least-significant byte follows the most-significant byte.
- The Data Valid condition is optional and is used for defining the Data Valid condition of the bus.
- The Abort Condition is optional and is used for defining the Abort condition of the bus.
- The Port Address is optional and only applies to the inbound port address. You can select any group for the port address. You can enter a mask value for the port address.
- The Port Address Condition is an optional selection. If you enter a port address condition, enter a port address in the Value box.

To Use

Enter a name and a description for the bus. Use the drop-down lists to define the field, group name, and operation. If you are comparing the operation against a value, enter the value and the radix. Click the Add button to add the definition to the list. You can also modify an operation by selecting the operation, making the change, and then clicking the Modify button.

Stack Definition Pane

Overview

The Stack Definition pane allows you to create or modify protocol stacks and displays the properties of the selected protocol. A protocol stack is made up of protocols at different layers. There can be many protocols at a given layer.

Directly below the Stack Definition pane is the Stack Protocol Information pane, which displays the properties of the selected protocol within the stack.

Behavior

The Stack Definition pane allows you to open, edit, and save protocol stacks as well as adding and deleting protocols to a stack. In the Stack Definition pane, you can also define and modify relations between the protocols in a stack.

The Stack Protocol Information pane displays the properties of the selected protocol in the Stack Definition pane. The application displays properties of the selected protocol classified as name, description, compile file name with date and time, source file name with date and time, type of relation, and relation parameters.

Select Stack Dialog Box

Overview

Use this dialog box to select and apply a stack for the current acquisition.

Behavior

The stack that you select has a relationship between the defined protocols. You can select only a single stack. The stack that you select will be applied to the current acquisition when you click OK. To search for a stack, enter a stack name in the Search box and then click the Next button.

To Use

From the Protocol window, click Select Stack in the Data menu. Click the stack that you want to use and then click OK to apply the stack for current acquisition.

Select Stack Page

Overview

Use this wizard page to select a stack from the list of available stacks.

Behavior

Selecting a stack from the list applies the stack to the data source defined in the previous wizard page.

To Use

Select a stack from the list and then click the Next button to continue defining the Protocol window.

Generic Framer Example

This example shows how the generic framer works for Utopia3 support packages.

Data Acquisition

The logic analyzer acquires the Utopia signals using synchronous sampling. Data and control signals are acquired on every rising edge of the clock. The control signals are Enb* and SOC. Set up the Data group and the Control group on the logic analyzer (as described in the following procedure).

Framing

1. In the Define Bus dialog box, enter the Bus name, description, and then select the data group.
2. For the Start of Packet (SOP), select the Control group that contains the SOC and Enb* signals. For example, if the Control group has the Enb* and SOC signals, enter the value of 01 for the SOP (the Enb* signal is low and the SOC signal is high to show the start of the cell).
3. Enter the packet length in bytes. For Utopia the packet length can be 52, 53, 54, or 56. However the packet length will be fixed for a system.
4. Acquire the data using the logic analyzer.

The Generic Framer looks for the 01 pattern in the SOP control group. When the pattern is matched, the framer starts to fill the cells. The data from the present sample is considered as the Start of the cell.

5. Create your own protocol (for example, Utopia3) and stack using the Protocol Designer for further decoding of the Utopia3 cells.

A typical Utopia3 cell format consists of the Header with 6 bytes followed by the payload with 48 bytes. The payload is called SAR (Segmentation and Reassembly) PDU (Protocol Data Unit); it will have AAL type data.

Limitation

You will be unable to decode beyond SAR PDU. (This limitation also applies to Utopia2 and Utopia3 support packages and dedicated framers for Utopia2 and Utopia3.)

Setting Up the Protocol Window

Follow these steps to set up the Protocol window: This procedure assumes that a support package is already loaded on the logic analyzer.

1. From the Window menu, select New Data Window or click the New Data Window icon in the toolbar.
2. In the New Data Window dialog box, select the type of data window as Protocol and click OK.
3. Select Data From, and then select a data source from the list. If the source you want is not listed, click Add Data Source. Select No data to create an empty window.
4. If you want to define a bus to attach to the data source, select Define Bus, or clear the check box to use the bus from the data source.
5. Select a stack from the predefined list.
6. Enter a name for the new Protocol window and click the Finish button to view the Protocol window.

Selecting Stacks

Follow these steps to select the stack:

1. Place the cursor in the Protocol window and click Data > Select Stack.
2. The Select Stack Dialog Box lists all stack files available in a predefined folder.
3. Click the stack that you want to apply for the current acquisition.
4. Click OK to apply the stack for current acquisition.

Filtering the Protocol Display

Follow these steps to filter the Protocol Display:

1. Place the cursor in the Protocol window and click Data > Filter Setup to display the Protocol Filter Setup dialog box.
2. Enable or disable the filter using the Enable Filter check box.
3. Select a protocol from the Protocols drop-down list.
4. Select which fields you want to filter from the Fields text box.
5. Select the operation to be used to compare field values from the Operation group box.
6. Specify the value or select predefined values and the format for the selected field. The packets that have the specified options are filtered and displayed in the Packet View.
7. Clear the Enable Fields selection to filter the packets based on the selected protocol.
8. Click OK.

Viewing Decoded Data in a Protocol Window

Follow these steps to view protocol decoding in a Protocol window:

1. Acquire data or load a reference memory.
2. If you already have a stack defined, create a new protocol window.
3. If you do not have a stack defined, invoke the Protocol Designer window, define the protocols and compile. If required, use one of the defined protocols from the Add New Protocol dialog boxes under the stack editor and define the stack.
4. Apply the stack file to the window and define the framer options. The desired packets from the acquisition (or reference memory) will be framed and decoded.

Protocol Setup Window Properties

Overview

The Protocol tab allows you to set up the name, description, and the color of the selected protocol.

Behavior

The Protocol tab displays the current name, description, conditions of protocol relations, and the color of the selected protocol. You cannot change the conditions of protocol relations in this tab page. You can select a display color and set the default color. The Preview pane displays a preview of the selected color.

To Use

1. In the Stack Definition pane, right-click the protocol you want to edit. The Protocol right-click menu appears.
2. From the Protocol shortcut menu, select Protocol Setup... to display Protocol Setup dialog box.

Protocol Relations Properties

Overview

The Relations tab allows you to define and modify relations between protocols in a protocol stack.

Behavior

The relations are viewed from a higher-order level in the protocol stack and are therefore called "on top of relations".

You can add protocols from the Available Protocols to Current Relations list to create relations between the protocols. You can define the type of relation as Unconditional, Single Parameter, and Raw Data.

- Use Unconditional to specify that the protocol defined as "on top of" is to maintain all protocol data without restriction.
- Use Single Parameter if you want the current protocol to receive data only if a certain parameter of the carrier protocol accepts one or more values. Enter the exact condition in the Name and Value fields.
- Use Raw Data as a condition for transfer of the protocol data. Enter the values of the data frame to be checked in the Byte offset, Value and Bit offset fields.

To Use

1. In the Stack Definition Pane, right-click the protocol you want to define. The Protocol right-click menu appears.
2. From the Protocol shortcut menu, select Modify Relations.... The Protocol Setup dialog box appears, displaying the Relations tab.

Stack Definition Display Properties

Overview

The Display properties option allows you to set the foreground color for the protocols. You can also set the default color.

Behavior

Selecting Set to default will change the foreground color of all the protocols to black.

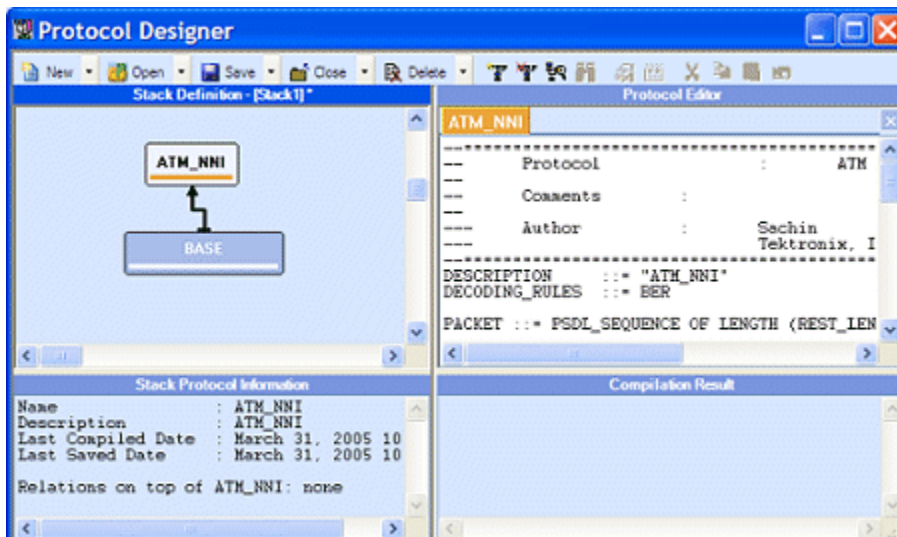
To Use

1. Right-click in the Stack Definition Pane.
2. Select Display Properties.... The Display Setup dialog box appears.
3. Change the properties as necessary.

Protocol Designer Window

The Protocol Designer is an Integrated Development Environment (IDE) that combines the functionality of a protocol and stack editor with a protocol compiler. There are four panes in the Protocol Designer window. Two of the panes are editors: the Stack Definition pane and the Protocol Editor pane. In the Stack Definition pane, you can add protocols, define a stack, and create relations between the protocols in a stack.

The Stack Protocol Information pane displays detailed information on a selected protocol including the relations of that protocol in the stack. The Compilation Result pane displays the results of compiling a protocol. To open the Protocol Designer window, click the Protocol Designer button on the toolbar.



NOTE. Tektronix recommends that you create and save new protocols and stacks using the Protocol Designer tool rather than another editor. You should open and edit existing protocol and stack files using the Protocol Designer. Avoid editing files directly in the Protocol folder and Stack folder.

Protocol Editor Pane

Overview

The Protocol Editor pane allows you to modify existing protocols or create new ones.

Behavior

Use the [Packet Structure Description Language \(PSDL\)](#) (see page 243) for defining and creating the protocols.

You can display several protocols in the Protocol editor at the same time. However, you can only edit or compile one protocol at a time.

When you compile the protocols from the Protocol Editor pane, the results of the compilation are displayed in the [Compilation Result Pane](#) (see page 272).

NOTE. Tektronix recommends that you use the Protocol Editor pane to edit existing protocol or stacks rather than using other editors.

Add Protocol Dialog Box

Overview

The Add protocol dialog box allows you to add protocols from the list of compiled protocol files (.tpa).

Behavior

The list displays descriptions for all available protocols. If you select a protocol description, the associated protocol description file is displayed below the list.

Enter a search string in the Search field to find a certain protocol.

Click Next to search the list for more protocols that match the search string. When you have reached the end of the list, the search will continue from the beginning.

To Use

1. In the Stack menu, click Add New Protocol... to add protocols to the stack.
2. In the Add Protocol dialog box, select a protocol to add and click OK.

Replace Protocol Dialog Box

Overview

The Replace protocol dialog box allows you to replace one protocol with a different version of the same protocol or with a different protocol.

Behavior

The properties (relations and display) of the respective protocols remain the same when replaced with a different version of the same protocol.

To Use

1. Right-click the protocol you want to replace..
2. From the Protocol shortcut menu, select Replace Protocol... to display the Replace with dialog box.
3. Select the replacement protocol and click OK.

Save Protocol or Stack Dialog Box

Overview

This dialog box is used to save a stack or protocol file. To access this dialog box from the toolbar in the Protocol Designer window, select Save, and then choose either Save Stack As... or Save Protocol As....

Behavior

Saving by using a name from the list causes the previously saved file to be overwritten.

To Use

Type the name you want to use to save the Stack or Protocol file and Click Save.

Open Protocol or Stack Dialog Box

Overview

Use this dialog box to open a previously saved stack or protocol. To access this dialog box from the toolbar in the Protocol Designer window, select Open, and then choose either Stack... or Protocol....

Behavior

When you select an available stack or protocol file and click Open, the selected file displays in the corresponding editor pane. You can enter a name (either complete or partial) in the Search field and click Next to search for that file.

To Use

From the Protocol Designer window, click Open and then select either Stack... or Protocol.... Select the available Stack or Protocol file that you want to open. You can search for the file by typing a name in the Search text box and then clicking Next.

Delete Protocol or Stack Dialog Box

Overview

Use this dialog box to delete a previously saved stack or protocol. To access this dialog box from the toolbar in the Protocol Designer window, select Open, and then choose either Stack... or Protocol....

Behavior

When you select an available stack or protocol file and click Delete, the selected file will be deleted from the editor pane. You can enter a name (either complete or partial) in the Search field and click Next to search for that file.

To Use

From the Protocol Designer window, click Delete and then select either Stack... or Protocol.... Select the available Stack or Protocol file that you want to delete; you can select multiple files and delete them in one step. You can search for the file by typing a name in the Search text box and then clicking Next.

Import Stack Dialog Box

Overview

Use this dialog box to import a stack exported from another Tektronix logic analyzer or computer.

Behavior

This dialog box operates the same as most Open dialog boxes except that it looks for Stack files with the .pde file extensions.

To Use

1. Open the Import Stack dialog box, open a Protocol Designer window.
2. From the Stack menu, select Import.
3. Navigate the file system to locate the stack and click Open.

Export Stack Dialog Box

Overview

Use this dialog box to export an opened stack and all of the protocols that belong to the stack to a .pde file. You can then use the stack and protocols on another Tektronix logic analyzer or computer.

Behavior

This dialog box operates the same as most Save dialog boxes except that it saves Stack files with the .pde file extension. The Export selection in the Stack menu is disabled if there is no open stack in the Protocol Designer window.

To Use

1. Open the Export Stack dialog box, open a Protocol Designer window.
2. Open a Stack Definition (click the Open button, select Stack, and select an available stack from the list).
3. From the Stack menu, select Export.
4. Navigate to the location where you want to save the file.
5. Enter a file name and enter comments as needed in the comment box.
6. Click the Save button to export the file to the desired location.

Add Protocols

1. From the Stack menu, select Add New Protocol... to add protocols to the stack.
2. In the Add Protocol dialog box, select a protocol to add and click OK.

NOTE. You can search for the protocols that you want to add by using the Search field, and then clicking Next to find the next occurrence of the search string.

Define Relations Between Protocols

1. In the Stack Definition pane, right-click the protocol for which you want to define relations. The Protocol shortcut menu appears.
2. In the Protocol shortcut menu, select Define Relations. The Protocol Setup dialog box appears, displaying the Relations tab.
3. Select the protocols from the Available Protocols list and click Add.
4. The Current Relations list displays the added protocols and the existing relations.
5. To remove any protocol from the Current Relations list, select the protocol, and then click the Remove button.

6. In the Type of Relation options do one of the following:
 - Select Unconditional to specify that the protocol defined as "on top of" is to maintain all protocol data without restriction.
 - Select Single Parameter if you want the current protocol to receive data only if a certain parameter of the carrier protocol accepts one or more values. Enter the exact condition in the Name and Value fields. Enter hexadecimal values preceded by 0x or binary values preceded by 0b.
 - Select Raw Data as a condition for transfer of the protocol data. Enter the values of the data frame to be checked in the Byte offset, Value, and Bit offset fields. Enter hexadecimal values preceded by 0x or binary values preceded by 0b.
7. Click Up or Down to determine the order of the relations, and then click OK.

Go To Protocol Dialog Box

1. Right-click in the Stack Definition pane. The Protocol shortcut menu appears.
2. In the Protocol shortcut menu, select Go to Protocol. The Go to Protocol dialog box appears.
3. From the Go to protocol drop-down list, select the protocol that you want to view, and then click OK.

Delete Protocols

1. In the Stack Definition pane, right-click the protocol you want to remove. The Protocol shortcut menu appears.
2. In the Protocol shortcut menu, select Remove Protocol to display a message to confirm the deletion.
3. To delete, click Yes. To cancel, click No.

Define Protocol Setup

1. In the Stack Definition pane, right-click the protocol you want to define. The Protocol shortcut menu appears.
2. In the Protocol shortcut menu, select Setup to display Protocol Setup dialog box.

3. In the Protocol tab, do the following:
 - Change the name of the protocol in the Name text box, if required.
 - Enter a description in the Description text box.
 - Use the Color drop-down list to select the foreground color.
 - Click Default to set the color of the protocol to default.
4. Click OK.

Create New Protocol Files

1. From the Protocol Designer window, click the New toolbar button, and then select Protocol to open a blank protocol definition file in the Protocol Editor.
2. Define your protocol in [PSDL Language \(see page 243\)](#).
 - List the description of the protocol. For example: DESCRIPTION::="IP – Internet Protocol, RFC791"
 - Specify the decoding rule to be used. For example: DECODING_RULES ::=BER
 - Define the length of each packet in the protocol. For example: PACKET ::=PSDL_SEQUENCE of LENGTH (PACKET_LENGTH){ PPP_Packet}
 - Define the sequence. For example: PPP_Packet::= PSDL_SEQUENCE "{" <[field name](#)> <[type name](#)> <[field label](#)>"}"
 - Specify the type and size of each variable in the sequence.

NOTE. *Tagging the Payload variable with USER_DATA indicates that the contents of that variable are to be handed to the next protocol layer.*

3. Click the Save toolbar button and select Save Protocol As... to display the Save Protocol dialog box.
4. Type a name for the protocol in the Name field and click Save.
5. Click the Compile toolbar button to compile the protocol. After compilation, the Protocol Designer saves the protocol in .tpa file format.

NOTE. *Any errors in the compilation are displayed in the Compilation Result pane.*

Type Name

Type name is a user-defined name for the type of a variable. It always begins with an uppercase character.

Field Label

The field label is a user-defined, optional name that is displayed in the Field column of the Decode view. It is usually the same as the packet field name.

Field Name

The field name is a user-defined, optional field that is assigned to increase the specification readability. The field name always starts in lower case.

Create New Stack Files

1. Click the New toolbar button, and then select Stack to display the Stack Definition pane with the Base layer and stack protocol information.
2. From the Edit menu, click Add New Protocol to add protocols to the stack.
3. In the Add Protocol dialog box, select a protocol to add and click OK.
4. [Create \(see page 291\)](#) or [modify \(see page 69\)](#) relations between the protocols.
5. The Stack Definition pane displays the protocol and protocol relations.
6. To save the stack, select Save Stack As From the File menu. The Save Stack dialog box appears.
7. Type a name for the stack in the Stack Name field, and then click Save.

Replace Protocol Files

1. In the Stack Definition pane, right-click the protocol you want to replace. The Protocol shortcut menu appears.
2. In the Protocol shortcut menu, select Replace Protocol to display Replace with dialog box.
3. Select the replacement protocol from the Available Protocols list and click OK.

Deskew Analysis Window

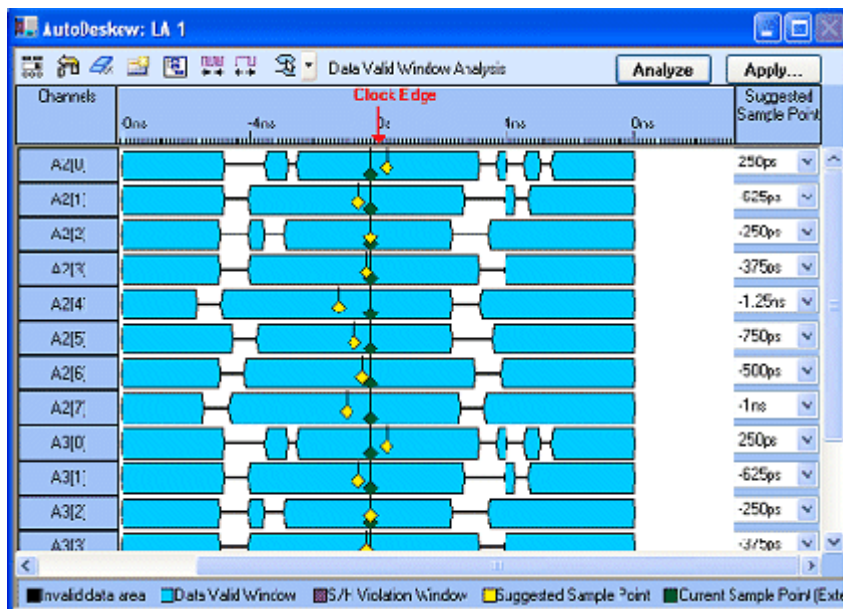
In the toolbar, click the S/H Analysis button.

Overview

The Deskew Analysis window provides the controls necessary for setting up the Deskew Analysis tool, displaying the analysis results, and applying the results for use with normal acquisitions. The Deskew Analysis works independently of the current logic analyzer sampling and trigger program settings. When you set up and perform the analysis, the current sampling and trigger program settings are not disturbed. However, you can apply the results to the synchronous sampling selections to improve the ability of normal acquisitions to find stable data. You can also apply the Deskew Analysis setup and hold violation settings to the trigger program violation settings.


Behavior


Use the Deskew Analysis window to select the analysis type, initiate analysis, apply analysis results, clear analysis results, and reset the setup to factory default settings.



In addition, you can use controls in this window to customize the colors of elements contained in the results area.

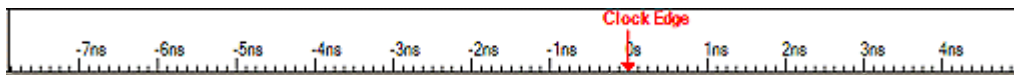
Toolbar

Click the Define Setup icon, , to open the Deskew Analysis Setup dialog box to define a new setup or edit an existing setup. Once you have defined the setup, close the dialog box and then click the Analysis

Type button  to select an analysis type. To begin the analysis, click the Analyze button. If you want to apply the results to one of the synchronous sampling modes, click the Apply button. Similarly when doing a Setup/Hold analysis, you can apply the settings by clicking the Apply button.

Analysis Display Area

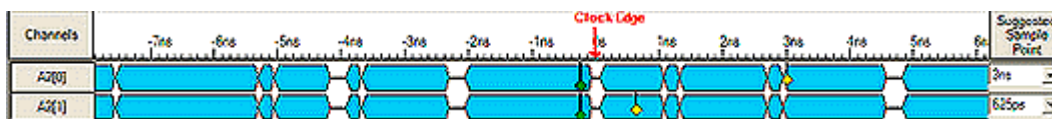
The Clock and Scale area displays clock and scale information. A representation of the reference clock 0 point (the point at which the Deskew Analysis reference clock changes) displays on the top, along with a representation of the analysis range. You can manually set the analysis range from 16 ns before the clock edge to 8 ns after the clock edge using the controls under the Analysis tab under the [Deskew Analysis Setup Dialog Box \(see page 299\)](#) (the Clock 0 point is not required as part of the analysis range). The Clock 0 point enables you to characterize a signal with respect to the reference clock edge. The clock tick intervals of the displayed search range will reflect the MagniVu resolution of the module being analyzed.



The Analysis Results area displays the Data Valid window or Setup/Hold Violation window analysis results in graphical format.

Data Valid Window Analysis Mode

In the Data Valid Windows analysis mode, each selected signal is analyzed with respect to the reference clock edge to determine the regions where the data is stable. This mode determines the transitions between adjacent MagniVu data samples. If two adjacent samples have the same polarity, they are considered stable with respect to each other. The Deskew Analysis scans samples throughout the analysis range, two samples at a time; the results area graphically shows the data valid windows as bars or windows. The bars or windows appear as elongated eyes. The regions between the data valid windows are regions of invalid or unstable data (represented by horizontal black lines).



The purpose of the data valid window analysis is to find regions where the data is stable and to pick a suggested sample point for each analyzed signal. You can then apply the suggested sample points to the signal setup and hold settings for synchronous sampling by clicking the Apply button in the toolbar.

After making all of the settings in the Deskew Analysis Setup dialog box, click the Analyze button to start the data valid window analysis. When the analysis is done, the results appear in the Analysis Display area. The suggested sample points found by the analysis appear as diamond markers. You can readjust the markers by dragging them or by selecting values to the right of the display area.

TIP. To set the Deskew Analysis in the data valid window mode, select Data Valid Window Analysis from the Analysis Type button in the toolbar.

Setup and Hold Violation Window Analysis Mode

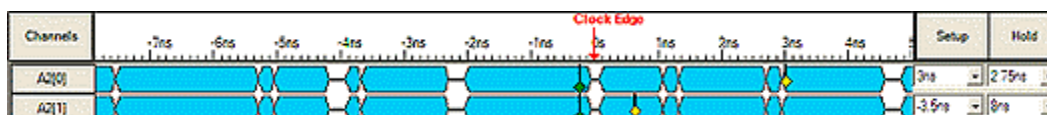
In Setup and Hold Violation Window analysis mode, selected signals are collectively analyzed for setup and hold violations. This analysis mode operates identically to the setup and hold violation trigger programs that analyze groups for violations. The purpose of the analysis is to determine if the data is stable in the regions that you analyzed.

The analysis proceeds throughout the defined setup and hold violation window analysis range for each signal. If any violations are found, they are reported on the entire set of signals being analyzed. Once the analysis completes, you can apply the range settings to the setup and hold violation trigger settings for each signal in the application by clicking the Apply button in the toolbar.

You can change the setup and hold violation range values by doing one of the following:

- Manually dragging each setup and hold violation window bar and resizing it to the MagniVu resolution of the module being analyzed. (Make sure that you select the Setup/Hold Violation Window check box in the [AutoDeskew Properties](#) (see page 301) dialog box so you can drag and resize the window bar.)
- Selecting values for each signal in the controls to the right of the display area.
- Using controls in the Deskew Analysis Setup dialog box under the Analysis tab to set the setup and hold violation range.

You can display graticule lines on both the top and bottom of each group's results by selecting the Show Graticule option in the Deskew Analysis Properties dialog box.



TIP. To set Deskew Analysis in the Setup and Hold Violation Window mode, select Setup/Hold Violation Window Analysis from the Analysis Type button in the toolbar.

Channel Column

The Channel column displays the group, probe, or user name of the displayed signals. The names depend on how the signal is selected in the Deskew Analysis Setup dialog box.

Suggested Sample Point Column

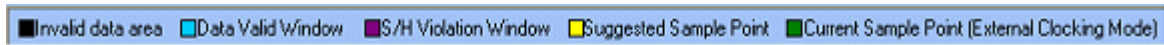
For each signal, the Suggested Sample Point column displays the position of the suggested MagniVu sample found by the Data Valid Window analysis. You can adjust the suggested position for each signal using the controls for this column. The Deskew Analysis must be in the Data Valid Window analysis mode to enable and display this column.

Setup and Hold Columns

For each signal the Setup and Hold column displays the setup and hold analysis range values used for the Setup and Hold Violation analysis. You can adjust the setup and hold values for each signal using the controls to the right of the display area.

Deskew Analysis Legend

The legend displays information for the analysis windows. You can use the Deskew Analysis Properties window to change the color associated with any of the analysis windows. Double-click on the colored square for any item in the legend to open the Deskew Analysis Properties dialog box.



Related Topics

[Deskew Analysis Properties Dialog Box \(see page 301\)](#)

[Deskew Analysis Setups Dialog Box \(see page 302\)](#)

Load Deskew Analysis Setup Dialog Box

Overview

The Load Deskew Analysis Setup dialog box allows you to load a Deskew Analysis setup from any saved system setup file. When you select a Deskew Analysis setup, the load operation replaces all the current Deskew Analysis settings.

Behavior

The operation of the Deskew Analysis Setup dialog box depends on what you select to load. If you select a module, then the selected setups are added to the module. If you select a setup, then the loaded setup replaces the one in the Deskew Analysis Setups dialog box.

Use the File Name box to enter the file name containing an Deskew Analysis setup to load or use the Browse button to browse the file system for a setup. The modules and setup, when found, will be listed in the Modules and their Setups text box. You can select any setup listed in this box. Once you highlight the desired setup, you can load the setup by clicking the Load button.

The Modules and their Setups box allow you to highlight a single selection or multiple selections, based upon the following scenarios:

- When you select a setup, the Modules and their Setups box allows you to select only one setup and, once loaded, it replaces the selected setup.
- When you select a module, the Modules and their Setups box allows you to select multiple setups to be loaded. This operation loads a copy of the selected setups and adds it to the list of setups for the selected module.

Click the Load button to load your selected Deskew Analysis setups. The loaded Deskew Analysis setup either creates a new setup or replaces an existing setup. If the Deskew Analysis setup being loaded uses groups or signals not included in the module, they are discarded and a message informs you of the setup revision. When loading, a Deskew Analysis setup will overwrite an existing setup; a message informs you of the impending change. You can choose to either continue loading the setup or cancel the operation. Setup naming conflicts are resolved by appending a number to the setup name.

Related Topic

[Deskew Analysis Setup Dialog Box \(see page 299\)](#)

Deskew Analysis Setup Dialog Box

Overview

Use the Deskew Analysis Setup dialog box to edit the settings of a selected setup, add a new setup, and specify analysis parameters (such as signals and sampling).

Behavior

The Deskew Analysis Setup box displays all module or instrument names and the name of any setups they contain. Modules that contain one or more setups are listed in < modulename > - <Deskew Analysis setup name> format.

Click the New Setup button to open the New Deskew Analysis Setup dialog box, which you can use to specify a new setup. To change the name or description of an existing setup, use the [Deskew Analysis Setups Dialog Box \(see page 302\)](#).

The Deskew Analysis Mode group box allows you to specify the mode in which the setup will be used. You can choose between Manual mode and Custom mode.

The Manual mode allows you to define the Signal setups and the Sampling setups in their respective tabs. In addition, it disables the Deskew Analysis Configurations box, the Deskew Analysis Settings box, and the Reload Custom Settings button.

The Custom mode derives the settings from the loaded support package if the package includes Deskew Analysis support (not all support packages have such support). Selecting Custom enables Deskew Analysis configurations, Deskew Analysis settings and the Reload custom settings button. Custom is disabled unless a support package with Deskew Analysis support is loaded in the selected module.

The following selections are available when you select the Custom mode:

- Reload custom settings

The reload custom settings button reloads the settings from the loaded support package.

- Deskew Analysis configurations

Deskew Analysis configurations lists all of the configurations defined in the loaded support package. You can create a configuration for each physical configuration of the SUT. If the loaded support package includes the Deskew Analysis support with multiple configurations, you can select the configuration from this list.

- Deskew Analysis settings

Deskew Analysis settings lists all of the settings available for the selected configuration. For each configuration there will be one or more settings. Each setting defines the signals to analyze, a sampling specification, and a range over which each signal will be analyzed. Using a custom setting allows you to override the signals to be analyzed, but there is no way to adjust the analysis sampling or the data valid window analysis range.

Signals Tab

Use the Signal tab to select the signals that you want to analyze and the signals that will have their data valid window analysis results applied. The data valid window analysis results are applied when you click the Apply button and specify the settings in the Apply dialog box.

Use the Signals tab to list signals by group, by probe, or by a user-defined name. Then specify the signals that you want to analyze by selecting the check box associated with each group or signal in the Analyze column. To specify the signals for which the analysis results will be applied when you click the Apply button, select the check boxes associated with each group or signal in the Apply column. To select or clear all Analyze or Apply check boxes, select the check box located next to the Analyze or Apply column title. By default, when you select an Analyze check box, the related Show check box is selected.

Sampling Tab

Use the Sampling tab to set up the definitions for the Data Valid Window and Setup and Hold Violations Window analysis. The Sampling tab contains inputs that you use to specify the clock, clock transition edge, and qualifiers to use for the analysis.

NOTE. When you choose Custom mode, sampling definitions are derived from the support package. All the controls that normally display on the Sampling tab are hidden. The system displays a message indicating that the sampling definitions will be handled by the Deskew Analysis setup you selected from the loaded support package.

Analysis Tab

Use the Analysis tab to specify parameters for Data Valid Window and Setup/Hold Violation analysis.

The Data Valid Window group box contains the Start (Setup) and End (Hold) controls that specify the beginning and end of the analysis range for all signals. Positive values represent times before the clock

edge, while negative values represent times after the clock edge. In the Deskew Analysis Manual mode, the start and end times are applied to all analysis signals and cannot be adjusted on a per-signal basis. In the Custom mode, these controls are disabled. However, the individual values show the aggregate analysis range over all signals specified in the loaded custom setting.

Use the Analysis Quality options to control the amount of time that Deskew Analysis uses to perform the data valid window analysis. Each selection displays the number of samples tested for each pair of MagniVu bits within the analysis range. For example, if your logic analyzer has 125 ps resolution, the number of such tests performed over a 24 ns range is 192. The number of samples that are examined depends on the Deskew Analysis sampling setting. In Deskew Analysis Manual mode using one clock, the number of acquisition samples examined corresponds to the number of clock edges. In the Custom mode, the sampling can be very complex such that the number of acquisition samples does not necessarily correspond to simply looking at a single clock edge.

The Setup/Hold Violation Window group box includes the Start and End controls that specify the setup and hold violation window analysis range for all signals. Positive values represent times before the clock edge, while negative values represent times after the clock edge. These controls are not affected by the Deskew Analysis Custom settings. Use the Stop Analysis Condition options to specify when to stop the analysis. Select Run to let the analysis continue until you click the Abort button in the progress dialog box.

Deskew Analysis Properties Dialog Box

Overview

The Deskew Analysis Properties dialog box contains controls that affect the display properties of the Deskew Analysis window.

Behavior

By selecting properties from this dialog box, you control which windows display in the analysis results area of the Deskew Analysis window, and the colors used to display them. The property selections will not display in the window if you do not select the check box.

The Show Graticule box allows you to indicate whether to display the graticule and specify the graticule color. You can also define the following settings for the Deskew Analysis window:

- The Invalid Data Region check box allows you to display the invalid data region using the specified color.
- The Data Valid Window check box allows you to display the Data Valid Window using the specified color.
- The Suggested Sample Point check box allows you to display the suggested sample point for each signal using the specified color.
- The Setup/Hold Violation Window check box allows you to display the Setup/Hold Violation window for each signal using the specified color.

- The Current Sample Point check box allows you to display the current sample point for each signal using the specified color. The actual sample point depends on the current sampling type.
- The Current Sampling Mode group box allows you to select synchronous sampling. The mode you choose controls the current sample point displayed in the Deskew Analysis window.

Related Topic

[Deskew Analysis Window \(see page 295\)](#)

Deskew Analysis Setups Dialog Box

Overview

The Deskew Analysis Setups dialog box displays the setups available in all physical modules. You can also view saved setups that were downloaded from a module that was loaded from another system.

Behavior

You use the Setups group box to manage Deskew Analysis setups. This includes creating, duplicating, deleting, and renaming setups. The type of setup you highlight in the Setup box (module vs Deskew Analysis setup) determines the functionality associated with the following toolbar icons. Right-click a setup to open a menu where you can manage the setups.

Click the New button to open the New Deskew Analysis Setup dialog box. Use this dialog box to create and name a new setup. When a module or an instrument is highlighted in the Setup box, the new setup is attached to that module or instrument. When a setup is highlighted, the new setup is associated with the highlighted setup. You can also use the New Deskew Analysis Setup dialog box to rename an existing setup.

Click the Load button to open the [Load Deskew Analysis Setup Dialog Box \(see page 298\)](#). Use this dialog box to load a setup from a saved system file. When a module is highlighted in the Setup box, the loaded setup is added as a new setup to the selected module or instrument. When a setup is highlighted, the loaded setup replaces the highlighted setup.

When a module is highlighted in the Setup box, the Cut button is disabled. When a setup is highlighted, clicking Cut copies the selected setup to the clipboard and removes it from the module. You cannot remove the last setup.

When a module is highlighted in the Setup box, the Copy button is disabled. When a setup is highlighted, clicking Copy copies the selected setup to the clipboard.

When a module or an instrument is highlighted in the Setup box, clicking the Paste button creates a new setup from the clipboard contents and attaches it to the selected module or instrument. The logic analyzer will add an integer to the clipboard setup name; you can change the description as needed. When a setup is highlighted, clicking the Paste button replaces the highlighted setup with the contents of the clipboard.

Related Topic

[Load Deskew Analysis Setup Dialog Box \(see page 298\)](#)

Apply Deskew Analysis Results Dialog Box

Overview

The Apply Deskew Analysis Results dialog box allows you to apply the Data Valid Window analysis results to current sampling mode or the Setup/Hold Violation Window analysis settings to the setup and hold storage violation or setup and hold event violation trigger settings. You can apply the data valid window analysis result to only the current sampling mode.

Behavior

The analysis results are applied only to those signals that you select in the Signal Setup tab of the Deskew Analysis Setup dialog box.

Selecting the Apply Suggested Setup/Hold Window Settings option enables the Sampling Mode box. Depending on the sampling, the setup and hold alignment values that you specified in the Deskew Analysis Setup dialog box will be applied to the setup and hold storage violation and event violation. The suggested sample points are actual MagniVu samples, while the setup and hold alignment values in the sampling dialog boxes are probe-tip values. Deskew Analysis does the necessary conversion between the suggested sample points and the setup and hold alignment values. The applied changes will display in the Deskew Analysis window.

If you select the Four Samples per Clock mode, then the Second Edge Delay group options are enabled. When you select the Set External Setup/Hold Values, the suggested sample points are used to set the Four Samples per Clock setup and hold alignment values. The applied changes will display in the Deskew Analysis window.

When you select Set External Second Edge, the suggested samples are used to set the External Second Edge Delay values. Deskew Analysis makes the necessary adjustments to the suggested sample point position to meet the constraints for a second edge delay value if the suggested sample point position violates those constraints. The applied changes will display in the Deskew Analysis window, but you can move the suggested sample point positions to meet any constraints.

Selecting the Apply Setup/Hold Violation Settings option disables the Sampling Mode box and Second Edge Delay group. This option applies setup and hold violation settings. You can adjust the setup and hold settings on a per-signal basis if they are not consistent with the per-signal alignments of the current synchronous sampling mode.

Related Topic

[Deskew Analysis Setup Dialog Box \(see page 299\)](#)

Data Measurements Overview

Data measurements can be taken on Waveform and Listing windows.

Overview

You can select the data measurements that you want to take on an active window from a list of available measurements. Data measurement results are displayed in the Statistics page of the Measurement tab and updated each time you run an acquisition. You can, however, configure data measurements without the acquisition data. You can quickly perform counts and averages on your data set. Data measurements can be exported to a file and saved with your TLA system.

Behavior

When system changes are made that affect the measurement results an icon appears in the Info column on the Statistics page.

To perform measurement analysis on virtual data, use the Recalc button. Clicking Run does not have any affect on the measurement analysis results if you are using virtual data.

Changes in the TLA setup or system will effect the data measurements in the following ways:

- If you delete a group, the measurement using the group is also deleted.
- If the active window that had data measurements taken on it is deleted from the system, all the measurements done on that window are also deleted.
- If you merge data sources, all measurements taken on that window are removed.
- If you return to a default setup, all data measurements are removed.
- Loading a TLA data window that had data measurements taken, restores the measurements that were taken on this window.
- Data measurements to not include suppressed or filtered samples.
- If the radix of any group is changed, the results and statistics for measurements done on that group is also updated.

- Measurement results are cleared when the module is turned off and Run is clicked.
- If the module is turned on in the System Window, measurement results are updated when you click Run.

LA Measurement Setup

Overview

The Setup page contains the controls necessary to add a new measurement. You can also enable or disable the measurements listed and specify whether to refresh the measurements results on successive acquisitions or accumulate results and statistics over multiple acquisitions.

Behavior

The Setup page is affected by the active data window. If no data window is active, then the Measurement tab is unavailable.

Removing a signal or group from a data window causes the associated measurement to be removed from the Setup page.

The icons in the Add Measurement group box display measurements that are available to add. You can create new measurements by dragging and dropping the measurement icons onto the data window display area. These measurements are then added to the Measurement grid. You can also create measurements by right-clicking and selecting Add LA Data Measurement from the right-click menu.

For a definition of each measurement, click one of the measurement type buttons:

The Measurements grid contains all the measurements that are added to the active data window. Selected measurements are enabled. Clearing the Enable check box disables the measurement. When a measurement is disabled, it will not be computed.

You can rename the measurement in the Name column if you so choose. The Source column reflects the signal or group on which the measurement is done. You cannot edit the Source column. The Gate column offers three choices: Display, Cursors, and Full Record for gating. Measurement analysis is immediately performed whenever this column is changed.

The type of measurement gating determines what samples are used in the measurement analysis.

Gating type	Measurement analysis
Full Record—spans across the full memory depth	Samples in the full memory depth are used for data analysis.
Display— spans across the visible region	Samples in the visible portion of the data window are used for data analysis.
Cursor— spans between Cursor1 and Cursor2	Samples between Cursor1 and Cursor2 are used for data analysis.

Measurement analysis is performed when:

- The Gate option on the Setup page is changed.

- The Gate option is set to Display and you zoom on the active data window.
- The Gate option is set to Display and data is suppressed or suppressed data is shown.
- The Gate option is set to Display and data is filtered.
- The Gate option is set to Cursors and you change the cursor positions on the active data window.

When measurement analysis is in progress, you can abort the analysis by clicking the Cancel button. You can also recalculate a completed analysis by clicking the Recalc button. This button toggles between Cancel and Recalc, depending on the state of the analysis.

To Use

Select a measurement by dragging and dropping an icon from the Add Measurement group box to a data window.

Select the Enable check box from the Measurements grid to enable the measurement. To disable the measurement, clear the check box. You can also enable or disable all measurements by clicking the Enable All or Disable All button. You can delete all measurements from the Measurements grid by clicking the Delete All button, or you can right-click and delete them individually.

The Clear Values button removes the results. If you want to accumulate results over several acquisitions, select the Accumulate check box. To refresh the results each time the measurements are analyzed, clear the Accumulate check box.

Data Measurement Statistics

Overview

This page displays the measurement results and statistics for the active data window. When the active window changes so will the Statistics page. If the active data window does not have any measurements, then the Statistics page is empty. The measurements are displayed in the order in which they are added.

For a definition of each available statistic, click [here \(see page 111\)](#):

Behavior

Removing a signal or group from a data window causes the associated measurement result to be removed from the Statistics page.

None of the columns are editable, but you can auto size the column by clicking the right border.

The Info column indicates that the results are outdated due to TLA system or setup changes. Data can be outdated for the following reasons:

- Group definition was changed
- The measurement was aborted while the analysis was in progress

- There is no data for analysis
- The results and statistics are cleared

To Use

Click Export to export the data to a file. You can export in either .TXT or .CSV file format.

To refresh outdated results, click the Run button in the TLA System window to acquire new data or click the RECALC button on the Measurement Setup page to recalculate the measurements on existing data.

Click the Clear Values button to remove all the measurement results and statistics from the Statistics page.

Graph Window


Overview

Use the Graph window to display data in an X-Y graph format. Use this format to identify valid data in patterns such as tracking the flow of digitized analog values throughout your target system by looking at data values written to a set of memory addresses.

Axis readout bars provide information on the data in the Graph window. You can customize the readouts using the Graph Window property page. A legend at the bottom of the graph display provides an overview of the contents of the Graph window. A progress bar below the legend shows the status of the graph when it takes time to update the graph data.

Data Series


Graph data is represented by a data series, a collection of related data points plotted in a graph.

- Each data series in the window can have unique color or patterns.
- Each data series can have a unique title. You can specify the minimum and maximum data ranges for the series as well as an offset for each axis.
- Add a new data series by clicking the Add Series toolbar button .
- Remove a data series by selecting it, and then clicking the Cut toolbar button. You can paste the removed data series by clicking the Paste toolbar button.

Marks

Use marks to take measurements, locate triggers, and mark specific events for evaluation.

- Marks you can move are cursors X1, Y1, X2, Y2, and any marks you add. The numbers 1 and 2 on the marks handles denote the X-axis (vertical) and Y-axis (horizontal) cursor marks. Readouts for the cursors, and the time differences (Delta Time) between the cursors are located at the top of the data area.
- Marks on the top and right sides of the window are data area marks. These marks are in the portion of the acquisition that is currently viewable. Use data area marks to move within the data area.

- You cannot move System Trigger, Module Trigger, or Begin and End marks.
- Marks on the bottom and left sides are overview marks. These marks show where marks are located relative to the entire acquisition. Use overview marks to move throughout the acquisition.
- Double-click a mark to open the Marks properties page. You can assign a unique name to the Mark in the Mark Name field.
- You can add marks by clicking  (the Add Mark toolbar button). Remove a mark by selecting the mark and using the Cut toolbar button. When you add or paste a mark, it appears in the center of the display.

System trigger



The system trigger is the reference point for the acquisition. Timing and location information is relative to the system trigger. Trigger marks cannot be moved.

Under some conditions the system trigger associated with a module's data might not be displayed in the data window. If the system trigger was caused by another module, whose data is not included in the current display, then the system trigger is not shown. Nevertheless, all time measurements still relate to that system trigger, even if it is not shown.

The system trigger associated with the current data is called the active system trigger. The system trigger associated with saved data is called the reference system trigger. The active system trigger is indicated by a yellow T; the reference system trigger is indicated by a gray T.

Module trigger



The point at which the module triggered. Trigger marks cannot be moved.

Begin data and end data



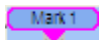
The start and end of a module's data record. These data marks cannot be moved.

Cursors 1 and 2



Moveable marks used for visual reference and for data measurements.

User mark



User-created marks. Use marks to make specific data easier to identify and find.

Lock Delta Value check box

Click this check box to lock the delta values. When you lock delta values, the cursors move across the data simultaneously, maintaining a fixed delta time.

■ Toolbar Area

Toolbar buttons allow you to customize the Graph window. You can access most of the Graph window controls directly from the toolbar buttons. Other buttons open either dialog boxes or property pages where you can supply specific information to adjust the Graph window. If you are unsure of the purpose of the individual buttons, use the tooltips to identify the buttons.

Add Series



Click the Add Series toolbar button to open the Add Series Dialog Box where you can add a new data series from different data sources.

Add Mark



Click the Add Mark toolbar button to add a mark to the data points in the series.

Cut Copy, Paste



Use these buttons to cut, copy, and paste the individual data series multiple data series, or marks.

Go To



Click the Go To button to open the Go To dialog box so you can easily go to a mark or data series.

Properties



Click the Properties button to open the Graph window property pages where you can change the properties of the data or the graph.

Automatic Update



Click this button to enable or disable the automatic update tool. When you select Enable Auto Update, the current data in the window is automatically updated when new acquisition data is available.

Hit Frequency Mode



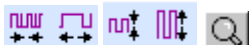
Click the Hit Frequency toolbar button to display the data points using the colors defined in the Hit Frequency property page. The color of the data points depend on the number of hits.

Persistence Mode



Click the Persistence toolbar button to retain the previous graph data along with the current graph data.

Zoom



You can zoom Graph window data similar to data in a Waveform window. The horizontal and vertical axis each has their own zoom controls. Click on one of the zoom buttons to zoom either in or out. You can also click and drag the mouse over the area that you want to zoom; when you release the mouse button, zoomed data will appear in the center of the Graph window.

Abort



Use the Abort toolbar button to stop the processing of data in the graph. The data series will not be updated. You may want to abort the data processing if the data takes too long to process due to size and other reasons.

- You can add a data series to the graph; add marks; and cut, copy, and paste a data series.
- You can enable automatic updates, the hit frequency mode, and the persistence mode by clicking the appropriate toolbar buttons.
- Use the zoom toolbar buttons to zoom in and out on the horizontal and vertical axis, or zoom to the current selection. When you use the zoom buttons, the display centers at the current location.
- Click the Abort toolbar button to abort any updates that may take a long time due to the amount of memory involved with the graph data.

Graph Window Properties

Overview

Use this property page to change the default appearance of the Graph window.

Behavior

Use the controls in this property sheet to help you do the following tasks:

- Display the graph graticule
- Specify the color, style, and line thickness of the graticule
- Specify the minimum and maximum ranges of the X-axis and Y-axis as well as specifying the values per division
- Label the X-axis and Y-axis
- Specify the readout radix
- Turn the graph readouts on or off
- Specify the color of readouts
- Specify the background color of the graph
- Enable or disable automatic updates of the graph. (You can also enable or disable the automatic updates by clicking the toolbar button in the Graph window.)

Series Data Properties

Overview

Use this property page to change the default settings of an individual data series.

Behavior

Use the Series drop-down list to select the series that you want to display in the Graph window. The text below the Series List displays the data source (logic analyzer module and slot location). You can change the title of the series by entering a new title in the Title box. Otherwise, you can use the default name.

The logic analyzer obtains the axis data range values from the selected series. You can adjust these values or add an offset value to either axis by entering values in the boxes.

You can normalize the data range for each axis. When you normalize the data ranges, you rescale the data in the ranges. For example, if you normalize all of the X-axis data to 1, all of the data would fall in proportion to its original values, between 0 and 1. This can be useful when you want to redraw the X-axis and Y-axis for normalized data series where you want to analyze vector displacement, which uses the origin of the graph (0,0) as part of the data measurement. The default range values are -1 and 1, but you can change these values as necessary to meet your needs.

Series Format Properties

Overview

Use this property page to define the appearance of the selected data series.

Behavior

The controls in this property page allow you to select one or more data series where you can specify the line type and data point type. You can further define the color and thickness of the lines or data points. A sample image of the selected data series gives a preview of the data series format.

To Use

Select the series that you want to change from the Series List. Click the Series Type drop-down list to select the series type, straight lines with data points, straight lines without data points, curved lines with data points, curved lines without data points, or data points only. You can further define the lines as dotted, dashed, or solid. You can select the type of data points, dots, plus signs (+), or asterisks (*). You can change the thickness of the lines and marks as well as specify the colors of the lines and data marks.

If you select a series type without data points, the data point elements are disabled. Similarly, if you select a series type with data points only, the line elements are disabled.

Hit Frequency Properties

Overview

Use this property page to define the colors for hit ranges.

Behavior

This window uses color schemes to help you determine the frequency of hits. The color schemes are similar to those used by Tektronix TDS oscilloscopes. For example, if you select a temperature color scheme, colors move from dark (cool) to bright (hot). Alternately, if you select a spectral color scheme, the colors move from bright to dark. The colors are fixed when you select a predefined color scheme. If you select the User-Defined color scheme, you can define your own colors.

The color ranges move from the maximum number of hits to minimum as indicated in the property page. If you click Invert Colors, the ranges move from the minimum number of hits to maximum.

To Use

Use the drop-down list to select a color scheme. If you select the User-Defined color scheme, you can define the colors by clicking the color buttons and selecting a color. Use the Default button to return the colors to their default settings.

After you close the property sheet, click the Hit Frequency button on the toolbar to display the data points in the graph based on the number of hits.

New Graph Data Window

Overview

Use this page to create a new Graph window and specify the data source for your new window.

To Use

- Click Data from to select a data source from the current system. If the data source you want is not listed, click the Add Data Source button to add one from the TLA system or logic analyzer file to the current system. You can also click the Browse button to search for a desired TLA system file.
- Click No data to create an empty Graph window. After the window displays, you can add data series from different sources.

Related Topic

[Adding a New Series \(see page 314\)](#)

New Graph Window Select Series

Use this page to specify the X-axis and Y-axis for the new Graph window.

For either axis, you can select one of the following:

- Sample Number. Use the drop-down list to select Begin, Trigger, or End.
- Row Number. This selection is the same as the Sample Number but does not include any gaps due to filtering.
- Timestamp. Use the drop-down list to select either an Absolute timestamp or the Previous timestamp.
- Value of Group. Use the drop-down list to select an Address, Data, or Control group.

The axis label will be the same as the type that you specify from the above list. After you set up the new window, you can use the [Graph Window \(see page 307\)](#) to further define each axis of the graph.

NOTE. *Once you have specified the axis type using the wizard, you cannot change the type without creating a new Graph window.*

Related Topic

[Add Series Dialog Box \(see page 313\)](#)

New Graph Window name

Enter a name for the new Graph window or use the default name. Then click the Finish button to create the new Graph window.

Add Series Dialog Box

Overview

Use this dialog box to add a data series to the existing graph from different data sources.

Behavior

The default X-axis and Y-axis data sources depend on the initial graph. You specify these when you create a new data window using the New Data Window wizard.

You can add a new series from different data sources such as a logic analyzer module or a reference memory. However, the data source must be from the same Data Source type. For example, you can't add an address group to the X-axis if the Data Source type was set up as a sample number. If you want to change the Data Source type, create a new Graph window through the New Data Window wizard.

The appearance of the dialog box changes depending on the Data Source type. For example, if you defined the X-axis Data Source type as Row Number, the selection is fixed to that type and is grayed out. If you defined the X-axis as a group, you can use the drop-down list to select the individual group or click the Add Data Source button to search the file system for a group.

To Use

Define the series for each axis and then click the Add Series button to add the series to the Series list. The series will be displayed in the form of X-axis versus Y-axis. To remove a series from the graph, select the series in the Series list and then click the Remove Series button.

To access this dialog box, click the Add Series button in the Graph window toolbar.

NOTE. *If you have an empty Graph window (a Graph window with no data) and you click the Add Series toolbar button, you will launch the Axis Type wizard where you can select the data source, define the X-axis and the Y-axis, before opening the Add Series dialog box.*

Graph Window Go To Dialog Box

Overview

Use this dialog box to locate a mark or a series in the Graph window.

Behavior

Use to scroll the Graph window to a new position by selecting any current mark or series. Select Mark to display a list of all marks related to the Graph window. Select Series to display a list of all series related to the Graph window.


To Use

Double-click on the desired mark or series in the list. The dialog box closes and displays the mark or series in the center of the window.

To open the Go To dialog box, open a data window and then click the Go To toolbar button.

Adding a New Series

1. In the System window, click the Graph data window icon.
2. From the Edit menu, click Add Series.
3. Select the Data Source.
4. If the data source you want is not listed, click Add Data Source to open the Add Data Source dialog box and browse the file system to locate the data source.
5. Select the series and click Add.
6. Click OK.


TIP. Click  to add a series.

Taking Cursor Measurements

In the Graph window use the X and Y cursors to take measurements:


1. Move each cursor to the points you want to measure.
2. The measurements are displayed in the cursor readouts.
3. Use Lock Delta Time to lock the cursors together.

Going to a Specific Series

1. In the System window, click the Graph data window icon.
2. Click  in the toolbar of the Graph window to open the Go To dialog box.
3. Select Series to display a list of all series in the Graph window.
4. Double-click the series that you want to see.

The series that you selected will display in the center of the Graph window.

Going to a Specific Graph Window Mark

1. In the System window, click the Graph data window icon.
2. Click  in the toolbar of the Graph window to open the Go To dialog box.
3. Select Mark to display a list of all marks in the Graph window.
4. Double-click the mark that you want to see.

The mark that you selected will display in the center of the Graph window.

Zooming a Graph

The zoom behavior is controlled by the [System Options: Preferences \(see page 85\)](#) settings. The default setting is to zoom immediately. You can zoom a Graph window three different ways:


- Zoom both the X-axis and Y-axis

- Zoom the horizontal (X) axis
- Zoom the vertical (Y) axis

To zoom a Graph window using the default zoom setting (immediately), perform the following steps:

1. In the System window, click the Graph data window icon.
2. Click and drag the mouse over the area that you want to zoom. When you release the mouse button, the Graph window will zoom the area of interest and center it in the window.

Viewing Persistence Data

1. In the System window, click the Graph data window icon.
2. Click  in the toolbar of the Graph window to open the Graph window property pages.
3. Click the Series Data tab.
4. Enter an offset value for the X-axis or Y-axis data range.
5. Click OK to close the dialog box.
6. Click the Hit Frequency toolbar button to enable the Hit Frequency mode (button should appear depressed).
7. Click the Run button to start an acquisition.

The number of hits will increase due the overlapping of data on the same region.

NOTE. *The Persistence data will not be visible when you resize the Graph window or scroll data in the Graph window.*


iVerify Window


From the toolbar, click the iVerify button.

Overview

iVerify allows you to generate eye diagrams, or voltage/time graphs, for both individual signals and groups of signals. Use the controls in the iVerify window to set up iVerify; and to analyze, display, and perform measurements on the eye diagrams.

iVerify Toolbar

Click the Define Setups icon, , to open the iVerify Setups dialog box to define a new iVerify setup or edit an existing setup. Once you have defined the setups, close the dialog box and click the Analyze button to begin building the eye diagram. The eye diagram will be drawn, based on the eye coloration selections made in the Eye Coloration tab of the iVerify Properties dialog box.

Click the Measurement icon, , to display a list of measurements for the eye diagram. Click a measurement button to perform a measurement on the eye diagram. Click the button a second time to turn off the measurement. When the measurements are made, the corresponding cursors are drawn on the eye diagram. In case of histogram, the cursor is drawn on the eye diagram and the corresponding histogram is drawn on top or left of the eye diagram, depending on whether it is a horizontal or vertical histogram.

Selecting any measurement from the Measurement icon in the toolbar will overlay the measurement diagram on the eye diagram and display the corresponding measurement tab.

Measurement Information View

This area of the iVerify window provides general information about the eye diagram. In addition, it provides information about each measurement on the eye diagram. Measurement information parameters and measurement results are shown in separate tabs.

■ General tab

When analysis is complete, the General tab displays peak-to-peak information. In addition, it displays information pertaining to your cursor's present position over the eye diagram such as the time and volt value, number of hits, and displays the highlighted signal.

■ 4 Point Mask tab

The 4-Point Mask tab displays 4 point mask measurement information, such as the mask coordinate values and the number of hits for each signal within the 4 point mask.

■ 6 Point Mask tab

The 6-Point Mask tab displays 6 point mask measurement information, such as the mask coordinate values and the number of hits for each signal within the 6 point mask.

■ Eye Limit Cursor tab

The Eye Limit Cursor tab displays Eye Limit Cursor measurement information, such as the cursor position and the eye height and width.

■ Upper Lower Mask tab

The Upper Lower Mask tab displays upper mask and lower mask measurement information, such as the upper mask voltage value, lower mask voltage value, the number of hits for each signal in the masking region, and upper violation and lower violation.

■ Slope tab

The Slope tab displays information about the slope measurement such as the upper point coordinates, lower point coordinates, and slope value.

- Horizontal Histogram tab

The Horizontal Histogram tab displays the voltage level of the histogram cursor value. In addition, it displays information pertaining to the cursor's present position over the eye diagram, such as the number of hits for each signal over the time level.

- Vertical Histogram tab

The Vertical Histogram tab displays the time level of the histogram cursor value. In addition, it displays information pertaining to the cursor's present position over the eye diagram, such as the number of hits for each signal at that voltage level.

Display Time and Voltage Ranges

Use the Display Time Range and Display Voltage Range to select the horizontal and vertical display settings for the eye diagram. These controls do not affect the acquisition of data from the external oscilloscope.

The Min and Max boxes allow you to set the minimum and maximum time value from where iVerify will plot the data. These boxes allow you to choose start and end points for the eye display from the data acquired by the external oscilloscope. Use the Min and Max controls to choose the time period of interest from all data acquired by the external oscilloscope. By default, the external oscilloscope displays all acquired data.

The Per Div boxes allow you to divide the eye display area into equal horizontal and vertical divisions and spread the values accordingly.

The Offset box allows you to move the eye diagram to the left or right by the values specified in the Offset edit controls. You can also use this offset to measure the time between two points.

NOTE. *The Min and Max time controls are relative to the clock shown above the diagram display area.*

Related Topics

[iVerify Properties Dialog Box \(see page 322\)](#)

[iVerify Setup Dialog Box \(see page 320\)](#)

iVerify Setups Dialog Box

Overview

The iVerify dialog box displays iVerify setups that are available in all physical TLA7000 Series logic analyzer modules. You can also view saved iVerify setups that were downloaded from a module that was loaded from another system.

Behavior

You can use the iVerify Setups group box to manage iVerify setups. This includes creating, duplicating, deleting and renaming setups. Right-click an iVerify setup to open a menu where you can manage the setups.

Click the New button to open the New iVerify Setup Dialog Box. Use this dialog box to create and name a new iVerify setup under a selected module. When a module is highlighted in the Setup box, the newly added iVerify setup is attached to that module.

Click the Load button to open the [Load iVerify Setup Dialog Box \(see page 319\)](#). Use this dialog box to load an iVerify setup from a saved file. When a module is highlighted in the Setup box, the loaded setup is added as a new setup to the selected module. When an iVerify setup is highlighted, the loaded setup replaces the highlighted setup.

When a module is highlighted in the Setup box, the Cut button is disabled. When a setup is highlighted, clicking Cut copies the selected setup to the clipboard and removes it from the module. You cannot remove the last setup.

When a module is highlighted in the Setup box, the Copy button is disabled. When a setup is highlighted, clicking Copy copies the selected setup to the clipboard.

When a module is highlighted in the Setup box, clicking the Paste button creates a new setup from the clipboard contents and attaches it to the selected module. The logic analyzer will add an integer to the clipboard setup name; you can change the description as needed. When a setup is highlighted, clicking the Paste button creates a new setup from the clipboard contents and attaches it to the selected module.

Related Topics

[Load iVerify Setup Dialog Box \(see page 319\)](#)

[iVerify Setup Dialog Box \(see page 320\)](#)

Load iVerify Setup Dialog Box

Overview

The Load iVerify Setup dialog box allows you to load an iVerify setup from any saved system setup file. When you select an iVerify setup, the load operation replaces all the settings. However, when you select a module, a new iVerify setup is added.

Behavior

The operation of the iVerify Setups dialog box depends on what you select to load. If you select a module in the iVerify Setups dialog box, you can select multiple setups in their individual setup boxes. If you select a setup in the Load iVerify Setup dialog box, then the loaded setup replaces the one in the iVerify Setups dialog box.

Use the File Name box to enter the file name containing a module setup or use the Browse button to browse the file system for a setup. The modules and their setups, when found, will be listed in the Modules and their Setups text box. You can select any setup listed in this box. Once you highlight the desired setup, you can load the setup by clicking the Load button. The loaded setup either creates a new setup or replaces an existing one.

If you select a module in the iVerify Setups dialog box, the Modules and their Setups box allows you to select multiple setups. However, if you select a setup in the iVerify Setups dialog box, then the Modules and their Setups box allows you to select only one setup.

If you select a setup from the iVerify Setups dialog box, and then select a setup from the Modules and their Setups box and click the Load button, a system confirmation message displays. Click OK to confirm that you intend to replace the current setup with the setup you selected in the Modules and their Setups box.

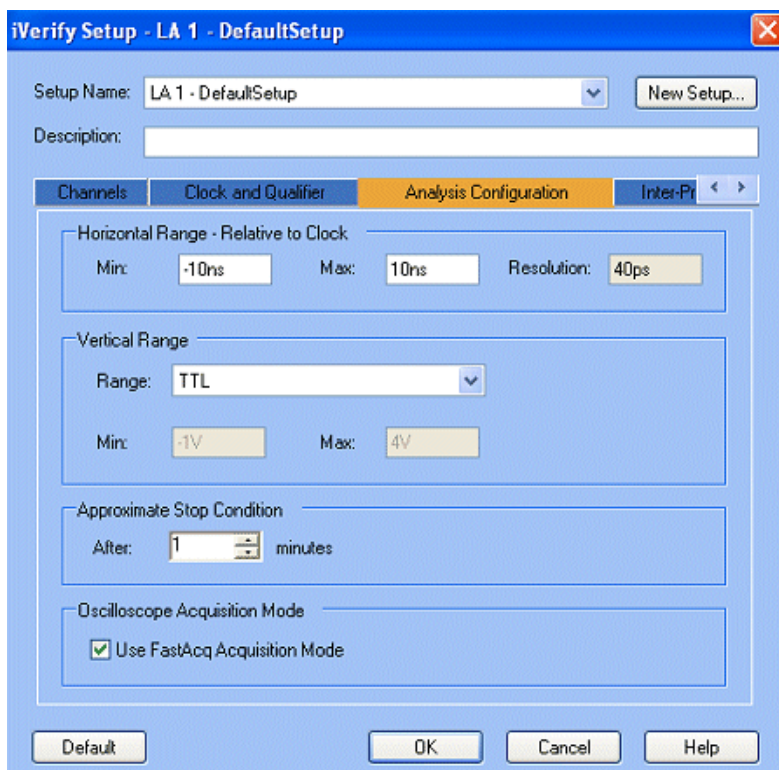
NOTE. *An error message displays if the setup you are trying to load uses groups or signals that are not included in the selected module. Also, an error message displays when you try to load a module without any saved data.*

Related Topic

[iVerify Setups Dialog Box \(see page 318\)](#)

iVerify Setup Dialog Box

The iVerify Setup dialog box allows you to edit the settings of a selected iVerify setup or add a new iVerify setup. In addition, this dialog box allows you to define various analysis parameters through the respective tabs. Note that only TLA7000 Series logic analyzer modules and related setups display in the iVerify Setup dialog box.



Behavior

Use the Setup Name box to select the modules and the iVerify setups that they contain. Modules that contain one or more iVerify setups are listed in <modulename> - <iVerify setup name> format.

The Description box allows you to enter a new description for, or change the existing description of, the loaded iVerify setup. Click the New Setup button to open the New iVerify Setup dialog box, which you use to specify a new iVerify setup.

NOTE. You need to provide information in the Channels, Clock and Qualifier, and Analysis Configuration tabs before viewing the Inter-Probing tab. This is because the optimum analog routing information contained in the Inter-Probing tab depends upon the information contained in these tabs. If the setup that you have specified is physically invalid, an error message appears.

Related Topics

[iVerify Setup Dialog Box: Channel Tab \(see page 329\)](#)

[iVerify Setup Dialog Box: Clock/Qualifier Tab \(see page 330\)](#)

[iVerify Setup Dialog Box: Analysis Configuration Tab \(see page 327\)](#)

[iVerify Setup Dialog Box: Inter-Probing Tab \(see page 331\)](#)

iVerify Properties Dialog Box

The iVerify Properties dialog box shows the properties of the iVerify Window. These properties include information about the logic analyzer and external oscilloscope modules, functionality that you use to set eye diagram colors, and functionality that you use to color the measurement diagrams and set their properties.

Behavior

The contents of the iVerify Properties dialog box are logically organized into three tabs. Click the tabs to navigate to the contents for each iVerify Properties tab.

Related Topics

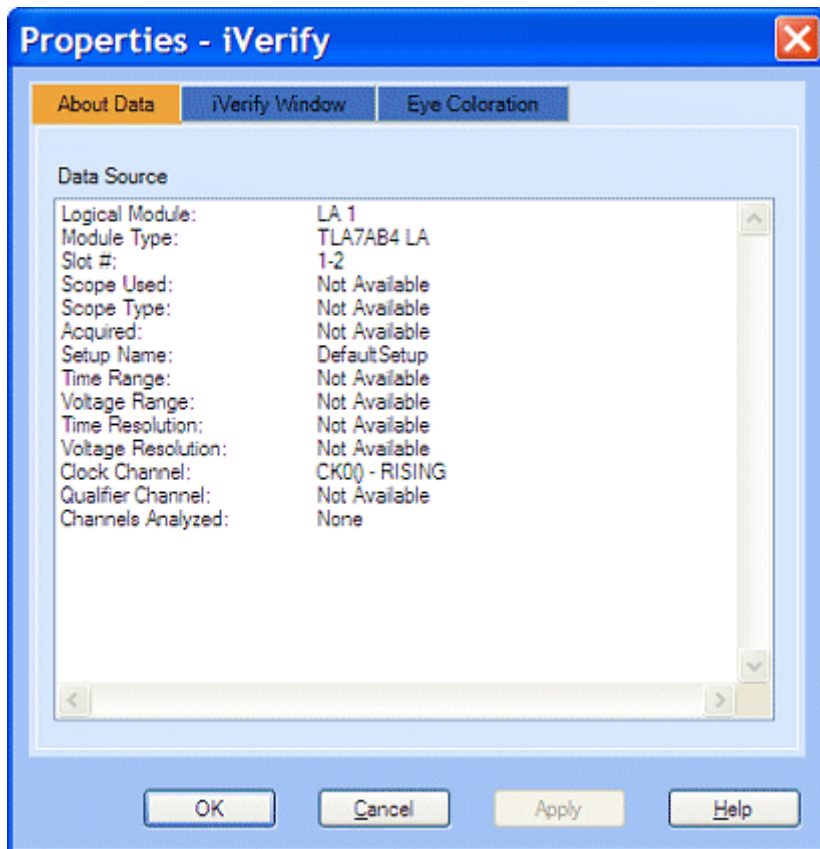
[About Data Tab \(see page 322\)](#)

[iVerify Window Tab \(see page 323\)](#)

[Eye Coloration Tab \(see page 325\)](#)

iVerify Properties - About Data Tab

The About Data tab contains information about both the logic analyzer and the external oscilloscope for the current acquisition. This information includes the version and quantities of logic analyzer modules installed in the mainframe, the type of external oscilloscope, signal information, and other configuration information. Click the tabs in the graphic to navigate to the help contents for each iVerify Properties tab.



Behavior

The About Data tab contents is read-only information.

Related Topics

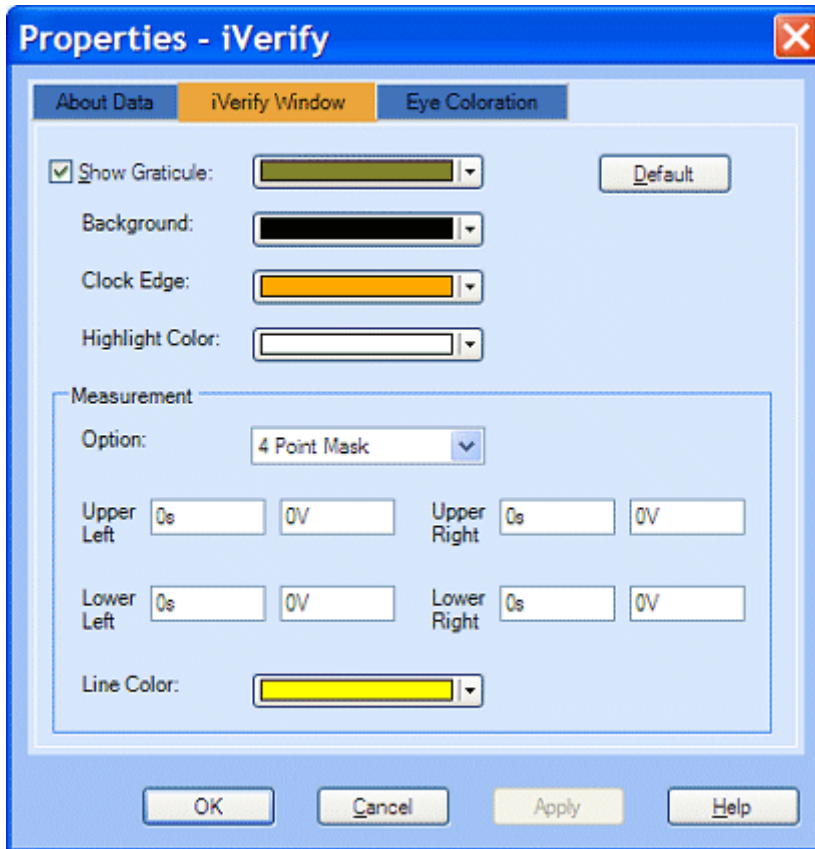
[iVerify Properties Dialog Box \(see page 322\)](#)

[iVerify Window Tab \(see page 323\)](#)

[Eye Coloration Tab \(see page 325\)](#)

iVerify Properties - iVerify Window Tab

The iVerify Window tab allows you to color the measurement drawings that are overlaid with the eye diagram. It also allows you to specify the position and color of the measurement drawings.



The Show Graticule box allows you to indicate whether to display the graticule and specify the graticule color. You can also define the following settings for the eye diagram:

- The Background drop-down list allows you to select the background color of the eye diagram.
- The Clock Edge drop-down list allows you to select the color of the clock edge in the eye diagram.
- The Highlight Color drop-down allows you to select the color for the highlighted signal.
- The Line Color drop-down list allows you to select the color of the measurement lines.

The controls that are available in the Measurement group box depend on your Option selection. The following selections are available:

- 4 Point Mask. The 4 Point Mask option allows you to view or edit the mask coordinate values and the line color for this measurement in the Measurement group box.
- 6 Point Mask. Selecting the 6 Point Mask option allows you to view or edit the mask coordinate values and the line color for this measurement in the Measurement group box.
- Eye Limit Cursor. Selecting the Eye Limit Cursor measurement option allows you to view or edit the time, voltage, and line color for this measurement in the Measurement group box.

- Upper/Lower Mask. Selecting the Upper/Lower Mask measurement option allows you to view or edit the upper and lower mask positions and the line color for this measurement in the Measurement group box.
- Slope. Selecting the Slope measurement option allows you to view or edit point 1, point 2, and line color for this measurement in the Measurement group box.
- Horizontal Histogram. Selecting the Horizontal Histogram measurement option allows you to view or edit the scale, voltage and line color for this measurement in the Measurement group box.
- Vertical Histogram. Selecting the Vertical Histogram measurement option allows you to view or edit the scale, time, and line color for this measurement in the Measurement group box.

Related Topics

[iVerify Properties Dialog Box \(see page 322\)](#)

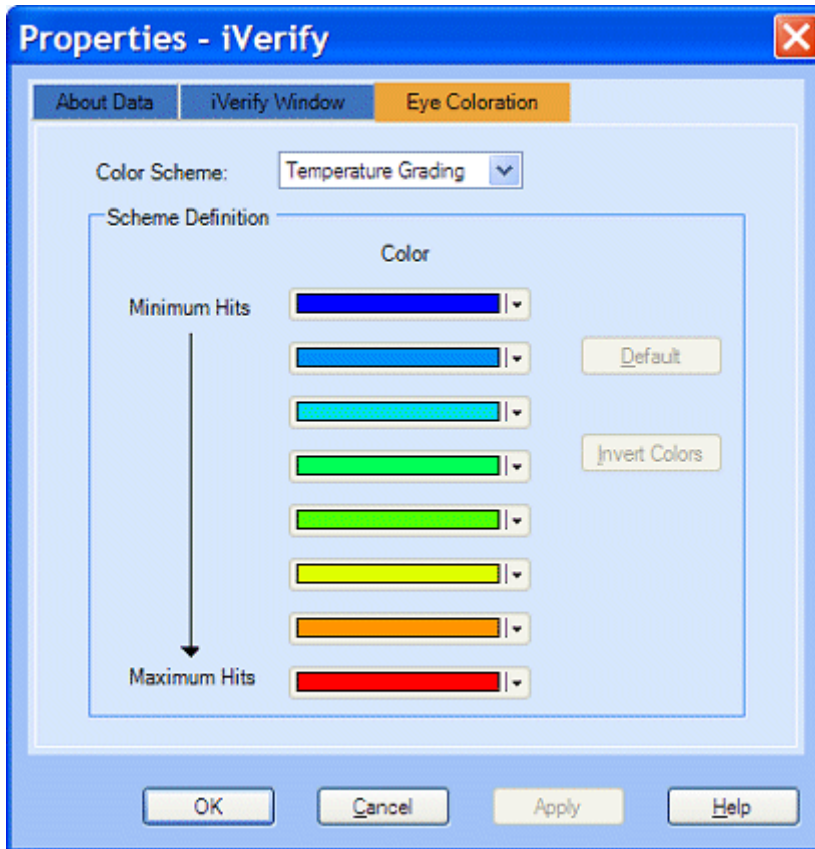
[About Data Tab \(see page 322\)](#)

[Eye Coloration Tab \(see page 325\)](#)

iVerify Properties - Eye Coloration Tab

The Eye Coloration tab allows you to set coloration for the eye diagram. Click the tabs in the graphic to navigate to the help contents for each iVerify Properties tab.

You can select a color scheme that controls the colors of the eye diagram or, if you choose the color setting of User Defined, you can individually set the colors used in the eye diagram.



The Color Scheme drop-down list shows all the color schemes that you can apply to the eye diagram. A description of the color scheme selections follows:

- Temperature Grading. Using Temperature Grading, the areas of the eye with the highest sample density appear in warmer colors (red shades), and the areas of lowest sample density appear in cooler colors (blue shades). Temperature Grading is the default eye diagram color scheme.
- Spectral Grading. Using Spectral Grading, the areas of the eye with the highest sample density appear in blue shades, and the areas of lowest sample density appear in red shades.
- Monochrome Gray. The Monochrome Gray color scheme displays the eye diagram in shades of gray. The areas of the waveform with the highest sample density appear in lighter shades of gray, and the areas of lowest sample density appear in darker shades of gray.
- Monochrome Green. The Monochrome Green color scheme displays the eye diagram in shades of green. The areas of the waveform with the highest sample density appear in lighter shades of green, and the areas of lowest sample density appear in darker shades of green.
- User Defined. Selecting the User Defined color scheme displays the eye diagram in shades of the color that you select from the Scheme Definition group box.

Using the color boxes contained in the Scheme Definition group box, you can individually assign colors that will indicate sample density in the eye diagram. However, this group box is enabled only when you select the User Defined color scheme.

Click the Default button to return the eye diagram coloration to the Temperature Grading color scheme, which is the default color scheme.

Click the Invert Colors button to swap the colors that indicate the highest and lowest sample density.

Related Topics

[iVerify Properties Dialog Box \(see page 322\)](#)

[About Data Tab \(see page 322\)](#)

[iVerify Window Tab \(see page 323\)](#)

iVerify Statistical Analysis Dialog Box

Overview

The iVerify Statistical Analysis dialog box displays information regarding measurements that you performed. This dialog box numerically represents the contents of the measurement data contained in the iVerify window.

You can select the measurements to be performed from the toolbar of the iVerify window or by selecting Data > Measurements from the iVerify menu bar.

Behavior

The iVerify Statistical Analysis dialog box content is read-only information. You can save the statistics to a text file by clicking the Save As button and then saving the file in a desired location.

Related Topics

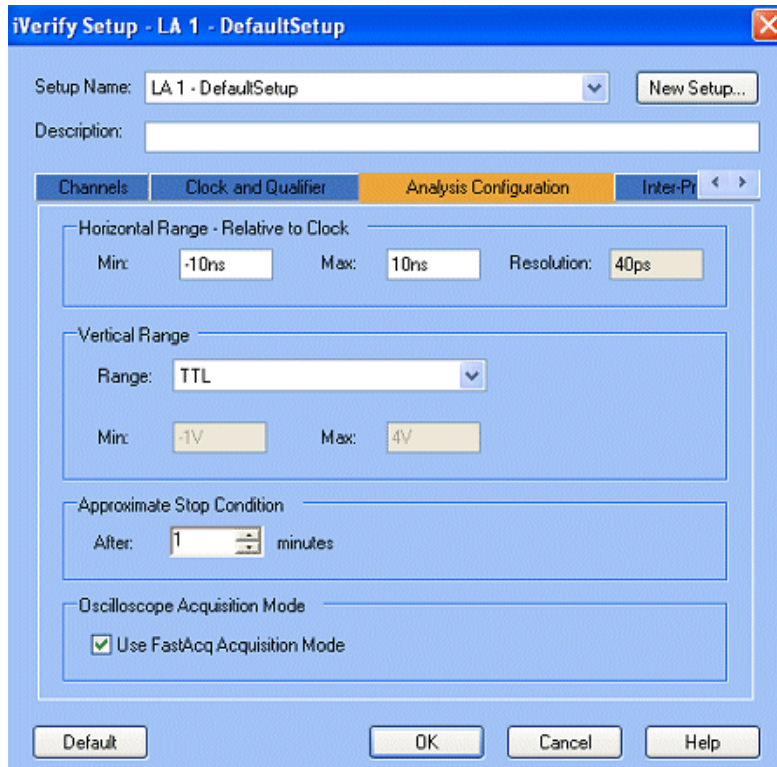
[iVerify Window \(see page 316\)](#)

[iVerify Properties Dialog Box \(see page 322\)](#)

iVerify Setup Dialog Box: Analysis Configuration Tab

Use the Analysis tab to select the necessary settings for the external oscilloscope which will be used for analysis. This tab allows you to set up the time and voltage settings, and time limit for analysis.

The Horizontal Range group box contains Min and Max controls that you set to affect the frequency horizontal scale parameters in the external oscilloscope.



The values that you enter in the Min and Max fields are rounded as necessary to the closest horizontal scale, which is between -40s and 40s.

The Horizontal Range group box contains a read-only Resolution box that internally calculates and displays the horizontal resolution for both the given number of clocks and horizontal scale.

The Vertical Range group box contains controls that affect the voltage range for the oscilloscope signals. The Range box allows you to select from predefined voltage ranges for the external oscilloscope signals. If you select the range setting of User Defined, you can then use the Min and Max controls to specify the minimum and maximum voltage limits for the external oscilloscope. Otherwise, these controls are read-only. The values that you enter in the Min and Max fields are rounded as necessary to the closest vertical scale, which is between 5 V and -2.5 V.

The Approximate Stop Condition group box contains the After control, which allows you to limit the time that the iVerify takes to perform analysis.

The Oscilloscope Acquisition Mode group box allows you to use the FastAcq acquisition mode of the oscilloscope if it is available on the external oscilloscope.

NOTE. You need to provide information in the Channels, Clock and Qualifier, and Analysis Configuration tabs before viewing the Inter-Probing tab. This is because the optimum analog routing information contained in the Inter-Probing tab depends upon the information contained in these tabs. If the setup that you have specified is physically invalid, an error message appears.

Related Topics

[iVerify Setup Dialog Box \(see page 320\)](#)

[iVerify Setup Dialog Box: Channel Tab \(see page 329\)](#)

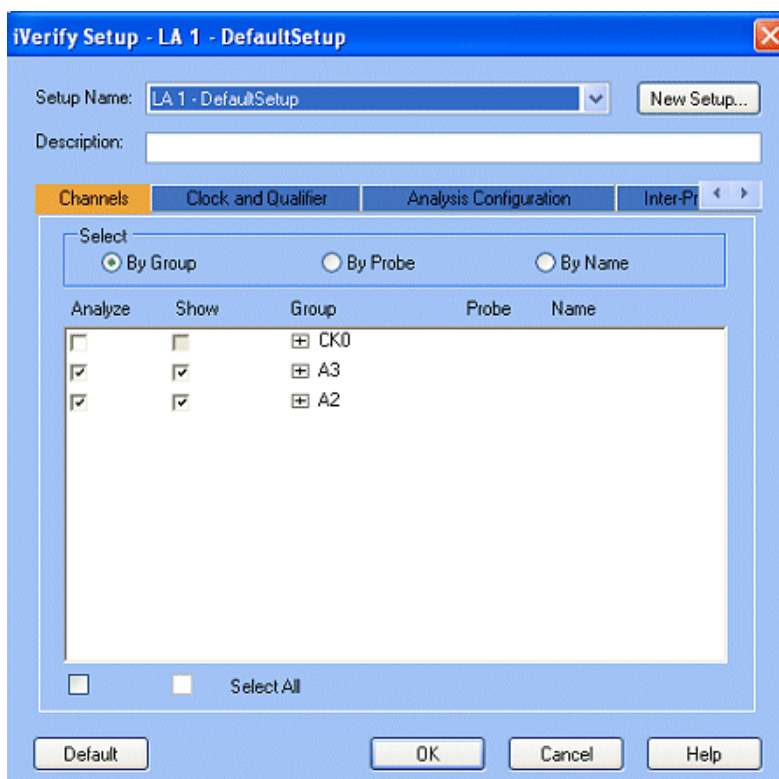
[iVerify Setup Dialog Box: Clock/Qualifier Tab \(see page 330\)](#)

[iVerify Setup Dialog Box: Inter-Probing Tab \(see page 331\)](#)

iVerify Setup Dialog Box: Channel Tab

The Channels tab allows you to select both the signals that you want to analyze and the signal data that will display in the eye diagram, which is located in the iVerify window.

Use the Channels tab to list signals by group, by probe, or by name. Then specify the groups or signals that you want to analyze by selecting the check box associated with each group or signal in the Analyze column.



To specify the groups or signals for which the analysis results will be displayed in the eye diagram, select the check boxes associated with each group or signal in the Apply column. By default, when you select an Analyze check box, the related Show check box is selected.

NOTE. You need to provide information in the Channels, Clock and Qualifier, and Analysis Configuration tabs before viewing the Inter-Probing tab. This is because the optimum analog routing information contained in the Inter-Probing tab depends upon the information contained in these tabs. If the setup that you have specified is physically invalid, an error message displays.

Related Topics

[iVerify Setup Dialog Box \(see page 320\)](#)

[iVerify Setup Dialog Box: Clock/Qualifier Tab \(see page 330\)](#)

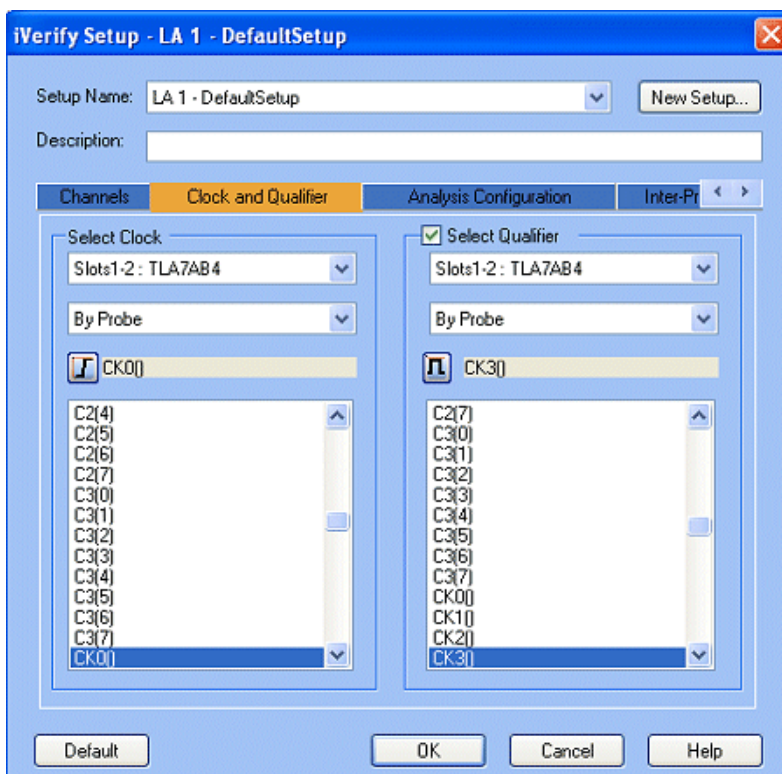
[iVerify Setup Dialog Box: Analysis Configuration Tab \(see page 327\)](#)

[iVerify Setup Dialog Box: Inter-Probing Tab \(see page 331\)](#)

iVerify Setup Dialog Box: Clock/Qualifier Tab

Use the Clock and Qualifier tab to set up the clock signal and qualifier signal and the edge that will be routed to the oscilloscope for analysis. Any of the signals displayed can be selected as a clock signal or qualifier.

The Channels to be Used as Clock group box contains a Select list which you use to arrange the signals that display in the clock list.



If you choose By Probe, the clock list will display all signals regardless of whether they are part of a group that is defined in the LA Setup window.

If you choose By Name, the clock list will display only the signals that have been assigned custom names in the LA Setup window.

Select the reference clock edge by clicking the clock edge icon until it displays your preference: rising edge or falling edge. Then select the signal to be used as a clock that is routed to the oscilloscope for analysis.

Select the Select Qualifier check box when you want to send a qualifier to the oscilloscope along with the clock signal. When this check box is selected, the analysis of signals selected in the iVerify setup occurs in the oscilloscope only if the qualifier is valid in the same timeframe.

The Channels to be Used as Qualifier functions similarly to the Channels to be Used as Clock group box. Select the high or low condition of the qualifier by clicking the qualifier edge icon until it displays your preference: high level or low level. Then select the signal to be used as a qualifier that is routed to the oscilloscope for analysis.

NOTE. *If you have a five-module setup, you cannot select any status bits.*

NOTE. *You need to provide information in the Channels, Clock and Qualifier, and Analysis Configuration tabs before viewing the Inter-Probing tab. This is because the optimum analog routing information contained in the Inter-Probing tab depends upon the information contained in these tabs. If the setup that you have specified is physically invalid, an error message appears.*

[iVerify Setup Dialog Box \(see page 320\)](#)

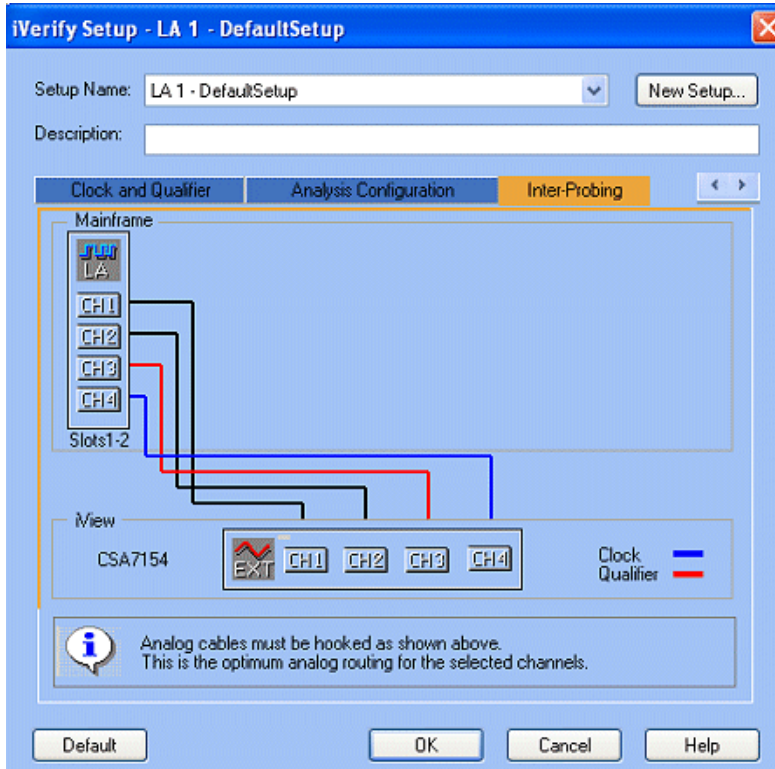
[iVerify Setup Dialog Box: Channel Tab \(see page 329\)](#)

[iVerify Setup Dialog Box: Analysis Configuration Tab \(see page 327\)](#)

[iVerify Setup Dialog Box: Inter-Probing Tab \(see page 331\)](#)

iVerify Setup Dialog Box: Inter-Probing Tab

The Inter-Probing tab displays optimum analog routing information that you make between the logic analyzer and external oscilloscope to achieve analysis results for the selected signals. The logic analyzer modules that are displayed on this tab depend upon the signal selections that you made in the Channels tab of the iVerify Setup dialog box.



Behavior

The Inter-Probing tab is a read-only tab.

NOTE. You need to provide information in the Channels, Clock and Qualifier, and Analysis Configuration tabs before viewing the Inter-Probing tab. This is because the optimum analog routing information contained in the Inter-Probing tab depends upon the information contained in these tabs. If the setup that you have specified is physically invalid, an error message appears.

Related Topics

[iVerify Setup Dialog Box \(see page 320\)](#)

[iVerify Setup Dialog Box: Channel Tab \(see page 329\)](#)

[iVerify Setup Dialog Box: Clock/Qualifier Tab \(see page 330\)](#)

[iVerify Setup Dialog Box: Analysis Configuration Tab \(see page 327\)](#)

Microprocessor and Bus Support Package Basic Operation

Microprocessor and bus support packages are used to disassemble data from microprocessor and bus based systems. For information on a specific support package, refer to the documentation that came with the support package.

The support can be used to perform timing analysis.

To acquire and display timing data, you need to change the sampling selection and trigger program. The support includes a saved system setup file to view data in a Waveform window.

Related Topics

[Sampling \(see page 14\)](#)

[LA module Trigger Window \(see page 143\)](#)

Load Support Package

Loads the selected support package onto the selected logic analyzer module. The destination module must have sufficient signal width for the support package to load.

From the File menu, click Load Support Package.

Group Definitions for Microprocessor and Bus Support

Each support package contains group definitions. Many supports have groups defined for: Address, Data, and Control signals, and additional groups defined for other signals. If you want to know which signal is in what group, refer to the signal assignment tables in the support instruction manual.

Microprocessor and Bus Support Package Triggering

All the Trigger window selections available with your analyzer are also available with the support package. However, if the support package displays exception cycle types, the cycle types are computed and cannot be used to control triggering.

A control group symbol table, which includes special symbols for triggering, is supplied with many support packages.

You can trigger on pattern or range symbol values. However, only the = and Is Not operators in the Clause definition dialog box can use pattern symbol values. All operators can use range symbol values.

Disassembly Display Formats

Four different display formats offer optional selections to modify the way disassembled data is displayed. This makes it easier to analyze data.

The display formats are: [Hardware](#), [Software](#), [Control Flow](#), and [Subroutine](#). Other display options may be available depending on your support package.

To select a disassembly display format:

1. Open a Listing window.
2. From the View menu select Properties.
3. Click the Disassembly tab.
4. Go to the Show box and select the display format.

Saved System Timing Format File

In the Waveform window, every signal shows as a waveform, and groups show as busforms.

The support package includes a saved system setup file that you can use to view data in the Waveform window. To load this file, do the following:

1. Go to the File menu and select Load System.
2. Click Browse.
3. Move through the directory structure as follows:
C:\Program files\TLA700\Supports\(\support package folder)\timing.tla
4. Click Yes or No to indicate whether or not you want to save your current system.

NOTE. A message may appear stating that the configuration of the file does not match your configuration. Click OK and follow the directions in the Load System Options dialog box, or click Help in the dialog box for more information.

5. Click the waveform data icon after it appears in the System window to view the timing file.

Custom Sampling

When your instrument has a support package loaded, a custom sampling tab is available in the LA Setup window. The name of the support package appears on the tab label.

Use Custom sampling to select sampling options associated with the support package. It gives special access to the advanced sampling features of the logic analyzer to track the complex behavior of a digital device or bus protocol. The available selections vary with each support package. Custom sampling can enable and disable a variety of microprocessor-specific clock cycle types (such as DMA cycles). For additional, information see the instructions that accompanied your support package.

Clocking Problems

If you have clocking problems check the following:

- A clock is available. Click Status in the toolbar to open the Status Monitor and verify that a clock is available. If no clock is available, check your connections.
- The system under test is powered on and running. Verify the system is not halted.
- Custom is the sampling selection in the Setup window.
- Connections between the acquisition module and the probe adapter are correct. Check for activity in the Signal Mapping area of the Setup window.

The clock and probe connections between the interface housings and probe adapter are correct (module name, clock, section names, and signal numbers match), are properly oriented, and are fully engaged.

The connections between the interface housings and acquisition module probe cables have matched color labels, matched slot numbers, and are properly keyed.

The connections between the acquisition module probe cables and probe connectors have matched color labels, matched slot numbers, and are properly keyed.

- The configuration of jumpers or switches on the probe adapter is correct. Refer to the support product instruction manual for configuration information.

If the previous suggestions do not fix the problem try the following:

Reload the support package to restore the analyzer to a default state.

If there are loading problems, remove one or more of the protective sockets from the probe adapter. These sockets might add enough additional capacitance or delay to your system under test to affect it.

If the analyzer is still not performing correctly after trying these solutions, there might be a problem with your system under test. Perform timing analysis on your system to ensure the signals are valid when the probe adapter samples them.

Incorrect Data

If the data acquired is obviously incorrect, check that:

- The correct support application is loaded into the system.

- If the microprocessor has an internal instruction cache, you should disable it. The support product instruction manual contains information on when and how to disable the instruction cache.
- Custom Sampling is selected in the Setup menu.
- The connections between the acquisition module and the probe adapter are correct:
- The clock and 17-channel probe connections between the interface housings and probe adapter are correct (module name, clock, section names, and signal numbers match), are properly oriented, and are fully engaged.
- The connections between the interface housings and acquisition module probe cables have matched slot numbers, and are properly keyed.
- The connections between the acquisition module probe cables and probe connectors have matched slot numbers, and are properly keyed.
- The configuration of jumpers or switches on the probe adapter is correct. Refer to the support product instruction manual for configuration information.

If the previous suggestions do not fix the problem, try reloading the support package to restore the analyzer to a known state.

Extended Diagnostics Properties

Use to run extended diagnostics for the following purposes:

- Incoming inspection. Run these diagnostics when first receiving the analyzer.
- Functional procedure. Run these diagnostics after the analyzer or module has been repaired.
- Suspected problem. Run these diagnostics if you think your analyzer is not performing correctly.

NOTE. *Disconnect all probes from the analyzer before running the diagnostics.*

Test Name

This column has an indented hierarchical structure.

All Modules, All Tests – Runs all tests in the list.

TLA700 System – Selects all mainframe tests.

Last Result

This column shows the pass or fail result of the most recent test. When a test is running, the word "Running" displays until the test is complete and results show. When a series of tests is running and an individual test is complete, results are displayed and the next test is executed. The result Unknown indicates that a particular test is in question. Run the diagnostic again. If Unknown appears again, contact your local Tektronix service representative.

Fail Count

This column shows the number of times a test has failed since the RUN button on this page was clicked. This count is mainly used when the run mode is set to Continuous. One Time and Until Fail run modes always display a count of one. All values are cleared when you click the RUN button.

Contact Technical Support for questions regarding diagnostic test results.

To Use

To open the Extended diagnostics tab from the System menu, click Calibration and Diagnostics and then click Extended Diagnostics.

Related Topics

[Running Extended Diagnostics \(see page 133\)](#)

Troubleshooting Triggering

Overview

If the logic analyzer does not trigger, you should check the following:

- Check that your target system is powered on.
- Check the System window to verify that required modules are turned on.
- If the module has correctly sampled data, acquired the specified events, triggered, but has not stopped, the module probably has not acquired enough additional data to fill acquisition memory. Click Stop to manually stop the acquisition, and then change the [Module Memory Depth \(see page 27\)](#) (reduce) or [Trigger Position \(see page 151\)](#) (increase).

LA Triggering

The following conditions apply to the logic analyzer only:

- Check the Status Monitor for a clock warning. If the logic analyzer does not receive a synchronous clock, the Status Monitor displays the following message: External Clock Source Idle.
- Check for signal activity at the probe tip. If there is no activity, check the probe connections.
- Check the clock signal to ensure that the logic analyzer actually receives clock data. Problems with the clock signal can impact setups using synchronous sampling, custom sampling (microprocessor support packages only), or storage qualification.
- Similar to the previous item, check clock qualifier signals and clock equations.
- Check the threshold voltages.
- If the logic analyzer is correctly sampling data and the data events have occurred, but the trigger program did not generate a trigger, check the trigger program itself. The trigger program might not be reaching the state that generates the trigger. Use the Status Monitor to track trigger program progress and identify the state in which trigger progress stops.
- Check the complexity of your trigger program. If your trigger program is too narrowly defined, or over-specified, the trigger program might not acquire the desired data or might not trigger. A less-specific trigger program might acquire the desired data and also verify that you have set up the proper sampling and threshold levels.

Oscilloscope Triggering

The following condition applies to the oscilloscope module only:

- Check the trigger Mode setting. If Mode is set to Normal, and the data does not meet the trigger conditions, the module will not trigger. (Conversely, if Mode is set to Auto, the module will trigger after a set length of time, even if the specified data does not occur.)

Arming or Intermodule Triggering

The following conditions apply only if you are using arming or intermodule triggering:

- Internal signal logic function. If your trigger depends on setting an internal signal, and that signal has been set to Wired-And, all modules in the system must set the signal or it cannot assert.
- If any modules are turned off, check that the trigger program is not waiting for input from an inactive module.

First Transition Indication Problems

- Because the logic analyzer does not clear the first transition indication, check that the modules always show a first transition indication (even if there are no probes attached).
- To avoid triggering on a false first transition indication, set the first state to "If Anything, Go to Next State." This will use a state, but it ensures that you do not trigger on a false transition indication.

Related Topic

[Troubleshooting LA Power Trigger Programs \(see page 104\)](#)

Symbol Files

Overview

Symbol files are files of alphanumeric symbol names and associated data values. Symbol files are used to map a group value to a text string and/or color. For example, a symbol file could map an address group value to a function name. This is faster and easier to interpret than a numeric value.

Behavior

The logic analyzer uses two kinds of symbol files: Pattern Symbol Files and Range symbol files. The range symbol files can contain [Function Symbols \(see page 343\)](#), [Variable Symbols \(see page 344\)](#), [Source Code Symbols \(see page 345\)](#), and [Color Symbols \(see page 354\)](#). The following table shows the types of symbol files and the windows in which they are commonly used.

Window	Pattern symbol files	Range symbol files			
		Functions	Variables	Source code	Color
Listing	Yes	Yes	Yes	Yes	Yes
Waveform	Yes	Yes	Yes	Yes	Yes
Histogram	No	Yes	Yes	No	No
Source	No	No	No	Yes	No
Trigger	Yes	Yes	Yes	Yes	No

Often, the application software will define symbols for you. For example, when you load a microprocessor support package, a symbol file is also loaded (typically, to the control group). These symbols represent data values that correspond to bus cycle types.

Symbol files work like look-up tables. For example, if the address of a printer I/O port is F734BC, you can define a pattern symbol, printer-port, corresponding to that value for use in the Trigger window. Then, in the If/Then Clause dialog box, you can specify the symbol name as the word recognizer value and cause the module to trigger when printer-port (F734BC) appears on the address bus. You can also

specify symbolic display radix for the address group and the symbol name printer-port will be displayed in the Listing window every time F734BC appears on the address bus.

Symbol Value Width

Overview

Symbol values are limited to a width of 32 bits. If your application uses groups wider than 32 bits (for example, in the Trigger window), you could experience some unexpected results when using symbols with those groups.

Behavior

For triggering purposes, if the symbols are wider than 32 bits, zeros are added to the most-significant bits. In pattern symbols, don't cares (Xs) are added to the most significant bits. Refer to the tables below for examples on how the Trigger window handles symbols wider than 32 bits.

Pattern example	Value	Symbol
32-bit example	FFFFAB23	IO-3
40-bit example (32-bit with padding for most-significant bits)	XXFFFFAB23	IO-3

Range example	Lower bound	Upper bound	Symbol
32-bit example	FFFF0055	FFFFAC00	Sym 12
40-bit example (32-bit with padding for most-significant bits)	00FFFF0055	00FFFFAC00	Sym 12

Range Symbol Files

Overview

Range symbols define a range of 32-bit addresses represented by a contiguous set of integers, marked by specific lower and upper bounds. The types of range symbols are: [Function Symbols \(see page 343\)](#), [Variable Symbols \(see page 344\)](#), [Source Code Symbols \(see page 345\)](#) (source), and [Color Symbols \(see page 354\)](#).

Behavior

The various range symbols may overlap each other. When overlaps occur, and the logic analyzer needs to convert a numeric value to a symbol, it may need to choose between the symbols. The logic analyzer

assigns a preference order to symbol types with function symbols having the highest precedence, followed by variable symbols, and then source symbols. Color symbols are looked up separately and may overlap with any of the other types.

Most TLA applications will typically not display source symbols when function symbols are present because of the preference rule (source symbols are covered or overlapped by function symbols). There may be situations when you prefer the source symbols. You can then use the Load Symbols Options dialog box to specify that function symbols should not be read when loading the symbol files.

Pattern Symbol Files

Pattern symbols consist of data patterns of up to 32 bits. Each bit in a pattern symbol can be 0, 1, or X (don't care). Pattern symbols are used when a group of signals define a logical state. Each pattern symbol may also have a color associated with it.

For example, a microprocessor has a set of pins that indicate the type of bus cycle in progress. A memory-read cycle is indicated when the RD~ and MREQ~ pins are 0 (logic low) and the BUSAK~ and M1~ pins are 1 (logic high).

You can define a pattern symbol name called mem-read that corresponds to bit pattern 1100 and mark all memory read bus cycles in the Listing window. For other bus cycles the logic state of these pins is also important and you can define different bit patterns for each cycle type.

Function Range Symbols

Overview

Function range symbols define the beginning and ending addresses where instructions that implement a function (subroutine) are located in memory.

Functions are composed of multiple source statements. The address range of a function symbol can overlap a set of ranges defined by one or more source symbols. A partial range file with functions is shown below:

```
#+ Function
# Symbol Name Low High
# =====
displayBanner 006035ba 00603675
buildMenus 00603676 006036e5
displayLCDmenu 006036e6 0060372f
```

Behavior

The first line tells the file reader that the following symbols represent functions.

The next two lines are comments used as headers.

The symbol name can contain up to 220 ASCII characters. However, the file name may be truncated when displayed. To include embedded spaces in the symbol name, enclose the symbol name in double quotes.

The Low and High columns describe the lower and upper bounds. The bounds are expressed as 32-bit values; the radix is specified in the header. The bounds are inclusive; the specified range includes both bound values, as well as all values between the bounds.

Variable Range Symbols

Overview

Variable range symbols define the beginning and ending addresses where the value of a variable is located in memory. Variable symbols can also be overlapped by either function or source symbols. A partial range file with variables is shown below:

```
#+ variable
# Symbol Name Low High
# =====
menu 00000100 00000102
userMenu1 000004c0 000004c2
binBits 000004d4 000004d6
```

Behavior

The first line tells the file reader that the following symbols represent variables. If this line is not included, the file reader will assume that the symbols represent functions rather than variables.

The next two lines are comments used as headers.

The symbol name can contain up to 220 ASCII characters; however, the file name may be truncated when displayed. To include embedded spaces in the symbol name, enclose the symbol name in double quotes.

The Low and High columns describe the lower and upper bounds. The bounds are expressed as 32-bit values; the radix is specified in the header. The bounds are inclusive; the specified range includes both bound values, as well as all values between the bounds. If the variable is a single byte in memory, the lower and upper bound values are equal.

Source Range Symbols

Overview

Source symbols are similar to function range symbols, except that the address range for a source symbol describes the location of the instructions that implement only one source statement. Source symbols also contain the following information that defines the location of the source code associated with a symbol:

- Source file name
- Source line number
- An optional column range

A partial range file with source symbols is shown below:

```
#+ Source stoplite
# Line Low High Beg End Symbol Name
# =====
27 006043ec 006043ef 0 25 # stoplite_27_25
35 006043f0 006043f5 0 23 # stoplite_35_23
47 006043f6 006043ff 0 30 # stoplite_47_30
48 00604400 00604409 0 30 # stoplite_48_30
49 0060430a 00604413 0 30 # stoplite_49_30
50 00604414 0060441d 0 30 # stoplite_50_30
51 0060441e 00604427 0 30 # stoplite_51_30
```

Behavior

The first line tells the file reader that the following symbols represent source code. The first line also specifies the name of the source file (stoplite). If this line is not included, the file reader will assume that the symbols represent functions rather than variables and source symbols. Each set of symbols for a unique source file must be introduced with an initial source directive to specify the file name for the symbols.

The next two lines are comments used as headers.

The line number specifies the line of the source file containing the source statement. Line numbers are always expressed in decimal.

The Low and High columns describe the lower and upper bounds. The bounds represent the range of addresses occupied by the instructions that implement a single source code statement. The radix of the bound is defined in the file header. The value of the bounds can range from 0x00000000 to 0xFFFFFFFF. The bounds are inclusive; the specified range includes both bound values, as well as all addresses between the bounds. If the source symbol is a single byte in memory, the lower and upper bound values are equal.

The Beg and End columns are optional. When present, they define the beginning and ending column position for the source statement. This makes it possible to define the exact location of the statement on the line.

The symbol name appears as a comment at the end of the line. The file reader ignores the symbol name because it is a comment. When the logic analyzer exports range symbol files, the symbol name is a concatenation of the symbol file name, line number, and column number (if present). This is the symbol value that the logic analyzer displays for addresses that fall within the lower and upper bound for a source file.

Symbols

Symbols simplify tasks, such as setting up triggers or identifying specific values within the data. When you program a trigger or view data, it is cumbersome to remember which numeric group values correspond to particular machine instructions or code modules. The logic analyzer makes this task more manageable by allowing you to assign symbolic names or mnemonics to group values.

For example, assign the symbol WRITE to the control bus event that causes the target system to write to a memory location. Then, if you want to trigger when a write cycle occurs, enter WRITE in the trigger program in place of the actual data value. You can also choose to have WRITE appear in the Listing window for quick identification of the instruction.

Symbols Dialog Box

Overview

Use the Symbols dialog box to provide an overview of all currently loaded symbol files. With this dialog box you can:

- Display symbol status and content information of currently loaded symbol files
- Load symbol files
- Reload symbol files
- Unload symbol files
- Export symbol files

Status Information

The following status information is available for each currently loaded symbol file:

- Last Loaded - Describes the last time the file was loaded into the TLA application software. It also includes error and warning messages associated with the last load.

- File Modified - Displays the time the file was last modified. It also displays a message if the file might need to be reloaded (such as when the file is modified after it was first loaded).
- Format Loaded - Displays the format of the loaded file.
- Can Be Unloaded - Describes if the file can be unloaded or if the file is currently in use. Symbol files can only be unloaded if no data windows or setups are using them.

Content Information

The following content information is displayed:

- Symbol Type - Shows whether the file is a pattern symbol file or a range symbol file. If the file is a range file, this field also lists the types of symbols loaded.
- Total Symbols - Shows the number of symbols loaded from the file.
- Total Source Files - Lists the number of source files referenced by source code symbols loaded from the file.
- Range - Lists the minimum and maximum address values and offset information.

Behavior

Click Load to load a new symbol file. You do not need to load all symbol files with the Symbols dialog box. Symbol files can be loaded through other windows. You can also display information on all symbol files currently used by the logic analyzer by selecting any loaded symbol file.

NOTE. *When you load a symbol file, the symbols are placed in memory. Because symbol files consume memory, you should unload unused symbol files to keep memory available for your main application.*

Click Unload to unload a symbol file from the application software.

NOTE. *The Unload button is inactive when the loaded symbol file is in use.*

Click Export to save the current file as a TSF-format symbol file. You can view the exported symbol file with applications such as Wordpad. You can then edit the symbol file by saving it under a new name and using a text editor, such as Wordpad, to make any changes. Save the edited file in ASCII format. You can then load the edited symbol file.

To Use

To open the Symbols dialog box from the System menu, click Symbols.

Related Topic

[Load Symbols Options Dialog Box \(see page 348\)](#)

Load Symbols Options Dialog Box

Overview

Use the Load Symbols Options dialog box to specify options for range symbol files before loading them into the system. You can specify any of the following symbol file formats (check with your Tektronix representative for a complete listing of available formats): [OMF51](#), [OMF86](#), [OMF166](#), [OMF286/386](#), [COFF](#), [IEEE695](#), and [ELF/DWARF or ELF/STAB](#).

In addition to the above formats, you can also use the [Tektronix Symbol file Format \(see page 350\)](#) if your code generation tools do not operate on any of the listed formats.

Behavior

Select one or more of the symbol types to load. If you want to use the symbol file with the Source window, be sure to include the Source Code as the symbol type.

The Bound 1 and Bound 2 fields define the range of symbol addresses that will be loaded. You can enter any hexadecimal values from 0 through FFFFFFFF. Values outside of these limits are ignored.

NOTE. *If the lower bound of an overlapping range symbol is within the Bound 1 and Bound 2 limits and the higher bound is not, the entire symbol will be valid. However, if the higher bound is within the Bound 1 and Bound 2 limits and the lower bound is not, the entire symbol will be ignored.*

Select either Default Offset or Custom Offset to apply an offset to the symbol values. If you select Default Offset, the default offset is read from the source file and applied to each symbol in the file. If you select Custom Offset, you can add or subtract the specified offset value to each symbol in the file as it is loaded. You can choose any 32-bit hexadecimal value from 0 to FFFFFFFF.

When you enter bound values, enter the values without an offset value. If your application adds an offset, subtract the offset value before you enter the bound values.

To Use

To open the Load Symbols Options dialog box from the System menu, click Symbols and then click the Load button. Then select a symbol file and click Options.

Related Topic

[Symbol Files \(see page 341\)](#)

OMF51 File Format

This format holds symbolic information and executable images for an 8051 or equivalent microprocessor.

OMF86 File Format

This format holds symbolic information and executable images for an 8086 or equivalent microprocessor. It is also used for code intended to run on 80286, 80386, or higher-level microprocessors in real or 8086-compatible mode.

OMF166 File Format

This format holds symbolic information and executable images for the Siemens (Infinion) C166 or microprocessor family, or compatible microprocessor.

OMF80286 and OMF80386 File Formats

These formats hold symbolic information and executable images for 80286, 80386, or equivalent microprocessors. They are also used for executable images intended to run on the 8086 or other microprocessors in the 80x86 families.

COFF File Format

The Common Object File Format (COFF) contains a number of variations and extensions, such as ECOFF, XCOFF, and COFF2.

This flexibility enables the file format to be used with a wide variety of different microprocessors. Some code-generation tool vendors also extend this format in nonstandard ways that may make their files unreadable by Tektronix Logic Analyzers.

IEEE695 File Format

This object file format refers to the IEEE695 specification. This format is used primarily by compilers for a wide variety of Motorola microprocessors, as well as Motorola-compatible microprocessors from other vendors. This format provides for the inclusion of column information in source symbols, but not all compilers use this capability.

ELF/DWARF and ELF/STABS File Formats

Elf is an object file format used primarily with PowerPC microprocessors. The Dwarf 1, Dwarf 2 and Stabs names refer to the format of the portion of an Elf file that contains debug information. This format provides for the inclusion of column information in source symbols, but not all compilers use this capability.

NOTE. *Although Dwarf 1 and Dwarf 2 debug formats are able to support symbol values in excess of 32 bit resolution, the Logic Analyzer will use only the least significant 32 bits. The logic analyzer will issue a warning if any symbols exceeding 32 bits in size are encountered while loading the symbol file.*

Exporting Symbol Tables

Use this export dialog box to export symbol tables in the TSF format. The TSF format is ASCII, which makes it more suitable for display and modification. The exported file can also be reloaded if desired.

Related Topic

[Symbols Dialog Box \(see page 346\)](#)

TLA Symbol File Format Overview

The logic analyzer can extract range symbol information directly from object files.

Users whose code generation tools do not generate some file formats can use TLA Symbol File (TSF) format (a text format). The logic analyzer uses the TSF format when it exports symbol files. The logic analyzer can also read files in this format.

TLA symbol files consist of alphanumeric symbol names and associated data values. The files contain a header line and lines defining the symbol names and values. Fields on a line are separated with white space (blank spaces or tabs). The formats for the pattern symbol files and the various range symbol files differ; if you need to use pattern symbols and range symbols, use separate files.

Both TLA range and pattern symbol files have a .tsf file name extension (filename.tsf).

The first few lines of a TSF symbol file are typically comment lines describing when and how the file was generated.

```
# TLA Symbol File
```

```
# Created on Friday, May 29, 1998 at 09:52:03
```

```
# From file: "c:\quickstart\tla7qs.x"
```

All lines in a TSF file that begin with a number sign (#) character are treated as comments, unless the very next character is a plus sign (+). The plus sign signifies a file reader directive. The number sign character

can also initiate a comment on the end of a symbol definition or other uncommented lines. All text between the number sign and the end of the line is treated as a comment and ignored.

Related Topics

[TLA Symbol File Format - TSF Headers \(see page 351\)](#)

[TLA Symbol File Format - TSF Pattern Symbols \(see page 352\)](#)

[TLA Symbol File Format - TSF Range Symbols \(see page 353\)](#)

TLA Symbol File Format - TSF Headers

A TSF header identifies the file format version to potential file readers. It specifies whether the file contains pattern or range symbols, the radix used to specify symbol values, and an optional offset amount to be added to each symbol value (for range symbol files).

The TSF Header is a file directive, which means that the following information begins with the special character combination of a number sign character followed immediately by a plus sign (#+). This is not a comment line. The special character sequence is used to mark instructions to the file reader; these instructions are called file directives.

The following examples show sample file headers for a pattern symbol file and a range symbol file. The first two lines are comments included for readability; they are not required as part of the file header.

```
# TSF Format Type Display Radix File Radix
# =====
#+ Version 2.1.0 PATTERN HEX HEX
# TSF Format Type Display Radix File Radix Offset
# =====
#+ Version 2.1.0 RANGE HEX HEX 00000000
```

The File Format Version number contains three fields. The first two fields are the format major and minor version numbers. The logic analyzer will only read TSF files where the major and minor version number of the file is less than or equal to that of the TLA TSF symbol file reader. The third field denotes minor format changes, which do not impact the file reader.

The File Format Version number is followed by a key word, PATTERN or RANGE, which signifies the type of symbols found in the file. TSF files can contain either type of symbols, but no single file can contain both. The header specifies the type for all symbols in the file.

The Display Radix field sets the default radix that will be used to display the numeric symbol value. For range symbol files, this field must be one of the key words: HEX, DEC, OCT, or BIN. For pattern symbol files, only the key words HEX, OCT, or BIN are allowed.

The File Radix field specifies the radix used by the symbol values in this file. Like the Display Radix field, the File Radix field must be one of the key words: HEX, DEC, OCT, or BIN for range symbol files, or HEX, OCT, or BIN for pattern symbol files.

The Offset field specifies an offset value and is applicable only to range symbol files. The offset value is specified in the radix indicated by the File Radix field. This offset value will be added to the lower and upper bounds of each range symbol that is read from the file. The offset is a 32-bit value, so it can have any value between 0x00000000 and 0xFFFFFFFF. If the sum of the offset and a range symbol bound value exceeds the 32-bit limit of 0xFFFFFFFF, the overflow bit is discarded. Negative offsets are specified by using a twos complement value for the offset.

Related Topics

[TLA Symbol File Format Overview \(see page 350\)](#)

[TLA Symbol File Format - TSF Pattern Symbols \(see page 352\)](#)

[TLA Symbol File Format - TSF Range Symbols \(see page 353\)](#)

TLA Symbol File Format - TSF Pattern Symbols

Each pattern symbol in a TSF pattern file consists of two fields. The first field is the symbol name, and the second is the symbol pattern. The symbol name is a sequence of ASCII characters of up to 220 characters in length, although it is impractical to display symbol values much longer than 32 characters. Symbol names longer than 220 characters will be truncated during loading. The characters in a symbol name can be any character with an ASCII value between 0x21 (the exclamation point character, !) and 0x7E (the tilde character, ~). You can use symbol names with embedded spaces by enclosing the Symbol Name in double quotes.

```
# Symbol Symbol Optional Foreground
# Name Pattern and Background Color
# =====
NUL X0000000
SOH X0000001 @red @yellow
STX X0000010
"ETX 0x03" X0000011
```

The symbol pattern consists of numerals in the radix specified by the File Radix field in the header, and Xs signifying don't-care values. The number of bits represented by each character position in the pattern depends on the selected radix. For a HEX radix, each character represents 4 bits. For an OCT radix, each character represents 3 bits; for a BIN radix, each character represents one bit.

Symbol order in a TSF pattern file is important. When selecting the symbol to display for a particular value, the logic analyzer scans the list of pattern symbols from top to bottom. It selects the first symbol for which all non-don't-care bits of the symbol match the corresponding bits of the target symbol.

Related Topics

[TLA Symbol File Format Overview \(see page 350\)](#)

[TLA Symbol File Format - TSF Headers \(see page 351\)](#)

[TLA Symbol File Format - TSF Range Symbols \(see page 353\)](#)

TLA Symbol File Format - TSF Range Symbols

There are four different types of range symbols:

- [Function \(see page 356\)](#)
- [Variable \(see page 359\)](#)
- [Source \(see page 356\)](#)
- [Color \(see page 354\)](#)

Each of these types define a range of 32-bit addresses associated with some entity.

Function range symbols define the beginning and ending addresses where instructions that implement a function are located in memory.

Variable range symbols define the beginning and ending addresses where the value of a variable is located in memory.

Source range symbols are similar to function range symbols, except that the address range for a source symbol describes the location of the instructions that implement just one source statement. (Source symbols also contain file name, line number, and an optional column range that define the location of the source code associated with the symbol.)

Color range symbols define a display color for any value that falls within a range.

Each of the types of range symbols appears in a separate section of the file. Each section begins with a file directive indicating the type of symbols that follow. The sections may appear in any order, and can be broken up and separated by other sections if necessary.

The first section might be variables, followed by functions, then followed by another variable section. Each section is also optional, however some logic analyzer applications are will not use the symbol file if the appropriate type of symbols are not present. For example, the source window will only correlate with a listing window if the symbol file contains source symbols.

Range symbol names follow the same rules as pattern symbol names. The symbol name is a sequence of ASCII characters of up to 220 characters in length, even though it is impractical to display symbol values much longer than 32 characters.

Symbol names longer than 220 characters will be truncated during loading. The characters in a symbol name can be any character with an ASCII value between 0x21 (the exclamation point character, !) and 0x7E (the tilde character, ~). You can use symbol names with embedded spaces by enclosing the Symbol Name in double quotes.

Range symbol address ranges can overlap. This overlap sometimes produces unexpected results. When overlaps occur, and the logic analyzer needs to convert a numeric value to a symbol, it must choose between the overlapping symbols. When choosing between overlapping symbols, the logic analyzer assigns a precedence order to the symbol types. Function symbols have the highest precedence, followed by variable symbols, and then source symbols.

Related Topics

[TLA Symbol File Format Overview \(see page 350\)](#)

[TLA Symbol File Format - TSF Headers \(see page 351\)](#)

[TLA Symbol File Format - TSF Pattern Symbols \(see page 352\)](#)

TSF Range Symbols - Color

Color range symbols define the beginning and ending group values where a color is displayed. Any group value in the acquisition falling within this range will be shown in the defined color. The value range of a color symbol can overlap with the set of ranges defined by one or more Function, Variable, or Source symbols. In many instances, a Color symbol will use exactly the same range bounds as a Function or Variable symbol.

A partial range file with colors is shown below:

```
#+ color
# color Low High
# =====
@magenta 006035ba 00603640
@yellow @navy 00603541 00603675
@default @green 006036e6 0060372f
```

The first line tells the file reader that the following symbols represent colors. The next two lines are comments used as headers. The first color name specifies the foreground color, and the second optional color name specifies the background color. The available color names (keywords) are:

@black, @blue, @cyan, @lime, @magenta, @red, @yellow, @white, @navy, @teal, @green, @purple, @maroon, @olive, @gray, and @silver.

The special color name @default gives the default text coloring specified by choosing a color in the Column tab of the Listing Window Property Sheet or the Waveform tab of the Waveform Window Property Sheet.

The Low and High columns describe the lower and upper bounds. The bounds are expressed as 32-bit values; the radix is specified in the header. The bounds are inclusive; the specified range includes both bound values, as well as all values between the bounds.

There is a different TSF symbol file syntax for adding color to range and pattern symbol files. For pattern symbol files, color is an attribute of existing pattern symbols. It is placed on the same line as the rest of the

symbol, and to the right of the pattern definition. For range symbol files, color symbols are added in a separate section of the symbol file, just like Function, Variable, and Source symbols. This allows the color information to be independent of the other defined range symbols.

A partial TSF Pattern Symbols file is shown below:

```
# Symbol Name Pattern Color
# =====
NUL X000 0000
SOH X000 0001 @blue
STX X000 0010 @white @red
SBZ X000 0100 @default @teal
```

Each Pattern symbol in a TSF Pattern file consists of three fields. The first field is the Symbol Name, the second is the Symbol Pattern, and the third is Symbol Color.

The Symbol Color specification may contain zero, one, or two color keywords. If only one color is given, it specifies the foreground color of the data. If two colors are given, they specify the foreground color followed by the background color.

To supply a background color, provide a foreground color. If you want to use the group's defined color from the Column or Waveform Property Page, you can use @default as the foreground color.

In addition to specifying the colors by name, you can also specify the color symbols using the RGB hexadecimal format. For example the earlier example of the partial range file with colors is shown below using the RGB format:

```
#+ Color
# Color Low High
# =====
@FF00FF 006035ba 00603640
@FFFF00 @000080 00603541 00603675
@default @008000 006036e6 0060372f
```

Similarly the partial TSF Pattern Symbols file using RGB is shown below:

```
# Symbol Name Pattern Color
# =====
NUL X000 0000
SOH X000 0001 @0000FF
STX X000 0010 @ FFFFFFFF @FF0000
SBZ X000 0100 @default @008080
```

Related Topics[TLA Symbol File Format - TSF Range Symbols \(see page 353\)](#)[TSF Range Symbols - Function \(see page 356\)](#)[TSF Range Symbols - Variable \(see page 359\)](#)[TSF Range Symbols - Color \(see page 354\)](#)

TSF Range Symbols - Function

The file directive "#+ Function" introduces the function symbol section of a TSF Range file. The file directive tells the file reader that the following symbols represent functions, as opposed to variables or source statements. If no symbol type file directive is given, the function symbol type is assumed.

```
#+ Function
```

```
# Symbol Name Low High
```

```
# =====
```

```
displayBanner 006035ba 00603675
```

```
buildMenus 00603676 006036e5
```

```
displayLCDmenu 006036e6 0060372f
```

Function symbols consist of three fields: the symbol name, the lower bound, and the upper bound.

The lower and upper bound values are each 32-bit values, defined by numerals in the radix specified by the File Radix field in the file header. These values define the lower and upper limits of the range of addresses occupied by the instructions that implement a function. Both values are inclusive, which means that the specified range includes both of the bound values, as well as all of the addresses between them.

Related Topics[TLA Symbol File Format - TSF Range Symbols \(see page 353\)](#)[TSF Range Symbols - Variable \(see page 359\)](#)[TSF Range Symbols - Source \(see page 356\)](#)[TSF Range Symbols - Color \(see page 354\)](#)

TSF Range Symbols - Source

The file directive "#+ Source" introduces the source symbol section of a TSF range file. The file directive tells the file reader that the following symbols represent source statements, as opposed to functions or variables. The source file directive must be followed by a file name, which specifies the name of the source

file containing the following source statements. Each new set of symbols for a unique source file must be introduced with an additional source file directive to specify the file name for those symbols. If no symbol type file directive is given, the function symbol type is assumed.

```
#+ Source stoplite
# Line Low High Beg End Symbol Name
# =====
27 006043ec 006043ef 0 25 # stoplite_27_25
35 006043f0 006043f5 0 23 # stoplite_35_23
47 006043f6 006043ff 0 30 # stoplite_47_30
48 00604400 00604409 0 30 # stoplite_48_30
49 0060440a 00604413 0 30 # stoplite_49_30
50 00604414 0060441d 0 30 # stoplite_50_30
51 0060441e 00604427 0 30 # stoplite_51_30
52 00604428 00604431 0 30 # stoplite_52_30
56 00604432 00604437 0 17 # stoplite_56_17
59 00604438 00604439 0 18 # stoplite_59_18
60 0060443a 00604445 0 37 # stoplite_60_37
61 00604446 0060444d 0 33 # stoplite_61_33
59 0060444e 0060444f 35 39 # stoplite_59_35
59 00604450 00604455 19 34 # stoplite_59_19
71 00604456 00604457 0 37 # stoplite_71_37
74 00604458 0060445f 0 35 # stoplite_74_35
77 00604460 00604467 0 36 # stoplite_77_36
80 00604468 0060446f 0 36 # stoplite_80_36
83 00604470 0060447b 0 43 # stoplite_83_43
87 0060447c 00604483 0 34 # stoplite_87_34
71 00604484 0060448d 0 37 # stoplite_71_37
84 0060448e 00604490 0 29 # stoplite_84_29
```

The source symbols section consists of five fields for each source statement: line number, lower address bound, upper address bound, beginning column value, and ending column value.

Source symbols do not have a name in the same sense as function or variable symbols, because there is no name associated with each of the executable statements in a source file. Instead, a source symbol has a file name, specified in the source directive, and a line number. The line number specifies the line

of the source file that contains the source statement. Line numbers are always in decimal, regardless of the file radix in the header.

The lower and upper bound values for a source symbol are similar to those of function and variable symbols. For source symbols, these bounds represent the range of addresses occupied by the instructions that implement a single source statement.

The bound values are defined in the file radix specified in the header and can range in value from 0x00000000 to 0xFFFFFFFF. The lower and upper bound addresses are inclusive, just like other symbols. A source symbol for an instruction occupying a single memory location has matching lower and upper bound values.

The beginning and ending column fields of a source statement are optional. When present, and non-zero, they define the beginning and ending column position for the source statement.

The beginning and ending column values define the location of the statement in the line. This is especially useful when there are multiple statements on a single line, because it makes it possible to define a separate symbol for each statement. When the column information is not present, or is set to zero, the symbol is assumed to correspond to the entire line.

Only a few compilers generate column information, but when the information is present in the symbol file, the logic analyzer uses the column information to provide highly accurate source code correlation. The example source symbols on the previous page show typical column values.

In most cases only one of the column values is non-zero. This is because those lines contain only one statement, and the compiler specified only the column at which the statement ended. Some compilers specify only the beginning column position for such lines, in which case the beginning value would be non-zero and the ending value zero.

Note that there are three symbols corresponding to line 59 of the file. Line 59 of the original source file contains the following statement:

```
for (i = 0; i < NUM_STATES ; i++)
```

This one line contains three separate statements. The first statement is the initialization ($i = 0$), the second statement is the test ($i < \text{NUM_STATES}$), and the third statement is the increment ($i++$).

Although all three statements appear on the same line, each generates a separate set of instructions, and the symbols in the example define unique address ranges for each. This enables the Source window to accurately indicate which of the three statements on the line is associated with any given address.

Each of the source symbols in the example includes a comment at the end of the line showing a symbol name. Since this is a comment, it is ignored by the symbol file reader, and is optional. When symbol files are exported by the logic analyzer, they contain comments that show the derived symbol name created by the logic analyzer itself. The name is a concatenation of the symbol file name, line number, and column number (if present). This is the symbol value that the logic analyzer will display for addresses that fall within the lower and upper bounds of a source symbol.

Related Topics

[TLA Symbol File Format - TSF Range Symbols \(see page 353\)](#)

[TSF Range Symbols - Function \(see page 356\)](#)

[TLA Symbol File Format - TSF Range Symbols \(see page 353\)](#)

[TSF Range Symbols - Color \(see page 354\)](#)

TSF Range Symbols - Variable

The file directive "#+ Variable" introduces the variable symbol section of a TSF Range file. The file directive tells the file reader that the following symbols represent variables, as opposed to functions or source statements. If no symbol type file directive is given, the function symbol type is assumed.

```
#+ variable
# Symbol Name Low High
# =====
menu 00000100 00000102
userMenu1 000004c0 000004c2
binBits 000004d4 000004d6
```

Variable symbols consist of three fields: the symbol name, the lower bound, and the upper bound. Variable symbols are just like function symbols except that they define the address range occupied by a variable instead of function. The lower and upper bound values of a variable symbol are inclusive, meaning that they are themselves a part of the range. For variables occupying only a single byte in memory, the lower and upper bound values are equal.

Related Topics

[TLA Symbol File Format - TSF Range Symbols \(see page 353\)](#)

[TSF Range Symbols - Function \(see page 356\)](#)

[TSF Range Symbols - Source \(see page 356\)](#)

[TSF Range Symbols - Color \(see page 354\)](#)

System Window

From the System window you can access all other setup windows, trigger windows, and data windows. The System window also shows the analyzer configuration, arming, and triggering relationships.

Logic Analyzers

An icon represents each installed instrument. Inside each icon are specific controls where you can do the following tasks:

- Open the properties tab by double-clicking an open area inside the icon.
- Open the logic analyzer Setup window by clicking the Setup button.
- Open the logic analyzer Trigger window by clicking the Trigger button.
- Turn the logic analyzer on or off by clicking the On/Off button.
- Name the logic analyzer by entering text in the text box.

Click the logic analyzer to see which data windows contain data for the logic analyzer.



Active Logic Analyzer

The logic analyzer highlighted by the pointer in the System window is the active logic analyzer.

Identifying Modules


If you are unsure which physical module an icon represents, double-click the icon to open its System Properties dialog box. The System Property dialog box lists information about the module, including the mainframe slot number in which the module is installed.

Disabling Modules

To disable a module, click the module's On/Off button.

When you disable a module, first verify that no other trigger program is dependent on the module's output.

Data Windows

An icon represents each data window. You can use the New Data Window dialog box to create Histogram or Source windows or other windows. To create a new data window click  in the toolbar.

NOTE. You can change the default data window for logic analyzer in the Defaults property page. From the System menu, select Options and click the Defaults tab to display the Defaults tab.

Inside each data window icon are specific controls where you can do the following:

- Open a data window by clicking its icon located above the text box.
- Open the Properties dialog box to access the product's properties tabs by double-clicking the mouse in an open area close to the icon.
- See which data windows are associated with a logic analyzer (or data window), by clicking the logic analyzer (or data window text box). White arrows appear, linking the logic analyzers and data windows.

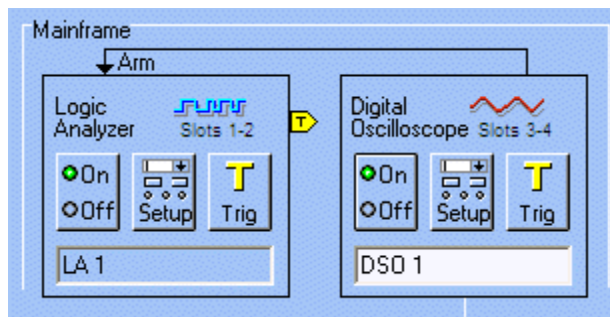


Renaming a Window

You can rename the window by selecting the current window label and typing over it. Window names must be unique and the window name is limited to the space available.

Triggering and Arming Relationships

A logic analyzer that is programmed to cause a [system trigger](#) has a trigger indicator symbol on the right edge of the logic analyzer icon. A black arrow from one logic analyzer to another indicates one module is programmed to [arm](#) another through the first module's [trigger program](#). The example shows a logic analyzer module arming the DSO module.



Context Menu

You can right-click the mouse in different parts of the System window to display a menu that contains a list of useful tasks such as opening or displaying properties for the trigger, setup, and data windows.

Related Topics

[Triggering other Modules from an LA \(see page 103\)](#)

[Arming other Modules from an LA \(see page 103\)](#)

Armed Module

An armed module is a module that begins running its trigger program once it has received an arm signal from another module. This allows the armed module to run its trigger program when a condition is met in the other module's trigger program.

System Trigger (Trigger All Modules)

The system trigger is an overriding command that causes all modules to acquire their post trigger data, regardless of whether they have been armed or have fulfilled their own trigger conditions.

The system trigger is the primary data acquisition reference point. In data windows, timing and location information is relative to the system trigger.

When the system trigger occurs, the trigger signal will appear at the external System Trigger Out connectors of the instrument.

Trigger Program

The trigger program is a series of conditions, similar to software code that defines the data you want to capture and view. The trigger program also specifies actions that are expected when conditions (data events) are true.

The trigger program filters acquired data to find one or more specific data events. The trigger program can accept signals from other modules or signals from external test equipment through the external System Trigger In or the External Signal In connectors. Similarly, the trigger program can also send signals from the module to external equipment through the System Trigger Out and the External Signal Out connectors.

New Listing/Waveform Data window

Use this dialog to create a new Listing or Waveform window. Click one of the following selections to specify the data source for your new data window:

- Click **Data from** to select a data source from the current system. If the data source you want is not listed, click the Add Data Source button to add one from the TLA system or logic analyzer file to the current system. You can also click the Browse button to search for a desired TLA system file.
- Click **No data** to create an empty Listing or Waveform window. After the window displays, you can add columns or waveforms from different sources.
- Click **Same columns/windows as active data window** to add the columns or waveforms from the active window to your new data window.
- Click **MagniVu columns/waveforms based on the active data window** to view MagniVu data (higher resolution) of the same columns or waveforms as the active window.

Related Topic

[Adding a Waveform \(see page 118\)](#)

New Data Window Name

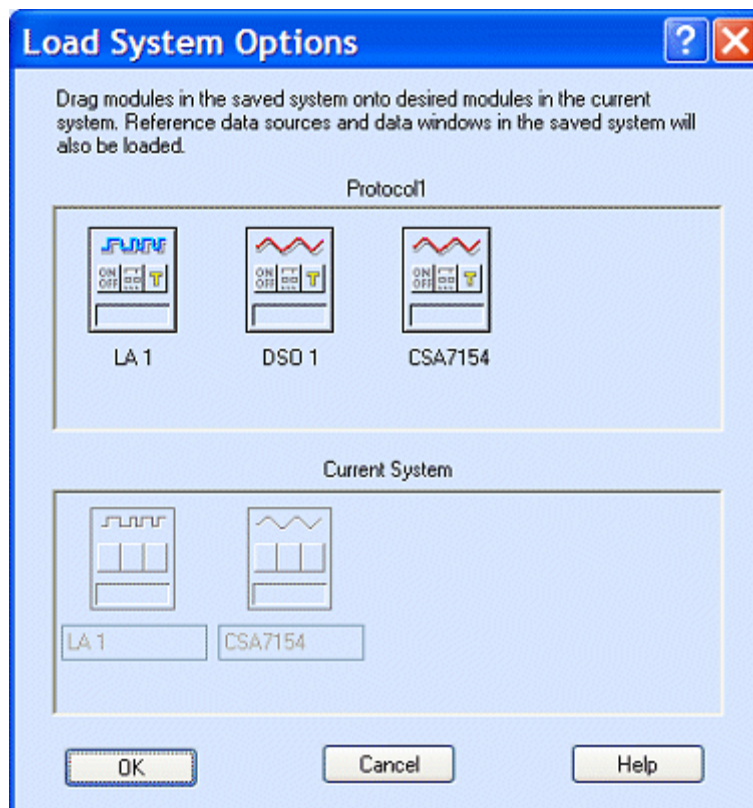
Enter a name for the new data window or use the default name. Then click the Finish button to create the new data window.

Name Data Window Dialog Box

Use to specify a new name for a data window. This dialog box displays when you attempt to name a data window that is not unique.

Load System Options Dialog Box

Use to load part of a saved system file into the current system. This is useful when you need to use portions of a saved system file with the current system. A saved system consists of one or more modules, data windows, and links to data sources in other TLA files. A saved module consists of the module's setup and trigger settings as well as data.



From a saved module file, you can load any of the following:

- An LA module trigger program
- A module setup and trigger program
- Saved data from the module

From a saved system file, you can load any of the following:

- Any previously listed module choices
- Full system setup, including data windows
- Saved data from one or more modules

Behavior

The Load System dialog box is automatically opened when you perform a Load System and the saved system file does not match the current logic analyzer configuration (and both systems contain at least one module).

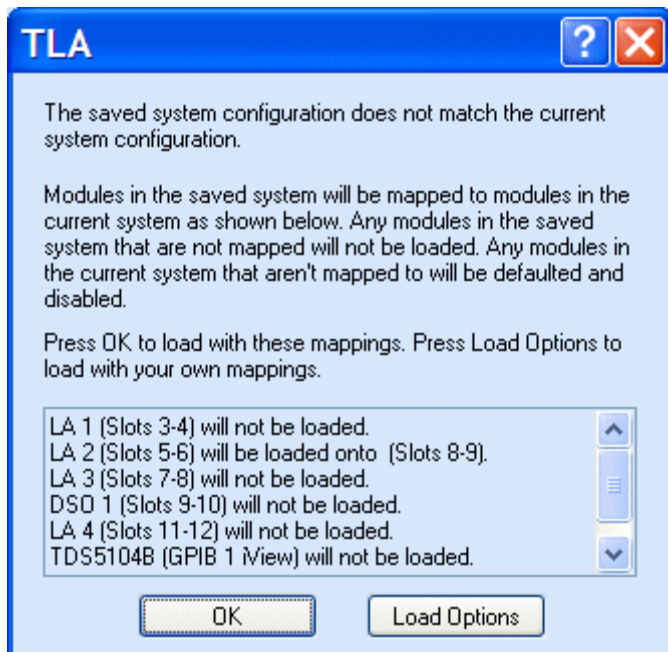
When necessary, the analyzer adjusts the following setup parameters to match hardware limits:

- Memory depth
- Trigger position. Relative position is maintained
- Sample period

If you try to load a module that is not compatible with the saved module, the cursor changes to a no drop shape. The following might cause the saved file to be incompatible:

- The width of the current module is less than the width of the saved module.
- The module type is not the same. Use a saved DSO module for a DSO, and a saved logic analyzer module for a logic analyzer.

If you load a saved system with a different module configuration than your current system, the logic analyzer displays a dialog box similar to the following:



Click OK to accept the suggested configuration. If you click Cancel, the logic analyzer displays the Load System Option dialog box. Use this dialog box to load specific modules from the saved system.

NOTE. *If you load a setup that you saved with suppressed samples, the suppressed samples are not present in the setup.*

An unmerged module can be loaded into a merged module. The master of the merged pair must have a width equal to or greater than the unmerged module. A merged module can be loaded into a merged module, provided the master and slave of the destination has a width equal to or greater than the master and slave of the saved pair.

To Use

To open the Load System Options dialog box:

1. Go to the File menu from any window.
2. Select Load System.
3. Select a TLA system file. The Options button becomes active when a system file is selected.
4. Click Options.

To load a saved module, drag the module icon from the top of the Load System Options dialog box to a module icon at the bottom of the dialog box.

NOTE. *The module icon does not move; however the cursor changes shape to indicate a drag function.*

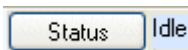
Related Topics

[Saving a System \(see page 78\)](#)



[Merge Modules Properties \(see page 372\)](#)

Status Monitor Dialog Box

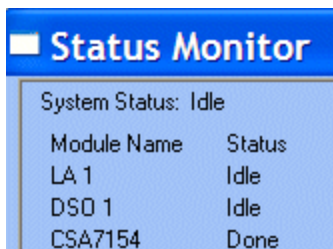
To open the Status Monitor dialog box, click the Status button on the toolbar.



The Status Monitor is used to determine the status of your acquisition while it is running.

The Run animation control indicates when the system is in the acquisition mode by changing from  to  when the Run button is clicked.

When you hover your mouse pointer over the  icon, a tool tip box provides you with the acquisition status of your installed modules as shown below.



The Status Monitor dialog box contains a summary area at the top that lists a one-line summary of each module installed.

The Status Monitor dialog box shows the following:

- Status of the acquisition: Idle, Waiting for Trigger, or Triggered.
- The Start Time of the acquisition.
- The name of the module.
- The status of memory for all modules during acquisition is shown on the left of the status dialog box. Blocks are filled in to the left of the T (trigger position). Blocks are filled to the right of the T after the module is triggered.
- The number of repetitive acquisitions when in Repetitive mode.
- The trigger state of the logic analyzer module trigger program that is executing.
- The number of samples remaining to be filled in memory.

- The value of each logic analyzer counter and timer when appropriate. Only shown for logic analyzer modules.
- The state of the module's intermodule output signal, shown as Sig/Arm Out.
- The status of data comparison during logic analyzer memory compare operations.
- Clock warning when the logic analyzer senses no clock activity. If this message appears, check for a missing clock signal or for an error in the sampling setup.

The Status Monitor dialog box is resizable.

Related Topic

[About Data Properties \(see page 196\)](#)

External Signal Selection

When using the External Signal In and External Signal Out functions, select an internal signal.

- Select None to prevent signals from being sent or received at the external connector.
- Select 10 MHz clock to send a 10 MHz clock signal to the External Signal Out connector. This signal will be available continuously at the connector until explicitly disconnected by changing the External Signal Out selection in the Signals tab.

NOTE. *The 10 MHz clock signal is interrupted briefly during the following actions: Load System, Default System, diagnostic execution, and calibration.*

- Select Signal 1-4 to assert External Signal Out when the specified signal is true. Signals 1 and 2 are high-speed, ECL rate signals, and are usually the preferred choice for both internal and external signal applications. Signals 3 and 4 are functionally equivalent to signals 1 and 2, but operate at the slower TTL-rate.

Signals 1 and 2	162 MHz maximum
	62.5 MHz maximum (at External Signal Out connector)
Signals 3 and 4	12.5 MHz maximum

Internal Signal Selection

Signals 1 and 2 are high-speed, ECL-rate signals, and are usually the preferred choice for both asynchronous and synchronous signal applications. Signals 3 and 4 are functionally equivalent to signals 1 and 2, but operate at the slower TTL rate.

Signals 1 and 2	162 MHz maximum (internal)
	62.5 MHz maximum (at External Signal Out connector)
Signals 3 and 4	12.5 MHz maximum

Selection Dialog Box

Use the Selection dialog box when you want to do one of the following tasks:

- Add a data source from a TLA system or module file to the current system. Click Browse to search for the required file.
- Load a data window from a TLA system file.
- Load a module setup onto a selected module in the current system.
- Load triggers. Click Browse to select a trigger from your modified trigger list.
- Load a trigger setup onto a selected logic analyzer module in the current system.
- Load a search definition onto a selected data window in the current system. Click Browse to select a previously saved search definition.

MagniVu Data

Overview

MagniVu data is the result of asynchronous digital oversampling architecture. This data is automatically acquired whenever you perform an acquisition. This means that, even if you specify synchronous acquisition, you can view the asynchronous resolution data at any time without having to reacquire.

Behavior

MagniVu acquisition rates differ based on the module that you have. If you are using a TLA7000 Series logic analyzer module or a TLA5000B Series logic analyzer, your acquisition rate is selectable, based on your [MagniVu Storage Rate \(see page 172\)](#), and is constrained by both PowerFlex and the current maximum acquisition rate for main memory.

Related Topic

[Adding MagniVu Data to a Listing or Waveform Window \(see page 75\)](#)

System Trigger (Trigger All Modules)

The system trigger is an overriding command that causes all modules to acquire their post trigger data, regardless of whether they have been armed or have fulfilled their own trigger conditions.

The system trigger is the primary data acquisition reference point. In data windows, timing and location information is relative to the system trigger.

When the system trigger occurs, the trigger signal will appear at the external System Trigger Out connectors of the analyzer.

High-Level Language support

You can correlate the High-Level Language (HLL) source code that you wrote with the disassembly listing, showing how the code was executed on your target system, and then acquired by the logic analyzer. The correlation is based on symbolic information that is extracted from your object file or load module. You can configure the logic analyzer to access your source files.

You can step through each executed source statement in the Source window and view the results in a correlated Listing window.

You can set user-defined marks as break points within the code and then trace the execution of the code between the marks.

The following file formats are supported: [OMF51](#), [OMF86](#), [OMF166](#), [OMF286/386](#), [IEEE695](#), [COFF](#), [ELF/DWARF](#) and [ELF/STABS](#).

In addition to the file formats listed below, Tektronix also provide the [Tektronix Symbol File Format \(TSF\)](#) (see page 350), a text format. For information on any other file formats, contact your local Tektronix representative.

Start-Up Properties

Use to select which system to open after you power on the analyzer. You are presented with the following choices:

Last System file opened in previous session. The TLA application automatically loads at start-up the last saved system file that you explicitly loaded.

Default System. The TLA application loads the default setups for all modules at start-up.

Saved System. The TLA application loads a saved system file that you specify at start-up.

The default system also consists of the following:

- Default setups for all modules.
- Default settings for internal signals specified in the System Configuration dialog box.

- One data window for each module. A Waveform window is created for each DSO module. A data window of the default type, as specified in the Defaults tab, is created for each logic analyzer module. Only data sources from the physical modules in the current system are included.
- All usable modules are on.

To open the Start-Up tab from the System menu, click Options and then click Start-Up.

Related Topic

[Defaults Properties \(see page 378\)](#)

Power-On Diagnostics Properties

Overview

Use to view the results of the power-on diagnostics. The Test Name column lists the mainframe, slot number and name of installed modules, and specific tests.

Behavior

If any test failed, the Power-On Diagnostics page opens as the first active window and the first failed test is highlighted. If a test fails you can either select the Extended Diagnostics tab and rerun the test, or dismiss this page by clicking the Close button. If you dismiss this page, the application finishes loading.

NOTE. *Mainframe power-on diagnostics only run when the system powers on.*

If a module is not recognized, the module type area is filled in with any available information or the word Unknown is displayed.

To Use

To open the Power-On Diagnostics tab from the System menu, click Calibration and Diagnostics and then click Power-On Diagnostics.

Related Topics

[Running Extended Diagnostics \(see page 133\)](#)

[Contacting Tektronix \)](#)

Self Calibration Properties

Overview

Use to run calibration procedures on installed modules, either individually or as merged pairs. You should run these procedures after repair and at least once a year for the logic analyzer module. The DSO module requires self-calibration if the ambient operating temperature has changed more than 5 °C since the last calibration or once a week if the vertical settings of 50 mV full scale or less are used. A warning message to calibrate appears if ambient temperature goes out of tolerance. Perform self-calibration after a 30-minute warm up.

NOTE. *Merged modules must be calibrated individually and as a set.*

Status list

The following is a list of status messages:

- **Running.** The calibration procedure is being executed. The text blinks to indicate progress.
- **Calibrated.** The module calibrated successfully.
- **Fail.** The module did not calibrate. Contact your local Tektronix service representative.
- **No Calibration Required.** The module either does not need to be calibrated or it does not have any self-calibration functions.
- **Unknown.** Try to calibrate again. If the module has the same status, contact your Tektronix service representative.

To Use

To open the Self Calibration tab from the System menu, click Calibration and Diagnostics, and then click Self Calibration.

Related Topics

[Power-On Diagnostics Properties \(see page 370\)](#)

[Extended Diagnostics Properties \(see page 337\)](#)

[Calibrating Modules \(see page 59\)](#)

[Contacting Tektronix \)](#)

Mainframe Properties

Use this information when you contact Technical Support.

To open the Mainframe tab from the System menu, click System Properties and then click Mainframe.

Modules Properties

Shows information about modules installed in the analyzer.

1. From the System menu, click System Properties.
2. Click the Modules tab.

Module Specifics Properties

Provides information about a selected module. You use this information when contacting Technical Support.

1. From the System menu, click System Properties.
2. Click the tab of the module for which you want information.

Merge Modules Properties

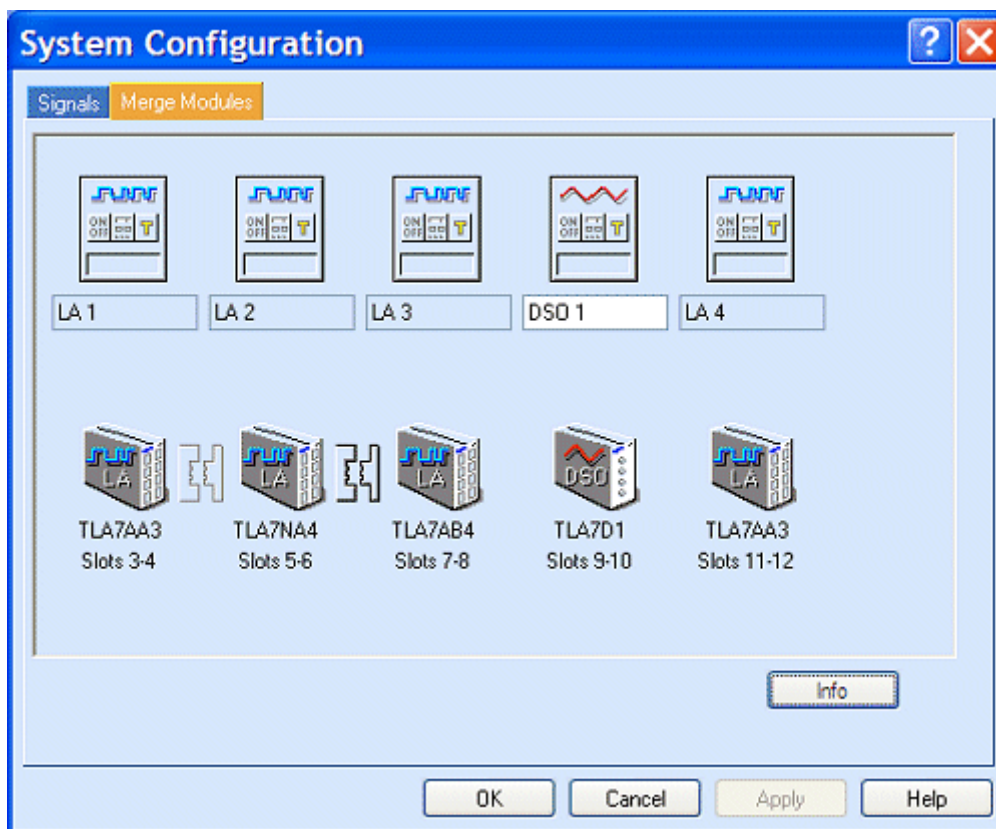
Overview

You can merge up to five logic analyzer modules to increase width. Merged modules function as a single unit and the data it acquires is automatically time-correlated. This is useful when analyzing buses or devices with a high count. Some microprocessor supports require merged modules.

Behavior

Modules must be physically connected and in the same mainframe before they can be merged by software. Modules must be adjacent and connected as described in the user manual. Only modules that have been physically connected and are capable of being merged are shown in the Merge Modules tab with a merge button. Refer to [Merging Modules \(see page 53\)](#) for the rules on merging modules.

The state of the button indicates whether the modules are currently merged, as shown:



To Use

To open the Merge Modules tab from the System menu, click System Configuration and then click Merge Modules. To merge or unmerge modules, click the merge button between the module icons.

Related Topics

[Merging Modules \(see page 53\)](#)

[Self Calibration Properties \(see page 371\)](#)

[Merged Module Synchronous Sampling \(see page 27\)](#)

Preferences Properties

Use the preference properties page to specify user preferences, such as hiding the Status bar, displaying confirmation messages, specifying click and drag zoom behavior, or showing the Easy Setup dialog. These settings cannot be modified anywhere else in the application.

To access user preferences from the System menu, click Options to display the Options dialog box. Then click the Preferences tab.

System Source Files Properties

Overview

The System Source Files tab defines the default location of source files for new Source windows. The settings you specify in this tab will be the default settings used to locate source files when you first create a Source window. You can specify a list of directory paths and a list of possible file suffixes.

Behavior

When specifying the search path, you can use an asterisk (*) as the final component of a search path directory. For example, if you specify the following path, C:\MySources*, the MySources directory and all directories (one-level deep) within MySources will be searched. However, you cannot use the asterisk as a general-purpose wild card character. For example, the use of an asterisk in C:\MySources** or C:\MySources\abc* is invalid.

Specify the complete absolute (not relative) path name for the file in the Search Path List. Otherwise the Source window will not locate the source file. Make sure that the source file you are looking for has a unique file name. When searching for source files it is possible to load the wrong file if there is another file with the same name in the directories specified by the search path. The Source window always loads the first file that matches the search criteria.

NOTE. *This tab is similar to the Source Window Source Files tab. The information that you supply in this property page will be used as default settings for new Source windows. The information that you supply for the Source Window Source Files tab is only used for the current Source window.*

To Use

To open this page from the System menu, click Options and then click Source Files.

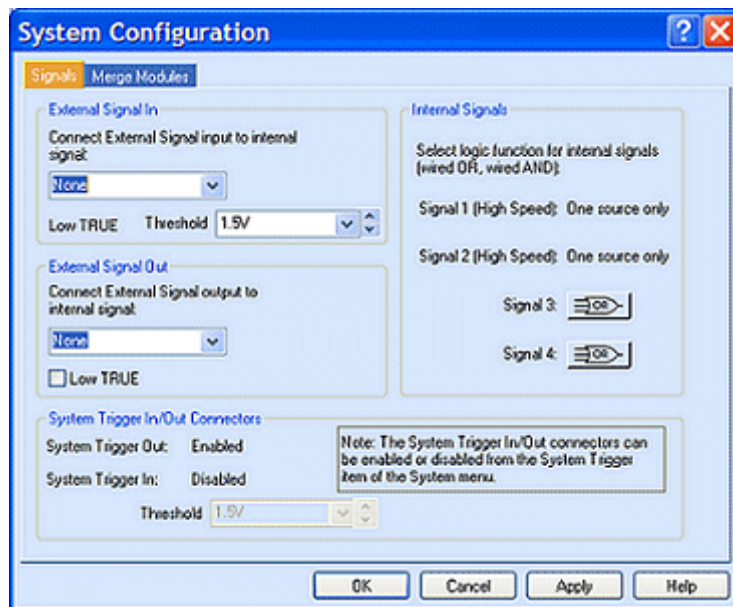
- Select Search Path List to specify source file locations.
- Select Suffix List to specify possible file suffixes. The default suffixes are .c and .s.
- Click Add to add items.
- Click Edit to edit the selected item.
- Click Delete to delete a selected item from the list.

Related Topic

[Source Window Source Files Properties \(see page 237\)](#)

Signals Properties

The logic analyzer has four internal signals that you can use to set up trigger conditions between modules, or to send or receive signals external to the logic analyzer. Use the Signals tab in conjunction with the module trigger programs to configure these signals for your application. The trigger programs and external signal input determines when the signals occur. The Signals tab specifies signal characteristics. To configure signals for your application from the System menu, click System Configuration, and then click the Signals tab.



Internal Signals

All logic analyzer modules can set and clear any of the four internal signals. The DSO can set but not clear any signal. The logical output of these signals can be used as an event in other trigger programs. You can also connect the internal signals to the External Signal In and External Signal Out connectors on the mainframe, so that you can use an external signal as a trigger event or send a signal out when a trigger condition is met.

Internal signaling is for users with special trigger programming requirements. Internal signaling adds flexibility to trigger programming, but also adds complexity. When using internal signaling, take care that the signals are correctly set and cleared, and that the trigger programs for all modules are compatible with regard to signal usage. Also, correctly set the internal signal attributes in the Signals tab of the System Configuration dialog box.

Internal signaling is done through trigger actions specified in the Clause Definition dialog box and the DSO Trigger tab.

NOTE. *Arming and internal signaling are mutually exclusive. You cannot simultaneously arm modules and set signals. You can test for a set signal (for example, If Signal X Is True), but unless you have designated an external signal for this purpose, you will be unable to set the signal when using the arming feature.*

External Signals

The logic analyzer can send and receive signals external to the logic analyzer using the mainframe External Signal In and External Signal Out connectors. Use the External Signal In function to include a signal from your system under test or other test equipment as part of the trigger program. Use the External Signal Out function to send a signal to the system under test or other test equipment from a trigger program.

The connection from the module to the external connector is made by one of the four internal signals. Designate which internal signal is to be used for this purpose.

The external signal connectors are located at the rear of the portable mainframe and at the front of the benchtop mainframe. The external signals all operate at TTL voltage levels.

The polarity for External Signal In is always set to Low TRUE. You have a choice of low or high true for External Signal out. For a high true signal, clear the selection Low TRUE.

Signal Logic Function

To use internal signals, select which internal signal logic function is appropriate for your trigger program. Signals 1 and 2 (high speed) can only be asserted by one module each. For signals 3 and 4 the modules can be wired-OR or wired-AND. Selecting the OR function means that any module can assert the signal. Selecting AND means that all modules must set the signal for it to be asserted. The same logic applies to clearing signals.

System Trigger Connectors

The System Trigger In/Out Connectors group box contains readouts that reflect the current state of the System Trigger In and System Trigger Out connectors. You can also set the threshold for the System trigger in connector.

NOTE. *Only modules that are on can send arming or internal signals.*

To Use

To open the Signals tab from the System menu, click System Configuration and then click Signals.

Related Topics

[Clause Definition Dialog Box \(see page 163\)](#)

[DSO Trigger Settings \(see page 385\)](#)

[External Signal Selection \(see page 367\)](#)

[Internal Signal Selection \(see page 367\)](#)

Color Properties

Use to create, remove, and modify color schemes.

1. Select an Element you want to change from the list and then select a color from the color matrix.
2. Click Save As to enter a new name for your scheme or click OK in the Save Scheme dialog box to keep the existing name and redefine the defaults for it.

NOTE. *You cannot redefine or remove factory defaults.*

The following table describes each element.

Element Name	Description
Data Background	Default background color for a data window's data area
Data Foreground	Default text color for the data in a data window
DSO Waveform	Default color for DSO waveform data
LA Waveform	Default color for logic analyzer waveform data
LA – MagniVu Waveform	Default color for logic analyzer MagniVu waveform data
Trigger	Default color for trigger marks
Cursor 1	Default color for cursor 1
Cursor 2	Default color for cursor 2
Mark	Default color for user marks
Compare: Acq = Ref	Default color for highlighting equalities between acquisition data and reference data
Compare: Acq != Ref	Default color for highlighting differences between acquisition data and reference data
Glitch	Default color for glitches
Waveform Readouts	Default color for DSO ground, volts readout, trigger level, magnitude readouts and busform readouts
Graticule	Default color for the graticule in a Waveform window
Highlight	Default color for disassembly highlighting in a Listing and Source window
Histogram Bar	Default color for bars in a Histogram window
Uncorrelated Source	Default color for uncorrelated text in a Source window

A default color scheme can be assigned for new data windows from the Defaults tab, which is located in the Options dialog box.

From the System menu, you can access the Color settings by clicking Options and then clicking the Color tab.

Related Topic

[Defaults Tab \(see page 378\)](#)

Software Properties

Shows what TLA software is installed in the analyzer.

1. From the System menu, click System Properties.
2. Click the Software tab.

Defaults Properties

Use to specify defaults used throughout the application. The settings you enter here will be the default settings used when creating new data windows.

To access these settings from the System menu, click Options and then click the Defaults tab.

Save Color Scheme Dialog Box

Use to save a new color scheme set in the Color tab of the Options dialog box.

To open this dialog box from the System menu, click Options, click the Color tab and then click Save As.

Related Topic

[Color Properties \(see page 377\)](#)

Presets Properties

Use the Options Presets to view and modify preset names and values. The presets are used as list entries in list box controls for selecting trigger thresholds, vertical range, vertical offset, and probe thresholds. You can modify all preset names and values. Preset names are limited to 12 characters. You can return all preset names and values to the factory settings by clicking Reset All. If you enter a value that is out of the range shown below, the value is set to the nearest in-range value.

To change options presets from the System menu, click Options and click the Presets tab.

Tools Properties

Overview

The Tools tab of the Options dialog box allows you to define up to 16 external applications (or tools) that may be launched from within the TLA application.

These tools can be associated with any program that runs under the Windows operating system or any type of document that Windows knows how to open. When a tool is defined here, it is automatically added to the Tools sub-menu of the System menu in the TLA menu bar. Also, a toolbar button for each tool is added to the TLA Toolbar.

Once a tool has been defined, you can run the program by choosing its name from the Tools menu or clicking on its toolbar button.

To Use

- Click the Add button to add a new tool.
- Click the Edit button to edit the properties of the selected tool, including its Menu Name, Command, Arguments, Initial Directory, and Run Minimized status.
- Click the Delete button to delete a selected tool from the list.
- Click the Move Up arrow button to move the selected tool up one position in the list.
- Click the Move Down arrow button to move the selected tool down one position in the list.

To open the Tools property page, from the TLA menu bar select System ® Options, and select the Tools tab.

NOTE. *Arguments and commands with embedded spaces must be enclosed in quotes in the Tool Properties (App Options) dialog box for the commands to work.*

Related Topic

[Tool Properties Dialog Box \(see page 379\)](#)

Tool Properties Dialog Box

Overview

Use the Tool Properties dialog box to specify the properties of an external tool.

To Use

Specify the name of the tool in the Menu Name box. This is the name that is displayed in the Tools menu. Placing an ampersand before a letter in the tool name makes that letter an accelerator key.

Enter the path and name of the document or application that you want to open in the Command box. You can click the Browse button to open a Windows file browser to help find the application. If a document is entered here, the executable to open the document is found based on its registered Windows file type. For example, a document with a .txt extension is opened with Notepad.

Use the Arguments box to enter any command line arguments that are required by the application.

Use the Initial Directory box to specify the default working directory for the application.

Click Run Minimized to specify that the application will run in a minimized window.

Click the Test Run button to launch the tool with the current dialog settings.

To open the Tools property page, from the TLA menu bar select System ® Options, and select the Tools tab. Click the Add button, or select the tool you want to modify and click the Edit button.

NOTE. Arguments and commands with embedded spaces must be enclosed in quotes in the Tool Properties (App Options) dialog box for the commands to work.

Related Topic

[Tools Properties \(see page 379\)](#)

General Purpose Data Window Shortcuts

To	Press
Switch data windows	CTRL+Tab
Scroll data up 10 pages	Shift+Page Up
Scroll data down 10 pages	Shift+Page Down
Scroll data to the top of the window	Shift+Home
Scroll data to the end of the window	Shift+End
Move active cursor up one page	CTRL+Page Up
Move active cursor down one page	CTRL+Page Down
Move active cursor to top of the data	CTRL+Home
Move active cursor to end of the data	CTRL+End
Move active cursor up 10 pages	CTRL+Shift+Page Up
Move active cursor down 10 pages	CTRL+Shift+Page Down
Move active cursor to the top of data	CTRL+Shift+Home
Move active cursor to the end of data	CTRL+Shift+End

The movement shortcuts follow these general rules:

- Arrows with no modifier keys scroll the data
- Arrows with the CTRL key move the active cursor
- The Shift key provides 10 times the movement

Related Topics

[Listing Window Shortcuts \(see page 383\)](#)

[Waveform Window Shortcuts \(see page 382\)](#)

[Source Window Shortcuts \(see page 384\)](#)

[Histogram Window Shortcuts \(see page 385\)](#)

[Menu Shortcut Keys \(see page 381\)](#)

Menu Shortcut Keys

You can use the following shortcut keys for manipulating menus, editing windows, and moving around in data windows.

File Menu

To	Press
Return the system to the default setups	CTRL+D
Load a saved system	CTRL+O
Save a system setup	CTRL+S
Print the active window	CTRL+P

Edit Menu

To	Press
Cut a selected item to the clipboard	CTRL+X
Copy a selected item to the clipboard	CTRL+C
Paste items from the clipboard	CTRL+V
Undo edit	CTRL+Z

Data Menu

To	Press
Search backward	CTRL+B
Search forward	CTRL+F

System Menu

To	Press
Display the Status Monitor	CTRL+M
Run or Stop	CTRL+R

Window Menu

To	Press
Create a new data window	CTRL+N

Related Topics

[Waveform Window Shortcuts \(see page 382\)](#)

[Listing Window Shortcuts \(see page 383\)](#)

[Histogram Window Shortcuts \(see page 385\)](#)

[Source Window Shortcuts \(see page 384\)](#)

[General Purpose Data Window Shortcuts \(see page 380\)](#)

Waveform Window Shortcuts

To	Press
Go to next Sample Suppression change	CTRL+0
Go to previous Sample Suppression change	CTRL+9
Go to next trigger	CTRL+T
Display Go To dialog box	CTRL+G
Move Cursor 1 to center of window	CTRL+1
Move Cursor 2 to center of window	CTRL+2
Zoom in	CTRL+I
Zoom out	CTRL+U
Add a mark	CTRL+K
Add a waveform	CTRL+W
Scroll data left 50 pixels	Shift+Left arrow
Scroll data right 50 pixels	Shift+Right arrow
Move active cursor left five pixels	CTRL+Left arrow
Move active cursor right five pixels	CTRL+Right arrow
Move active cursor left 50 pixels	CTRL+Shift+Left arrow
Move active cursor right 50 pixels	CTRL+Shift+Right arrow

The movement shortcuts follow these general rules:

- Arrows with no modifier keys scroll the data
- Arrows with the CTRL key move the active cursor
- The Shift key provides 10 times the movement

Related Topics

[Listing Window Shortcuts \(see page 383\)](#)

[Menu Shortcut Keys \(see page 381\)](#)

[General Purpose Data Window Shortcuts \(see page 380\)](#)

Listing Window Shortcuts

To	Press
Go to next Sample Suppression change	CTRL+0
Go to previous Sample Suppression change	CTRL+9
Go to next trigger	CTRL+T
Display Go To dialog box	CTRL+G
Move Cursor 1 to center of window	CTRL+1
Move Cursor 2 to center of window	CTRL+2
Add a mark	CTRL+K
Add a column	CTRL+L
Scroll data up 10 samples	Shift+Up arrow
Scroll up 10 lines	Shift+Up arrow
Scroll data down 10 samples	Shift+Down arrow
Scroll down 10 lines	Shift+Down arrow
Move active cursor up one sample	CTRL+Up arrow
Move active cursor up one line	CTRL+Up arrow
Move active cursor down one sample	CTRL+Down arrow
Move active cursor down one line	CTRL+Down arrow
Move active cursor up 10 samples	CTRL+Shift+Up arrow
Move active cursor up 10 lines	CTRL+Shift+Up arrow
Move active cursor down 10 lines	CTRL+Shift+Down arrow
Move active cursor down 10 samples	CTRL+Shift+Down arrow

The movement shortcuts follow these general rules:

- Arrows with no modifier keys scroll the data
- Arrows with the CTRL key move the active cursor
- The Shift key provides 10 times the movement

Related Topics[Waveform Window Shortcuts \(see page 382\)](#)[Histogram Window Shortcuts \(see page 385\)](#)[Source Window Shortcuts \(see page 384\)](#)[General Purpose Data Window Shortcuts \(see page 380\)](#)[Menu Shortcut Keys \(see page 381\)](#)**Source Window Shortcuts**

To	Press
Display Go To dialog box	CTRL+G
Move Cursor 1 to center of window	CTRL+1
Move Cursor 2 to center of window	CTRL+2
Add a mark	CTRL+K
Step Forward	CTRL+H
Step Backward	CTRL+J
Scroll data up 10 lines	Shift+Up arrow
Scroll data down 10 lines	Shift+Down arrow
Move active cursor up one line	CTRL+Up arrow
Move active cursor down one line	CTRL+Down arrow
Move active cursor up 10 lines	CTRL+Shift+Up arrow
Move active cursor down 10 lines	CTRL+Shift+Down arrow

The movement shortcuts follow these general rules:

- Arrows with no modifier keys scroll the data
- Arrows with the CTRL key move the active cursor
- The Shift key provides 10 times the movement

Related Topics[Listing Window Shortcuts \(see page 383\)](#)[Waveform Window Shortcuts \(see page 382\)](#)[General Purpose Data Window Shortcuts \(see page 380\)](#)[Menu Shortcut Keys \(see page 381\)](#)

Histogram Window Shortcuts

To	Press
Scroll up 10 ranges	Shift+Up arrow
Scroll down 10 ranges	Shift+Down arrow

The movement shortcuts follow these general rules:

- Arrows with no modifier keys scroll the data
- The Shift key provides 10 times the movement

Related Topics

[Listing Window Shortcuts \(see page 383\)](#)

[General Purpose Data Window Shortcuts \(see page 380\)](#)

[Menu Shortcut Keys \(see page 381\)](#)

DSO Trigger Settings

Overview

The DSO Trigger tab contains all controls for setting DSO trigger events and actions. Threshold changes are recognized and executed immediately during acquisitions. Execution of all other controls is delayed until the next acquisition.

Behavior

Use Mode to select whether the DSO waits for a trigger (Normal) or forces a trigger after a set length of time has passed without a trigger (Auto).

Use Action to select what happens when a trigger is recognized. The trigger event either triggers the DSO, triggers all modules, triggers and sets a signal, or triggers and arms another module.

Action Target selects the event or the condition Trigger Action will act on:

- Trigger & Set Signal. A Set Signal action allows you to set signal 1, 2, 3, or 4.
- Trigger & Arm. An Arm action allows you to arm other installed modules. The default is the module in the lowest-numbered slot that can be armed.

Use Trigger position to set the amount of data in the data record that occurs before the trigger.

Use Event Type to select the type of event the DSO will recognize as a trigger.

Related Topic

[System Trigger and Reference System Trigger Relationship \(see page 165\)](#)

Select Status Bits Type Dialog Box

Use the Select Status Bits Type dialog box to select the logic analyzer's status bits for additional triggering and data display.

Depending on your logic analyzer module, you can select one or more status bits for triggering and display. The status bits of a module (or master module in a merged set) are available for use in the Signal Mapping area of the LA Setup window. In most cases, you can use the status bits like any other signal bit such as comparing data, naming signals, and assigning the bits to a group.

However, because the status bits are not real data signals, you cannot access certain properties of the status bits such as thresholds, iCapture, and multiplexed (demux) sampling.

The available status bits depend on the logic analyzer module.

Saved Triggers

Overview

When saving from the Easy Trigger tab, use the Save dialog box to save your TLA file, which contains trigger state information as well as information about the currently selected Easy Trigger program. If you do not use an Easy Trigger program as the basis of your modified trigger design, but instead use the Power Trigger tab to develop your trigger program, only a TLA file with the state information is saved.

To Use

Saving your trigger program requires that you specify both a file name and the name of the folder where you would like your customized trigger programs to reside. In addition, you can enter comments about trigger program construction or functionality.

To open the Save dialog box from the File menu, click Save Trigger. You can alternatively open the Save dialog box by clicking the  button from the toolbar.

Sampling and Digitizing a Signal

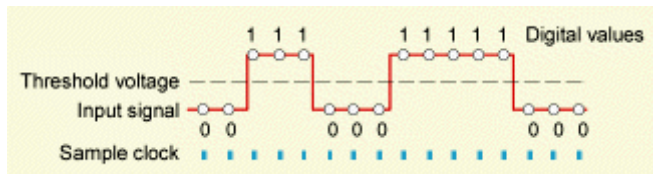
Overview

Acquisition is the process of sampling the input signal, digitizing it, and assembling it into a waveform record. The order and method of accomplishing these functions is different in the LA and DSO modules.

Logic Analyzer

The logic analyzer converts incoming data into binary using a comparator with a user-selectable threshold voltage.

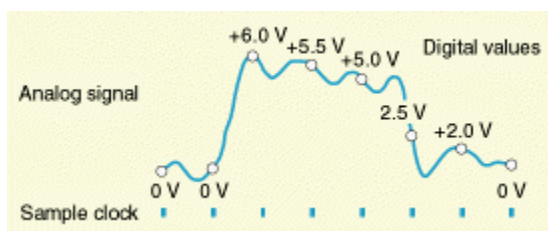
If the incoming signal is above the threshold voltage, the signal is converted to a one. If the incoming signal is below the threshold voltage, the signal is converted to a zero.



After digitizing the incoming data, the logic analyzer samples it at regular intervals. The sampled and digitized points are stored in memory along with the corresponding timing information.

Oscilloscope

The oscilloscope converts incoming data into 8-bit digital values.



After digitizing the incoming data, the oscilloscope samples the voltage levels at regular intervals. The sampled and digitized points are stored in memory along with the corresponding timing information.

Intermodule Interactions and Time Correlation

Each module has its own setup, trigger, and sampling functions. (Logic analyzer modules can include microprocessor support as part of their setup.) Each module also acquires and stores its own data.

Acquisition

When you start an acquisition, all modules start acquiring data together. (Exceptions are when one module has been programmed to arm another or when a module has been turned off.) Modules stop acquiring data individually, according to their trigger programming. You also have the option of setting the logic analyzer to operate in repetitive mode, in which the modules acquire data and update the data windows continually until you manually stop the acquisition.

Triggering

Modules readily communicate with one another by means of their trigger programs. You can specify functions such as the following:

- Trigger all modules (system trigger)
- One module arms another
- Modules respond to events declared by another module (internal signals)

Analysis

After the modules have captured and stored data, you can view the data in a Listing or Waveform window. All data is time-correlated in the display, regardless of its source. Due to the precise time stamp information stored with the data and the tightly-integrated communication between modules, the logic analyzer interleaves data acquired from various sources. Because time stamp information is always stored with the data, you can also compare saved data and current data with no loss of accuracy.

MagniVu data is also time-correlated with regular data. Because MagniVu data is always present, you can easily compare a normal acquisition with the MagniVu counterpart.

Related Topics

[Trigger Programs \(see page 181\)](#)

[MagniVu Data \(see page 388\)](#)

High Speed Timing

The logic analyzer provides high-speed timing support through MagniVu data. The MagniVu data is stored in a separate memory that is parallel to the main memory. All data from the sampler goes directly to the MagniVu memory. The MagniVu memory also works like a circular buffer.

Unlike the acquisition memory, the MagniVu memory does not qualify data storage through the trigger function block. Like normal acquisition data, MagniVu data can be displayed in the Listing or Waveform windows.

MagniVu data is continuously acquired on all signals at the fastest sample rate possible for your LA model. Some LA models allow you to adjust the trigger position. (See the following table.)

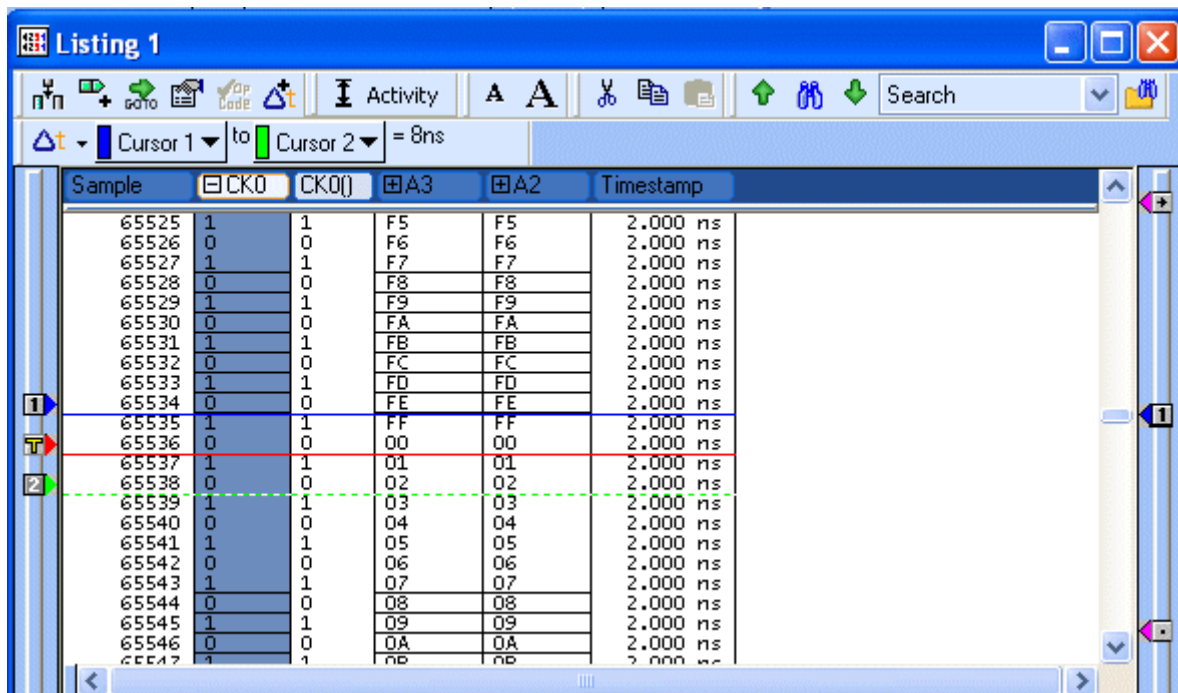
Instrument	Sample rate	Trigger position
TLA7000 Series module	125 ps with 16 Kb depth	adjustable
TLA5000B	20 ps	adjustable

Related Topic

[Adding MagniVu Data to a Listing or Waveform window \(see page 75\)](#)

Listing Data Concepts

In many cases, you will use the logic analyzer to observe the data flow in the target system. The data recorded by the logic analyzer can be displayed in a listing format, as shown:



Listing data is a table of sequential operations performed by the target system. In the Listing window, each data sample is displayed sequentially. Because each data sample includes time stamp information, it is a straightforward process to display acquisitions from multiple data sources. Samples from all specified data sources are interleaved in chronological order. For clarity, each line in the table represents a single data sample from a single data source.

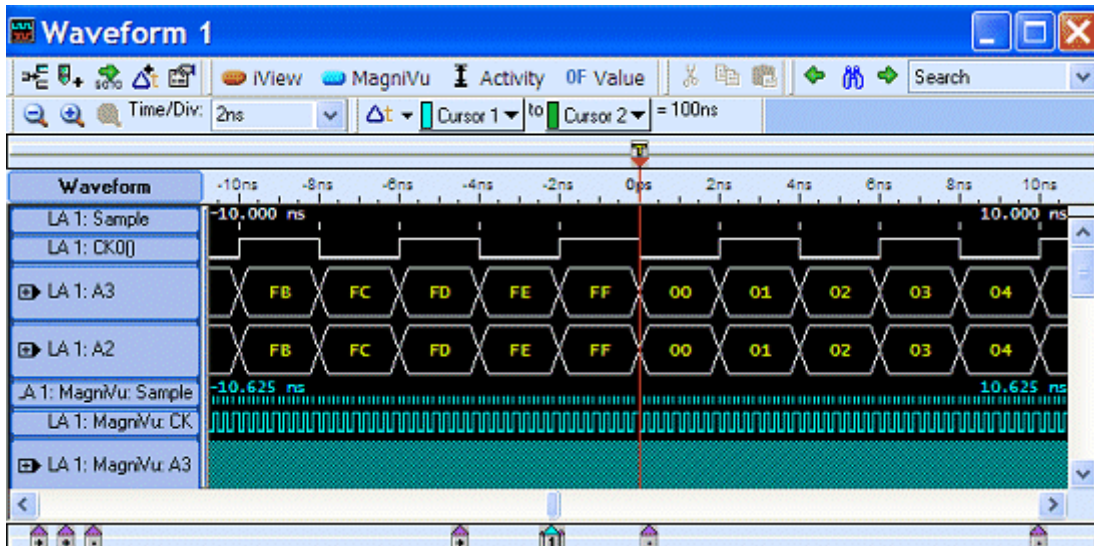
You control the presentation of the data by selecting the display radix of the columns. You can also make other format selections such as font size, color, and column width. You can use filters to display specific data.

You can expand or collapse the Group column to display the individual signals. You can also add individual signals.

You can include data acquired by the external oscilloscope in the Listing window. As with any module, the data samples from the oscilloscope are time-correlated with the other data and appear on separate lines.

Waveform Data Concepts

You can use the logic analyzer to observe the timing relationships between signals by displaying the recorded signal activity as a series of waveforms in the Waveform window. The following figure shows waveform data from a logic analyzer.



Waveform window

Each waveform is initially displayed in a separate track, but all waveforms are time-aligned horizontally and displayed in the same time scale. The inclusion of time stamp information with the stored data samples makes it easy to display time-correlated acquisitions from multiple data sources.

You control the horizontal scale of the acquired data in the display. (You do not, however, change setup parameters by changing settings in the waveform display.) You can change the signal group radix, waveform color, and waveform height. Like the Listing window, you can also use filters to display specific data.

When viewing logic analyzer data, you can view the data as individual signals. You can also display the logic analyzer data in groups of signals known as busforms. Use the busforms to display when data changes with respect to clock or control signals. Another method of displaying logic analyzer data is to overlay waveform signals. Use overlay waveforms to visually compare two or more waveforms at the same time.

When you are interested in displaying the value of a group of signals over a period of time, you can use magnitude mode. For example, using magnitude mode, you can view the signals connected to a 16-bit digital counter. The magnitude waveform appears as a sawtooth waveform as the counter values increase from the minimum value (00) to the maximum value (FF).

LA Versus DSO Module and External Oscilloscope Waveforms

A logic analyzer waveform appears to have zero-length rise and fall times. This is because the logic analyzer is recreating the waveform from the samples stored in memory, which are either a one or a zero.

No electronic signal is perfectly digital in nature; there is always some analog component. Consider a fast-rising pulse with ringing on the front edge, or glitches that can occur in a noisy circuit.

If you suspect problems caused by analog signal characteristics, such as signal voltages higher or lower than specified voltage levels or slow transition times, you should use a DSO module or an external oscilloscope to observe the voltage characteristics of the signal.

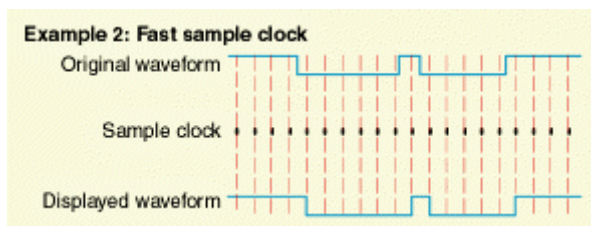
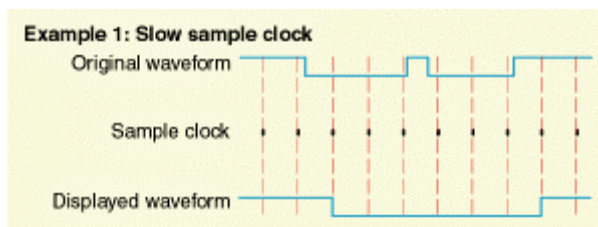
Related Topic

[Sampling Resolution \(see page 391\)](#)

Sampling Resolution

The accuracy of the waveform recreated from the sampled data depends on the sample rate used to record the incoming signals.

This is due to the fact that the waveform recreated by the logic analyzer is based on the sampled signals stored in memory. If the sample rate is too low the recorded data will produce an inaccurate display.



Related Topic

[Signal Resolution and Signal Duration \(see page 392\)](#)

Signal Resolution and Signal Duration

There is an important relationship between the [Sampling Resolution \(see page 391\)](#) of the recorded signal and the duration in terms of elapsed time.

Because the depth of the acquisition memory sets the total number of samples that can be recorded by the logic analyzer, increasing the sample rate provides better signal resolution but reduces the duration of the acquired signal. A faster sample rate will record a smaller portion of the signal, but with higher resolution.

It is important to remember that by using the [MagniVu \(see page 368\)](#) data feature, you can view high-resolution data centered around the logic analyzer trigger. This allows you to zoom in on the data while still maintaining visibility of a more extended signal duration.

Displaying Waveforms

Waveforms are rarely displayed in a one-to-one ratio between the sample points and pixels. Waveforms are usually displayed in a compressed or expanded format.

As a general rule:

- Waveforms are compressed when the time per pixel is greater than the time per sample.
- Waveforms are expanded when the time per pixel is less than the time per sample.

For compressed DSO waveforms, the display shows the lowest and highest points that occupy a given pixel column joined by a vertical line.

For expanded LA waveforms, the display points between the actual sample points are calculated.

For expanded DSO waveforms, Sin(x)/x interpolation computes the display points between the actual values acquired.

Autoset (DSO Module)

Autoset automatically chooses the DSO setup values, based on the input signal at the time you click Autoset. If you move the probe to another signal, click Autoset again to reset the setup values.

Autoset works best on repetitive signals that have 0 Volt offset. If the Autoset setup does not display the waveform as you want it, you can change the setup manually.

Autoset sets up all input signals, the horizontal, and the trigger regardless of which Autoset button is clicked.

Autoset makes the following selections automatically.

Vertical range:	A setting that does not clip the waveform
Sample period:	Period of the input signal divided by 100

Input coupling:	DC
Vertical offset:	0.0V
Bandwidth:	Full
Trigger level:	Center of the input signal range
Trigger type:	Edge
Trigger slope:	Positive
Trigger coupling:	DC
Trigger action:	Trigger (module)

NOTE. *If an acquisition is in progress, Autoset displays the Finishing Acquisition message box. Autoset uses Init @ 50% to adjust the trigger level to the center of the input signal range. Autoset does not alter any acquisition data or windows until the next acquisition.*

To use Autoset from the System window, click the DSO Setup button, and then click the Autoset button.

DSO Module Horizontal Settings

Horizontal settings control the rate at which the data is sampled and the amount of data acquired.

The sample period sets the interval between successive samples in a waveform record. Choose a sample period that is fast enough that the waveform will not be aliased, and slow enough to provide the waveform record length that you need. For best fidelity of repetitive waveforms, you should set the sample period to at least five times that of the waveform.

Memory depth sets the total number of data samples to be acquired. If you do not need to use the full memory depth to acquire data, select a smaller memory depth to get faster acquisitions.

To open the horizontal page from the System window, click the DSO Setup button, and then click the Horizontal tab.

Related Topic

[Autoset \(see page 392\)](#)

DSO Module Vertical Input Channel Settings

Overview

Channel settings adjust the vertical input parameters.

The vertical input voltage range control sets the attenuator full-scale voltage. If the range is selected from the list, it forces an offset value into the Offset control. You can select from the list of standard presets by clicking Options from the System menu, and then clicking the Presets tab.

Resolution

For best vertical resolution, set the range slightly larger than the expected input signal. Autoset automatically sets the range for best vertical resolution on signals with 0 offset.

Offset

The vertical input voltage offset is the offset voltage applied to the input signal. If the attached probe can be offset, it is applied at the probe tip. Otherwise, offset is applied at the BNC. If Range is changed using the preset list, Range also sets the Offset.

Bandwidth

Bandwidth is the range of frequencies that can be acquired and displayed accurately. Your bandwidth selection sets the upper limit of frequencies that will be acquired and displayed. Bandwidth filters reduce unwanted noise and aliasing, but can substantially change waveform fidelity.

- Select the 20 MHz filter to set the upper limit of frequencies that are acquired to 20 MHz. The 20 MHz filter does not provide complete anti-aliasing protection at slower sample rates.
- Select the 250 MHz filter to set the upper limit of frequencies that are acquired to 250 MHz.
- Select the Full bandwidth for best fidelity of signals with high frequency components. The Full bandwidth filter passes the full analog bandwidth.

Coupling

Coupling selects how the input signal is coupled to the vertical input channel.

- Use DC coupling to see the entire waveform (both AC and DC components).
- Use AC coupling to view only the AC components of the waveform (DC components are blocked).
- Use GND coupling to display a zero-volt waveform.

Signal Name

The Signal Name you enter becomes the new name for the waveform when it is displayed.

Termination

Select the vertical termination impedance appropriate for the input signal.

NOTE. Termination is at the BNC connector, not the probe tip. Input signal and vertical range and offset are limited by 50 W termination.

Probe Calibration

Probe calibration optimizes the signal path for this probe/channel/module combination.

Execute Probe Cal if any of these conditions have occurred:

- The ambient temperature has changed more than 5 degrees Celsius.
- The DSO notifies you that the probe is not calibrated.
- More than 1 month has passed since the probe was calibrated.

To open a vertical channel setting page from the System window, click the DSO Setup button, and then click a vertical channel tab.

Related Topics

[Calibrating Probes \(see page 59\)](#)

[Autoset \(see page 392\)](#)

DSO Module Setup

Before acquiring and displaying a waveform, first set up the DSO. Use the DSO Setup window to set the vertical, horizontal, and trigger parameters manually, or use Autoset for a quick, automatic setup based on the input signal.

NOTE. You cannot change setup parameters by changing the data display. Once you acquire data, you can manipulate the display, but that will not change the input settings used to acquire the data. Return to the DSO Setup window to alter input settings.

To open the DSO Setup window from the System window, click the DSO Setup button.

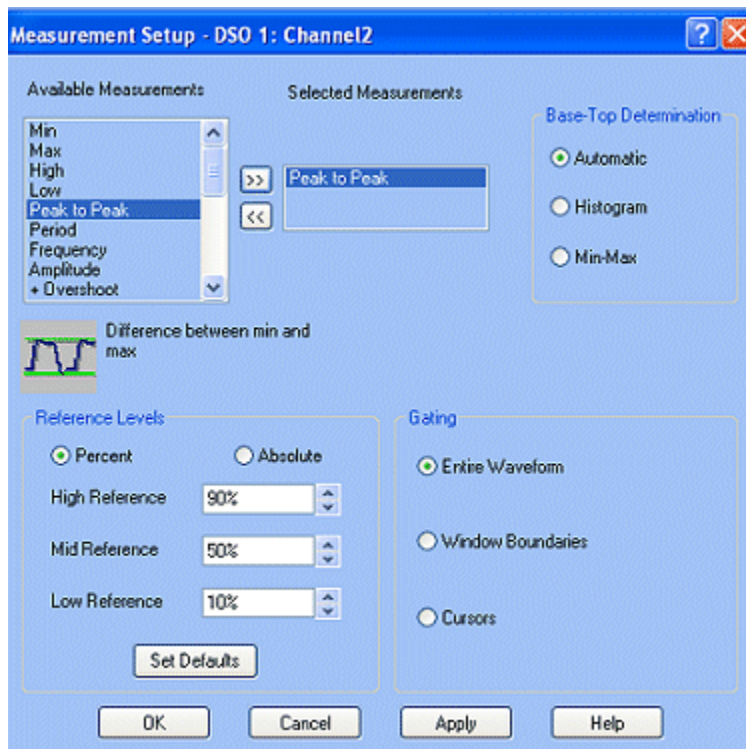
Click the following selections for more information:

- [Autoset](#)
- [DSO Vertical Input Channel Settings](#)
- [DSO Horizontal Settings](#)
- [DSO Trigger Settings](#)

DSO Measurement Setup

Overview

Use the Measurement Setup dialog box to select the measurements that will be performed on the DSO or external oscilloscope waveforms and to select the new measurement setup parameters. Click the Measurement Setup button in the Waveform Properties dialog box to access the Measurement Setup dialog box.



Behavior

- Available Measurements lists all of the available measurements that are supported on the DSO waveform. When you select an available measurement, a graphic and a brief description of the measurement are displayed beneath the Available Measurements list box. The following table lists the available measurements and provides a description of each.
- Selected Measurements displays up to three DSO waveform measurements that you select. You can modify the list of selected measurements using the Add >> and Remove << buttons.
- Base-Top Determination allows you to select the method for determining the base and top for a selected measurement: automatic, histogram, or min-max. The default setting is automatic, which lets the logic analyzer choose between histogram and min-max for the waveform under test.

- The Reference Levels group box enables you to select either absolute or percentage-based reference levels for the measured waveform. After selecting the type of waveform reference level, you can either set the High-, Low-, and Mid-Reference levels or elect to use the defaults.
- The Gating group box enables you to select the gating method for the measured waveform. You can select the entire waveform, window boundaries, or cursors to be used as the gating limits.

Available measurements	Description
High	The value used as 100% whenever high-reference, mid-reference, or low-reference values are needed, such as in fall time or rise time measurements. This method can be calculated using either the min/max or histogram method.
Low	The value used as 0% whenever high reference, mid reference, or low reference values are needed, such as in fall time or rise time measurements. This value can be calculated using either the min/max or histogram method.
Amplitude	This voltage measurement is the high value less the low value measured over the entire waveform or gated region.
Maximum	This voltage measurement is the maximum amplitude. It is typically the most positive peak voltage and is measured over the entire waveform or gated region.
Minimum	This voltage measurement is the minimum amplitude. It is typically the most negative peak voltage and is measured over the entire waveform or gated region.
Peak to Peak	This voltage measurement is the absolute difference between the maximum and minimum amplitude over the entire waveform or gated region.
Positive Overshoot	This voltage measurement is measured over the entire waveform or gated region and is expressed as: $\text{Positive Overshoot} = (\text{Maximum} - \text{High}) \div \text{Amplitude} \times 100\%$
Negative Overshoot	This voltage measurement is measured over the entire waveform or gated region and is expressed as: $\text{Negative Overshoot} = (\text{Low} - \text{Minimum}) \div \text{Amplitude} \times 100\%$
Mean	This voltage measurement is the arithmetic mean over the entire waveform or gated region.
RMS	This voltage measurement is the true Root Mean Square voltage over the entire waveform or gated region.
Rise Time	This timing measurement is the time required for the leading edge of the first pulse in the waveform or gated region to rise from the low reference value (default = 10%) to the high reference value (default = 90%) of the final value.
Fall Time	This timing measurement is the time required for the falling edge of the first pulse in the waveform or gated region to fall from the high reference value (default = 90%) to the low reference value (default = 10%) of the final value.

Available measurements	Description
Positive Width	This timing measurement is the distance (time) between the mid reference (default = 50%) amplitude points of a positive pulse. The measurement is made on the first pulse in the waveform or gated region.
Negative Width Measurement	This timing measurement is the distance (time) between the mid reference (default 50%) amplitude points of a negative pulse. The measurement is made on the first pulse in the waveform or gated region.
Period	The time required to complete the first cycle in a waveform or gated region. Period is the reciprocal of frequency and is measured in seconds.
Frequency	This timing measurement is a measure of the first cycle in a waveform or gated region. Frequency is the reciprocal of the period; it is measured in Hertz (Hz) where one Hz is one cycle per second.
Positive Duty Cycle	This timing measurement is the ratio of the positive pulse width to the signal period expressed as a percentage. The duty cycle is measured on the first cycle in the waveform or gated region. Positive Duty Cycle = (Positive Width) ÷ Period x 100% .
Negative Duty Cycle	This timing measurement is the ratio of the negative pulse width to the signal period expressed as a percentage. The duty cycle is measured on the first cycle in the waveform or gated region. Negative Duty Cycle = (Negative Width) ÷ Period x 100% .
Area	Area is a voltage-over-time measurement. The measurement is the area over the entire waveform or gated region expressed in volt-seconds. The area above the common reference point is positive and the area below the common reference point is negative.
Cycle Area	The voltage over time measurement. The measurement is the area over the first cycle in the waveform or the first cycle in the gated region expressed in volt-seconds. The area above the common reference point is positive and the area below the common reference point is negative.
Cycle Mean	This voltage measurement is the arithmetic mean over the first cycle in the waveform or the first cycle in the gated region.
Cycle RMS	This voltage measurement is the true Root Mean Square voltage over the first cycle in the waveform or the first cycle in the gated region.
Burst Width	This timing measurement is the duration of a burst and is measured over the entire waveform or gated region.

Reference levels	Description
High Reference	This defines the high reference level of a waveform; the default level is 90%. This reference level is used with the Low Reference level in the calculation of rise and fall times. To change this level, touch the control and then set the level with the multipurpose knob.
Mid Reference	This defines the middle reference level of a waveform; the default value is 50%. This reference value is primarily used for making measurements between edges such as pulse widths. To change this level, touch the control and then set the level with the multipurpose knob.
Low Reference	This defines the low reference level of a waveform; the default level is 10%. This reference level is used with the High Reference level in the calculation of rise and fall times. To change this level, touch the control and then set the level with the multipurpose knob.

Related Topics

[Waveform Options \(see page 216\)](#)

[Setting Up Waveform Measurements \(see page 126\)](#)

[Removing Selected DSO Waveform Measurements \(see page 127\)](#)

Oscilloscope Waveforms

Overview

Oscilloscope waveforms have a horizontal resolution of one display point per pixel width. Each horizontal, pixel-wide time slice has either a point or a vertical line to represent the vertical value of the acquired data sample. If the time per division is fast enough that each display point represents one or fewer acquired points, the vertical value is represented by a point. If the time per division is slower and multiple acquired data samples need to be represented within a pixel width, the vertical value is represented by a vertical line whose length and offset are determined by the acquired data values.

Behavior

When the time per division is decreased, waveforms are horizontally expanded, using sine x/x interpolation to create display points between the acquired sample points. A vertical value is calculated for every horizontal time point between the acquired data points.

When the time per division is increased, waveforms are horizontally compressed and decimated. The minimum and maximum values are used as the endpoints of the vertical line that represents the data values for that time point. The cursor readout displays the value of the top endpoint.

To Use

You can select whether to display range readouts. You can also select the readout color. For each waveform, two readouts show the maximum and minimum vertical input voltage settings. The readouts may contain symbols indicating AC or Ground coupling, 50 Ohms input impedance, and the bandwidth limit. The ground line is indicated as a horizontal dotted line. The trigger threshold is indicated with a T and an up arrow if the threshold is above the maximum voltage or a down arrow if the threshold is below the minimum voltage.

Waveform size is dependent on the signal size and the input voltage range. The entire input voltage range is used to set the default minimum and maximum values of the vertical range. Set the vertical range values using the Waveform Options dialog box. You access it from the Waveform Properties dialog box by clicking the Waveform tab, and then clicking the Options button.

To display no more than one data sample per display point, add a sample waveform from the same data source and decrease the time per division until you can see individual clock tick marks.

Related Topics

[Waveform Options \(see page 216\)](#)

[DSO setup \(see page 395\)](#)

[Sample Waveforms \(see page 215\)](#)

Logic Thresholds

Logic Threshold sets the voltage level that determines whether input signals are high or low (1 or 0). You can select from a list of presets and Init @ 50%.

The list for each signal contains presets from Options, Presets, Logic Threshold. The default threshold value is the midpoint of the vertical range of the trigger signal. The minimum and maximum threshold values are the Vertical Offset +/- the Vertical Range of the trigger signal. The probe type, the vertical range, and the trigger coupling all affect the threshold value.

Init @ 50% sets the threshold value to the midpoint of the trigger signal.

Probe Calibration

Overview

The Probe Calibration dialog box controls all DSO probe calibration cycles and directs you to perform any necessary steps. You can calibrate all attached probes or only the probe on the selected channel.

You can start a calibration cycle, view the progress of calibration, and see the results of calibration. Once calibration starts, the calibration cycle of a single probe cannot be stopped. After a calibration cycle completes, you can see the status of probe calibration.

Behavior

Instructions direct you to attach probes to the calibration fixture and the input channel.

Channel status shows the current probe and its calibration status.

- Channel number shows the channel number of the probe.
- Probe Calibration Status shows the calibration status for attached probes.

Calibrate Chan starts probe calibration for the selected channel.

Calibrate All starts probe calibration for all channels that can be calibrated. To stop calibration when calibration for the current probe is complete, click Abort.

To Use

To open the Probe Calibration dialog box from the System window, click the DSO Setup icon, click a channel tab and then click the Probe Cal button.

Related Topic

[Calibrating Probes \(see page 59\)](#)

Index

Symbols and Numbers

\$ character, 166

63-Sample Blocks, 25

63-Sample Blocks mode, 162

A

about data, 196

accelerator keys, 382

Acquisition, 366

gaps, 205

memory status, 366

sample, 299

storage, 151

Acquisition mode, 25

Acquisition options, 25

Acquisition overview, 57

Active cursor readout, 232

Active module, 359

Activity, 366

Add Column/Waveform Dialog
Box, 187

Add Protocol Dialog Box, 288

Add Protocols, 291

Add Transition dialog box, 159

Add Word dialog box, 160

Adding, 314

data source to an active Listing
window, 73

data source to an active
Waveform window, 135

histogram data from another
source, 226

MagniVu data to a Listing or
Waveform window, 75

mark to a data window, 137

new graph, 314

Opcode Mark, 114

reference data to a Listing
window, 113

reference data to a waveform
window, 132

state to a trigger program, 93

waveform, 118

Advanced trigger, 149

Alignment of time, 195

Analog multiplexing, 55

assigning data signals via the
iConnect Analog Feeds
dialog box, 29

changing signal routing, 67
setting up, 55

signal routing from the logic
analyzer, 48

signal routing to the
oscilloscope, 49

specifying physical
connections, 51

Analog outputs, 29

analog routing, 331

Analog signal routing from the
logic analyzer, 48

analyze data, 225

Apply Deskew Analysis Results
dialog box, 303

Applying filters to data
windows, 35

Argument and command
entry, 379

Tool Property dialog box, 379

Arming, 375

a module using an LA
module, 154

interaction, 375

other modules from an LA
module, 103

ASN.1 Types, 252

additions to PSDL, 258

PSDL, 252

Asynchronous, 368

digital oversampling, 368

Asynchronous sampling, 14

Autoset

DSO, 392

serial analyzer, 45

average, 306

Axis Type Wizard, 64

B

Background color, 123

Background color for Listing
window, 197

Base signal, 18

Binary file format, 189

Bus support triggering, 333

Bus-specific framers, 273

Busform, 129

displaying LA data as a
busform, 216

selecting the radix of busform
readouts, 85

waveforms, 213

C

Calibrating, 59

modules, 59

probes, 59

Calibration, 371

DSO probes, 59

probe calibration (DSO
module), 400

self calibration

properties, 371

when using a three way
merge, 371

Change detect, 157

Changing

color, 197

data series range, 87

defaults, 86

graphs, 87

If/Then clauses, 93

position of cursors, 77

signal routing, 67

start-up defaults, 86

waveform properties, 120

window properties, 68

zoom options, 85

Channel Activity Indicators, 46

Channel change

triggering on, 101

- Channel settings
 - DSO, 393
 - Channel-lane assignment
 - serial analyzer, 46
 - chart, 307
 - Choosing system to open at startup, 369
 - Clause
 - defining If/Then or Else If/Then, 163
 - Clock, 391
 - fast sample clock
 - example, 391
 - serial analyzer, 46
 - signal out, 375
 - slow sample clock
 - example, 391
 - SUT, 46
 - clock warning, 338
 - Code loop, 241
 - COFF file format, 348
 - Collapse signals, 139
 - Color, 378
 - background, 123
 - change of marks, 197
 - graticule, 123
 - properties, 377
 - scheme for active data window, 122
 - selecting the active data window, 122
 - tab, 377
 - Coloring data, 118
 - Column, 196
 - Add Column/Waveform dialog box, 187
 - changing properties, 275
 - going to a specific column, 115
 - properties, 196
 - selection, 195
 - showing and hiding, 275
 - tab, 196
 - Columns in a Protocol window, 271
 - Comment Lines, 40
 - Compare
 - limitations, 190
 - Compare Definition Dialog Box, 190
 - Comparing data, 62
 - Compilation Result Pane, 272
 - Conditional storage, 150
 - Configuration, 378
 - mainframe, 372
 - software, 378
 - Configure, 4
 - Configuring external and internal signals, 375
 - Connect, 3
 - Connections, 3
 - Connectors, 375
 - mainframe, 375
 - Contacting Tektronix, 2
 - Control Lines, 41
 - Control statements, 247
 - PSDL, 247
 - Controlling whether a waveform is visible, 121
 - count measurement, 105
 - Counters, 97
 - problems, 104
 - triggering an LA module on counter values, 97
 - Create New Protocol Files, 293
 - Create New Stack Files, 294
 - Creating
 - custom color schemes, 60
 - overlay waveform, 135
 - symbol files, 115
 - cursor position, 240
 - cursor readout, 232
 - Cursors, 227
 - Cursors locked together in different data windows, 187
 - Custom color schemes, 60
 - Custom sampling, 334
 - Customer feedback, 2
 - Cycle marking, 196
- ## D
- Data, 132
 - compare, 190
 - determining the data source of the selected waveform, 84
 - examining the listing data, 11
 - font size, 197
 - incorrect after acquisition, 335
 - MagniVu, 368
 - measurements setup, 110
 - name, 363
 - points, 205
 - properties, 196
 - searching on selected data, 12
 - triggering on group data values, 100
 - value triggering, 100
 - viewing sample data, 11
 - when to store, 95
 - data area, 9
 - Data Samples, 219
 - data series, 313
 - data source, 225
 - determining, 84
 - measuring, 225
 - properties, 221
 - time alignment, 195
 - Data sources, 190
 - Data storage
 - serial analyzer, 46
 - Data Valid Window, 301
 - Data window
 - creating a New Data window, 279
 - locked, 187
 - New Data Window name, 363
 - System window, 359
 - Decimal values, 153
 - signed, 153
 - Decode View, 271
 - Default setup, 369
 - default system, 369
 - Defaults, 86
 - changing, 86
 - properties, 378
 - tab, 378

- Define Bus dialog box, 280
 - define compare, 190
 - define filters, 33
 - Define protocols, 292
 - Define Search Dialog Box, 193
 - Define Snapshot dialog box, 165
 - Define Suppression dialog box, 199
 - Defining, 268
 - compare parameters, 60
 - data to search for, 193
 - histogram ranges, 225
 - If/Then clauses in a trigger program, 93
 - multiple transition recognizers, 94
 - protocol with PSDL, 291
 - source data to search for, 82
 - Definition of MagniVu data, 368
 - Delete Protocol or Stack Dialog Box, 289
 - Delete Protocols, 292
 - Delimiter, 43
 - When exporting data from a Listing window, 71
 - Delta, 129
 - time measurements, 129
 - volts measurements, 128
 - waveform measurements, 129
 - Deskew Analysis, 295
 - Properties, 301
 - Setup, 299
 - Setups, 302
 - Window, 295
 - Deskew panel, 16
 - destination signal, 29
 - Detecting violations, 161
 - Determining acquisition status, 366
 - Developing custom color schemes, 60
 - Diagnostics, 337
 - Digital signal routing, 66
 - from a logic analyzer output, 66
 - to an oscilloscope input, 66
 - Digitizing a signal, 386
 - Disabling modules, 359
 - Disassembler, 114
 - adding opcodes, 114
 - Disassembly, 334
 - display formats, 334
 - properties, 198
 - selecting display formats, 334
 - tab, 198
 - using microprocessor and bus support packages, 333
 - Displaying
 - disassembled data, 198
 - DSO data, 141
 - glitches, 120
 - graticules, 124
 - magnitude, 125
 - marks in a data window, 138
 - violations in a waveform window, 122
 - waveform readouts, 120
 - waveforms, 392
 - Dragging marks, 137
 - Drawing Style dialog box, 311
 - DSO, 59
 - Autoset, 392
 - calibrating probes, 59
 - compressed waveform format, 392
 - cursors, 141
 - displaying DSO data, 141
 - expanded waveform format, 392
 - horizontal settings, 393
 - setting up, 395
 - storing DSO data, 141
 - trigger settings, 385
 - vertical, 393
 - waveforms, 399
 - DSO probe, 59
 - DSO trigger, 385
 - dual channel measurements, 112
 - Dwarf file format, 348
- E**
- EasyTrigger, 183
 - library, 183
 - list contents, 183
 - program list, 183
 - Edit Protocol Files, 68
 - Edit Relations Between Protocols, 69
 - Edit stack files, 69
 - Editing, 70
 - Editing a Tool on the Tools Menu, 70
 - editing symbol files, 70
 - Elf/Dwarf file format, 348
 - Elf/Stabs file format, 348
 - Enhanced column headers, 190
 - Entering a symbol plus an offset in a trigger, 175
 - Error Message Dialog Box, 199
 - Event target value limit, 105
 - event type, 169
 - Examining a waveform, 9
 - Example, 11
 - viewing a magnitude waveform, 11
 - Executing an operation after an acquisition, 58
 - Expand signals, 138
 - Export, 278
 - protocol data, 71
 - Export data, 188
 - Export Data dialog box, 188
 - Export Data Options Dialog Box, 190
 - Export Histogram dialog box, 224
 - Export Histogram Options dialog box, 224
 - Export Stack Dialog Box, 290
 - Exporting, 350
 - data from a Listing window, 71
 - group width and radix information, 190
 - histogram data, 72
 - protocols, 290
 - signal setups, 37
 - stacks, 290
 - symbol tables, 350
 - Extended Diagnostics, 337
 - External oscilloscope, 48
 - communication, 47
 - setup window, 47
 - External Signal Selection, 367

External signals, 146
EyeFinder, 295
EyeScan, 316

F

Feedback to Tektronix, 2
feeds, 29
File, 348
 format, 348
Filters, 284
 applying, 35
 copying, 112
 creating, 113
 dialog box, 32
 loading, 76
 opening, 113
 protocol, 284
Filters dialog box, 33
Force Main Prefill, 171
Format, 196
 column, 196
 file, 334
 listing window, 389
 TLA Data Exchange, 188
Formatting
 disassembled data, 198
 Listing window, 197
Framers, 273
Frames, 273
full screen, 9
Function range symbols, 343
Functional procedure, 337

G

Gaps, 205
 time, 205
General purpose data window
 shortcuts, 380
Generic Framer Example, 282
Generic framers, 273
 limitations, 273
 usage, 273
Getting out of a loop in a Source
 window, 241
Glitch
 detect (LA), 161
 glitch detection, 161

Glitches, 185
 in a Listing window, 115
 LA, 185
 triggering an LA module, 96
Global storage, 140
 note on using transitions, 140
Go to (moving to a mark), 76
Go To dialog box, 195
Go To Protocol Dialog Box, 292
Going to a specific column, 115
Going to a Specific Graph Window
 Mark, 315
Going to a specific mark, 138
Going to a Specific Series, 315
GPIB, 48
Graph Window, 307
 adding data series, 64
 creating, 312
 labeling, 87
 properties, 87
Graticule, 89
 selecting the color, 123
 showing, 89
graticule display, 373
Group
 color, 35
 data range triggering, 100
 effects on data source, 221
 radix, 166
 ranges, 104
 triggering an LA module, 102
Group cards, 28
Group definitions
 bus supports, 333
 microprocessor and bus
 support, 333
 microprocessor supports, 333
Group width information, 190
 exporting, 190
Grouping signals, 28
Guidelines for merging
 modules, 53

H

Hardware order, 153
Height, 88
Hex View, 271
High Speed timing, 388

High-Level Language
 support, 369
Highlight controls, 198
Highlight Signal dialog box, 28
Highlighting
 disassembled data, 198
 lines, 198
 signals, 28
Histogram, 225
 data page, 62
 defining ranges, 225
 exporting data, 72
 options page, 62
 selecting data source, 226
Histogram window, 221
 color selection, 220
 data area, 220
 font size, 220
Histogram add counts dialog
 box, 224
Histogram data source, 221
Histogram scale dialog
 box, 224
Histogram window
 shortcuts, 385
 property page, 220
 radix, 220
Hit Frequency Properties, 311
HLL support, 369
Horizontal
 scale (time per division), 142
 setting up the DSO, 126
 settings(DSO), 393
host name, 8
Hot keys, 384

I

iCapture, 29
iConnect Analog Feeds dialog
 box, 29
Identifying modules, 359
IEEE695 file format, 348
If/Then clause
 changing an, 93
import, 36
Import signal setups, 38
Import Stack Dialog Box, 290
Incoming inspections, 337

- Incorrect data after acquisition, 335
 - Individual waveform properties, 209
 - Individual waveform properties within an overlay waveform, 211
 - Information, 196
 - about current data, 196
 - about installed software, 378
 - about modules, 372
 - about the mainframe, 372
 - Information Lines, 43
 - Input
 - DSO settings, 393
 - Installed software information, 378
 - inter-probing, 331
 - intermodule interaction, 387
 - Intermodule signals, 375
 - Internal, 367
 - signal selection, 367
 - signaling, 154
 - Internal signals, 375
 - configuring, 375
 - triggering an LA module, 98
 - iVerify, 318
 - Setup dialog box, 320
 - Setups dialog box, 318
 - Statistical Analysis dialog box, 327
 - iVerify measurement properties, 322
 - iVerify Properties, 322
 - About Data tab, 322
 - Eye Coloration tab, 325
 - iVerify Window Tab, 323
 - iVerify Properties dialog box, 322
 - Eye coloration, 322
 - iVerify Setup dialog box Channel tab, 329
 - iVerify Setup dialog box Clock/Qualifier tab, 330
 - iVerify window, 316
 - iView external oscilloscope setup window, 47
- J**
- jitter, 107
 - Jumping over "dead" space between samples, 187
- K**
- Key, 383
 - shortcut, 383
 - Keystroke Entry field, 28
- L**
- LA
 - module memory depth, 27
 - module setups, 54
 - trigger actions table, 168
 - trigger events, 167
 - trigger resources, 169
 - triggers, 145
 - waveforms, 213
 - LA Busform, 129
 - LA channels
 - selecting for an LA trigger channel event, 95
 - LA module
 - arming, 103
 - merging, 53
 - LA Probe Signals/Names table, 18
 - LA Setup window, 13
 - LA versus DSO module waveforms, 391
 - Labeling a graph, 87
 - Labels, 216
 - LAN, 5
 - Language, 369
 - high-level language (HLL) support, 369
 - Length and Value Expressions, 261
 - PSDL, 261
 - Lexical Items, 265
 - PSDL, 265
 - Limits, 216
 - Linked cursors, 187
 - Listing data concepts, 389
 - Listing window, 383
 - adding a data source, 73
 - adding reference data, 113
 - properties, 197
 - showing glitches, 115
 - showing qualification gaps, 114
 - tab, 197
 - use, 185
 - Listing-relative cursor positioning, 241
 - Listing/Waveform data page, 362
 - Load, 333
 - support package, 333
 - system options, 363
 - Load Deskew Analysis Setup dialog box, 298
 - Load iVerify Setup dialog box, 319
 - Load Symbols Options dialog box, 348
 - Loading
 - a MagniVu data source to a Listing or Waveform window, 75
 - a saved module, 73
 - a saved system, 72
 - an LA module with saved trigger settings, 74
 - filters, 34
 - merged modules, 363
 - part of a saved system, 363
 - portions of a saved system, 73
 - saved data to an active Listing window, 73
 - saved data to an active Waveform window, 135
 - source code symbol information, 75
 - the timing format file, 334
 - Lock delta time, 129
 - Lock windows, 187
 - Lock Windows dialog box, 187
 - Locking windows, 136
 - Logic analyzer, 131
 - measurements, 131
 - trigger window, 143
 - Logic thresholds, 378

- Logic thresholds for a DSO, 400
- Loop, 241
- Loop statements, 248
 - PSDL, 248
- Lower threshold, 378

M

- Magnitude, 216
- Magnitude mode, 390
- Magnitude waveform
 - viewing, 11
- MagniVu, 168
 - data, 368
 - data source to a Listing or Waveform window, 75
 - LA trigger action table, 168
 - signal resolution/signal duration relationship, 392
 - storage rate, 172
 - trigger position, 173
- Main window, 359
- Mainframe, 372
 - properties, 372
- Making graticules visible, 124
- Making violations visible in a Waveform window, 122
- Mark, 196
 - next and previous, 231
 - opcode, 196
 - selection, 314
- Mark Opcode dialog box, 196
- Marking cycles, 196
- Marks, 114
 - properties, 197
 - tab, 197
- Marks in a data window, 137
 - adding, 137
- Marks in a Protocol window, 271
- max, 111
- Measurement Setup, 305
- Measurements, 126
 - count, 105
 - data, 305
 - jitter, 107
 - min/max, 108
 - multiple, 129
 - on DSO data, 396
 - on LA data, 305
 - time, 109
- Measuring, 127
 - delta time, 129
 - delta volts, 128
 - histogram data, 225
 - time, 129
 - voltage, 127
- Memory, 90
 - selecting depth, 90
 - storing DSO data, 141
- Memory depth, 90
 - for an LA module, 90
 - LA, 27
 - most effective use of, 90
- Menu shortcut keys, 384
- Merged module synchronous sampling, 27
- Merged modules, 372
 - loading, 363
 - note on calibration, 371
 - properties, 372
- Merging LA modules, 53
- Meta-characters, 193
- Microprocessor and bus support
 - package basic operation, 333
- Microprocessor support, 333
 - triggering, 333
- min, 108
- Miscellaneous PSDL
 - commands, 262
- Modify, 377
 - color schemes, 377
 - location of cursors using the overview mark bar, 76
 - search path list, 233
 - suffix list, 233

- Module, 372
 - active, 359
 - arming using an LA module, 154
 - disabling, 359
 - identifying, 359
 - merged, 372
- Module specifics properties, 372
- module trigger, 146
- Modules, 59
 - calibrating, 59
 - merging, 53
 - properties, 372
 - triggering from an LA, 103
- More waveforms, 118
- Moving
 - cursors, 76
 - marks, 137
 - to a mark (Go To), 76
 - to a specific column, 115
- Multi-signal
 - busform LA waveforms, 213
 - magnitude LA waveforms, 214
 - waveforms, 213
- Multiple acquisitions, 58
- Multiple Samples per Clock Trigger Programs, 173

N

- Name
 - waveform, 216
- Name Data Window dialog box, 363
- network, 6
- Network preferences, 5
- network properties, 8
- Next Mark and Previous Mark buttons, 231

O

- Object file format, 348
- Object selection, 206
- offline, 3
- Offset, 378
 - entering a symbol plus an offset in a LA trigger, 175

- OMF166 file format, 348
 - OMF286 file format, 348
 - OMF386 file format, 348
 - OMF51 file format, 348
 - OMF86 file format, 348
 - On Run control
 - serial analyzer trigger window, 177
 - Opcode, 196
 - Opcodes, 114
 - adding, 114
 - Open dialog box (repetitive properties), 193
 - Opening, 58
 - a document after an acquisition, 58
 - a document or program after an acquisition, 58
 - a saved data window, 135
 - Operators, 245
 - PSDL, 245
 - Options, 81
 - collapse signals, 139
 - expand signals, 138
 - powering on and off, 9
 - setting waveform options, 125
 - Organizing signal groups, 22
 - outputs, 29
 - Overlapping symbols, 348
 - Overlay waveform, 135
 - properties, 208
 - setting the offset, 209
 - Overview mark bar, 76
- P**
- Packet structure definition, 245
 - PSDL, 245
 - Packet Structure Description Language, 243
 - Packet View, 275
 - column properties, 275
 - Pattern Generator, 88
 - Run Properties dialog box, 88
 - Pattern symbol, 117
 - file header, 117
 - files, 343
 - PG Run Properties dialog box, 88
 - Placement of system triggers, 148
 - Placing marks in a data window, 137
 - Polarity, 88
 - of LA signals, 18
 - population, 306
 - Positioning the trigger, 95
 - Postscript, 77
 - Power on and off options, 9
 - Power-On Diagnostics, 370
 - PowerTrigger, 159
 - detail, 149
 - properties, 149
 - tab, 149
 - tab overview, 149
 - troubleshoot, 104
 - Preferences properties, 373
 - Preprogrammed LA triggers, 183
 - Presets, 378
 - Presets tab, 378
 - Prime signal, 18
 - Print dialog, 77
 - Printing, 188
 - Probe
 - calibration (DSO), 400
 - signal grouping, 28
 - status (DSO), 400
 - problems, 335
 - sampling, 335
 - triggering, 175
 - programming tips for triggering an la module, 151
 - properties, 88
 - column, 196
 - graph window, 87
 - network, 8
 - protocol, 285
 - protocol display, 275
 - protocol framing, 277
 - protocol, 293
 - create new, 279
 - exporting, 290
 - importing, 290
 - saving, 289
 - Protocol Designer window, 286
 - Protocol Editor pane, 287
 - Protocol Filter Setup Dialog Box, 274
 - Protocol information pane, 281
 - Protocol properties, 277
 - protocol relations, 291
 - Protocol Search Dialog Box, 278
 - Protocol Setup window, 285
 - Protocol stacks, 289
 - importing, 290
 - Protocol window, 271
 - Define Bus dialog box, 280
 - frames and framers, 273
 - generic framer example, 282
 - properties, 275
 - Select Bus page, 280
 - Select Stack dialog box, 282
 - PSDL, 263
 - ASN.1 additions, 258
 - ASN.1 types, 252
 - control statements, 247
 - defining a protocol, 268
 - length and value
 - expressions, 261
 - lexical items, 265
 - loop statements, 248
 - miscellaneous
 - commands, 262
 - operators, 245
 - overview, 266
 - packet structure, 245
 - reserved words, 263
 - sample program, 249
 - spz file extensions, 266
 - syntax, 243
 - tags, 262
 - type definitions, 250
 - types, 258
 - value definitions, 261
- Q**
- Qualification gaps, 185
 - showing, 197
 - showing in a Listing window, 114
- R**
- Radix, 166
 - Radix of column, 196

- Radix information, 190
 - exporting, 190
- Range, 100
 - recognition problems, 104
 - recognition triggering, 100
 - recognizers, 153
 - recognizers and hardware order, 153
 - symbol file header, 117
 - symbol files, 342
- range recognizers, 151
- Range symbols, 344
 - Function, 343
 - Source, 345
 - Variable, 344
- Ranges properties, 222
- Readout radix, 85
- Readouts, 216
 - displaying, 120
- Reference, 386
 - digitizing a signal, 386
 - sampling a signal, 386
- Reference clock
 - serial analyzer, 46
- Reference data, 132
 - adding to a Listing window, 113
- Reference system trigger, 165
- Release notes, 1
- remote, 3
- Removing
 - data from a Listing or Waveform window, 139
 - LA signals, 18
 - selected DSO waveform measurements, 127
 - waveforms from an overlay waveform, 139
- Removing a Tool from the Tools Menu, 70
- Renaming, 359
 - a window, 359
- Repeating acquisitions, 58
- Repetitive acquisition data, 79
 - saving, 79
- repetitive acquisition mode, 57
- Repetitive mod, 57
- Repetitive operations, 193
- Repetitive Properties Dialog Box, 191
- Replace Protocol Dialog Box, 288
- Replace protocol files, 294
- Reserved words, 263
 - PSDL, 263
- Routing a signal from a logic analyzer output, 66
- Routing a signal to an oscilloscope input, 66
- Rules for merging modules, 53
- Running, 133
 - extended diagnostics, 133
 - multiple acquisitions, 58
- S**
- Sample
 - suppression, 90
 - waveforms, 215
- Sample Signal setup file, 38
- Sample Suppress dialog box, 200
- Samples + Glitches, 25
- Samples + Violations, 25
- Sampling, 335
 - a signal, 386
 - asynchronous, 14
 - custom, 334
 - LA setup window, 13
 - merged module external, 27
 - problems, 335
 - resolution, 391
 - support package, 334
 - synchronous, 14
- Save
 - color scheme, 378
 - dialog box, 78
 - module, 78
 - system, 78
 - trigger program, 386
- Saved
 - data, 73
 - module, 73
 - system, 363
 - system timing format file, 334
 - trigger, 74
- Saved setups, 53
 - loading to an LA module, 53
- Saving
 - a module, 79
 - a system, 78
 - data, 79
 - LA setups, 54
 - protocols, 289
 - repetitive acquisition data, 79
 - repetitive module setups and data, 80
 - repetitive system setups and data, 80
 - stacks, 290
- Scale, 224
 - horizontal, 142
 - in Histogram window, 224
- Scan Listing, 230
 - dialog box, 234
 - mode, 230
- Scan listing direction, 234
- Scrolling, 126
 - between samples in a Waveform window, 126
 - through source data, 229
- search, 6
- Search Definition, 195
- search options, 193
- Search path, 235
- Search Progress dialog box, 234
- Search protocol display, 83
- Searching
 - for signal names, 28
 - for source data, 82
 - in data windows, 82
 - on selected data, 12
- Seconds per division, 142
- Seeing
 - glitches in a Listing window, 115
 - qualification gaps, 114
- Select Axis Type, 63
- Select Bus Page, 280
- Select Signal dialog box, 157
- Select Stack Dialog Box, 282
- Select Stack Page, 282
- Selected waveform, 84

- Selecting
 - background color, 123
 - channels for an LA trigger
 - channel event, 95
 - disassembled data, 198
 - disassembly display
 - formats, 334
 - graticule color, 123
 - objects in a Listing window, 206
 - radix of busform readouts, 85
 - the active data window
 - color, 122
 - violation color, 122
 - waveform color, 119
 - when to store data, 95
- Selecting memory depth, 90
 - a general approach, 90
- Selecting Stacks, 283
- selecting trigger
 - external oscilloscope, 131
- Selection dialog box, 368
- Self calibration, 371
 - note on 3-way merge, 371
- Sending stacked marks to the back, 137
- Serial analyzer
 - Ack button, 44
 - activity indicators, 44
 - Autoset, 45
 - Channel-lane assignment, 46
 - Data storage, 46
 - left side controls, 45
 - link status, 44
 - On Run control, 177
 - Reference clock, 46
 - setup window, 43
 - Status area, 44
 - trigger events, 178
 - trigger tool bar, 177
 - trigger window, 177
- Server, 5
- Set thresholds, 13
- Setting
 - a trigger from Source window data, 242
 - default trigger tab, 86
 - overlay waveform offset, 209
 - system trigger source, 148
 - trigger position, 95
 - waveform limits, 84
 - waveform options, 125
 - waveform style, 83
- Setting up
 - analog feeds, 64
 - analog multiplexing, 55
 - data measurements, 110
 - Protocol window, 283
 - system inter-probing, 65
 - the DSO, 392
 - transitional storage for the LA module, 91
 - waveform measurements, 126
- Setting up external oscilloscope communication, 47
- Setup
 - for saving repetitive module setups and data, 80
 - for saving repetitive system setups and data, 80
 - Measurement, 396
- Setup and hold, 91
 - storage, 91
- Setup and Hold Storage dialog box, 91
- Setup and hold triggering, 92
- Setup and hold violation, 161
 - triggering an LA, 92
- Setup Properties, 285
- Setup window, 47
 - iView external oscilloscope, 47
 - logic analyzer, 13
 - serial analyzer, 43
- Setups, 72
- Setups for an LA module
 - loading, 53
 - saving, 54
- Shortcut keys, 382
- Show Samples Dialog Box, 199
- Show waveform, 88
- Showing glitches in a Listing window, 115
- Showing qualification gaps in a Listing window, 114
- Signal, 392
 - base, 18
 - duration, 392
 - prime, 18
 - resolution, 392
- Signal grouping table, 22
- Signal Import and Export Formats, 36
- Signal names for an LA, 18
- Signal polarity (LA module), 18
- Signal routing, 48
 - from the logic analyzer, 48
 - to the oscilloscope, 49
- Signal routing to the oscilloscope, 49
- Signal setups
 - exporting, 37
 - import behavior, 36
 - importing, 38
- Signals
 - selecting, 95
- Signals in and out, 154
- Signals properties, 375
- Signed, 216
 - data, 216
 - decimal values, 153
- Single channel
 - triggering on a, 92
- single run acquisition, 57
- Single-run mode, 57
- Single-signal LA waveforms, 215
- Snapshot event, 99
 - triggering on an LA, 99
- Software, 378
 - properties, 378
- Source, 64
 - data page, 64
 - range symbols, 345
- Source code, 75
- Source code loop, 241
- Source files, 233
- Source options page, 64
- source signals, 29

- source statement, 229
 - source symbols, 342
 - Source window, 236
 - Data properties, 239
 - Marks properties, 240
 - properties, 236
 - shortcuts, 384
 - Source File Locator dialog box, 235
 - Source Files Edit Dialog Box, 233
 - Source Files properties, 237
 - Source Go To dialog box, 234
 - Source Window Search dialog box, 234
 - Source-relative cursor positioning, 240
 - Specify groups, 13
 - Specifying
 - application defaults, 378
 - application preferences, 373
 - multiple acquisitions, 58
 - stop conditions for compare operations, 62
 - Stack, 294
 - create new, 294
 - Stack definition, 281
 - Stack Editor Display Properties, 286
 - Stack protocol information, 281
 - Stacks, 290
 - exporting, 290
 - importing, 290
 - saving, 290
 - selecting, 283
 - standard deviation, 306
 - Start-Up, 86
 - defaults, 86
 - properties, 369
 - Starting, 9
 - the instrument without the TLA application, 9
 - State, 93
 - adding to a trigger program, 93
 - statistic, 111
 - Statistical Analysis, 306
 - Status, 400
 - Status area
 - serial analyzer, 44
 - Status bits, 386
 - Status Monitor dialog box, 366
 - Step, 231
 - backwards, 231
 - forward, 231
 - Stopping
 - an acquisition after compare definitions are met, 62
 - Storage
 - qualification, 140
 - selection for an LA module, 140
 - setup, 140
 - Storage Definition Dialog Box, 92
 - transitional, 140
 - trigger position (LA), 151
 - Storing, 91
 - DSO data in memory, 141
 - LA module setup and hold violations, 91
 - Structure of an LA trigger program, 181
 - subnet, 8
 - Support package sampling, 334
 - Suppress Samples Dialog Box, 200
 - Suppression, 199
 - define dialog box, 199
 - SUT
 - clock, 46
 - Symbol
 - entering a symbol plus an offset in a LA trigger, 175
 - Symbol file header, 117
 - Pattern symbol file header, 117
 - Range symbol file header, 117
 - Symbol files, 70
 - creating, 115
 - editing, 70
 - Load Symbols Options dialog box, 348
 - pattern, 343
 - range, 342
 - Symbol value width, 342
 - Symbols dialog box, 346
 - symbol name, 345
 - Symbol table triggering, 333
 - Symbol value width, 342
 - Symbols, 166
 - Synchronous sampling, 14
 - Compound clock, 15
 - merged modules, 27
 - single clock, 15
 - Syntax, 243
 - PSDL, 243
 - System, 165
 - trigger and reference system trigger relationship, 165
 - trigger placement, 148
 - window, 359
 - System configuration, 375
 - System Inter-probing, 51
 - System options, 363
 - System tools, 134
 - customizing, 134
 - System trigger, 146
- ## T
- Tags, 262
 - PSDL, 262
 - Taking Cursor Measurements, 315
 - Taking measurements, 131
 - delta time, 129
 - delta volts, 128
 - logic analyzer, 131
 - multiple, 129
 - time, 129
 - voltage, 127
 - Target value limit, 105
 - TekLink cable, 4
 - Tektronix
 - customer feedback, 2
 - Text color of column, 196

- Threshold, 400
 - logic thresholds for a DSO module, 400
 - presets properties, 378
- Time, 195
 - Alignment dialog box, 195
 - alignment of correlated data sources, 136
 - gaps, 205
 - measurement, 129
 - per division, 142
 - scale (time/division), 120
- time correlation, 387
- time measurements, 305
- time stamp, 390
- Timer event target value limit, 105
- Timers, 97
 - testing, 97
 - triggering an LA, 97
- Timestamp, 196
 - column, 239
 - values, 196
- Timing analysis using support packages, 333
- Timing format file, 334
- Timing zoom (see MagniVu), 172
- TL708EX, 4
- TLA, 189
 - binary file format, 189
- TLA Data Exchange format, 188
- TLA Server, 5
- TLA Symbol File Format, 351
 - TSF Color Symbols, 354
 - TSF Function Symbols, 356
 - TSF Headers, 351
 - TSF Pattern Symbols, 352
 - TSF Range Symbols, 353
 - TSF Source Symbols, 356
 - TSF Variable Symbols, 359
- TLA5000B series, 1
- TLA7000 series, 1
- Tools properties, 379
- Tools Properties dialog box, 379
- Tracing the execution of source code, 241
- Transition, 158
 - adding, 159
 - indication, 104
 - recognizer, 91
 - word definition, 158
- Transition Definition dialog box, 158
- Transition recognizer, 94
 - defining multiples, 94
- Transitional storage, 150
- Transitions as events
 - workaround, 93
- Transitions note, 93
- Trigger
 - action table, 168
 - actions, 143
 - all modules, 369
 - clauses, 149
 - defining If/Then or Else
 - If/Then clause, 163
 - entering a symbol plus an offset in a LA trigger, 175
 - event table, 167
 - list contents, 183
 - loading an LA with saved settings, 74
 - placement, 148
 - position, 151
 - positioning in acquisition
 - memory, 95
 - programming tips, 151
 - reference, 165
 - serial analyzer trigger events, 178
 - setting the default tab, 86
 - setting the system source, 148
 - settings, 385
 - system, 146
 - system and reference
 - relationship, 165
 - window, 143
 - word definition, 159
- Trigger action, 155
 - LA, 168
 - signal restrictions, 155
- Trigger event, 167
- Trigger on a Glitch, 163
- Trigger position
 - for an LA module, 151
- Trigger program, 181
 - adding a state, 93
 - defining If/Then clauses, 93
 - Multiple samples per
 - clock, 173
 - setting up, 92
 - structure, 181
 - troubleshooting, 104
- Trigger program structure, 181
- Trigger resources (LA), 169
- Trigger Save dialog box, 386
- Trigger type, 143
- Trigger window, 173
 - Force Main Prefill check box, 171
 - MagniVu storage rate, 172
 - MagniVu Trigger Pos, 173
 - serial analyzer, 177
- Triggering, 333
 - an LA module, 92
 - bus supports, 333
 - microprocessor supports, 333
 - on group data ranges, 100
 - on word recognizers, 100
 - other modules from an
 - LA, 103
 - problems, 175
 - the entire system, 146
 - using symbol tables, 333
- Triggering an LA, 99
 - on a setup and hold violation, 96
 - on a snapshot event, 99
 - on a word event, 99
 - on timer values, 97
- Triggering an LA module, 92
 - on a glitch, 96
 - on a group, 102
 - on a single channel, 101
 - on an internal signal, 98
 - on counter values, 97
- Troubleshooting, 104
 - external oscilloscope, 48
 - hardware, 337
 - LA trigger programs, 104
- Troubleshooting triggering, 338

- Type Definitions, 250
 - PSDL, 250
- Types, 258
 - PSDL, 258
- Types of LA triggers, 145

U

- Upper threshold, 378
- user preferences, 373
- Using, 196
 - a system trigger, 103
 - parts of a saved system, 73
 - polarity, 196
 - support packages for timing analysis, 333

V

- Value definitions, 261
 - PSDL, 261
- Variable range symbols, 344
- Vertical
 - input settings (DSO), 393
 - range, 327
- Viewing, 284
 - compare data, 60
 - LA waveform in Magnitude mode, 125
 - protocol data, 284
 - saved data, 135
- Viewing decoded data in a Protocol Window, 284
- Viewing persistence data, 316
- Violation color, 122
- Violations, 89
 - for a waveform, 89
- Voltage, 127
 - measurements, 127

W

- Waveform
 - adding, 118
 - appearance, 119
 - color, 119
 - creating an overlay
 - waveform, 135
 - displaying, 392
 - DSO compressed format, 392
 - DSO expanded format, 392
 - LA compressed format, 392
 - LA expanded format, 392
 - LA versus DSO module, 391
 - labels, 216
 - limits, 84
 - names, 216
 - options, 216
 - overlay, 209
 - properties, 88
 - readouts, 120
 - removing from an overlay
 - waveform, 139
 - Removing selected DSO measurements, 127
 - selection, 195
 - setting options, 125
 - setting to overlay waveform offset, 209
 - setting up measurements, 126
 - size, 124
 - style, 83
 - tab, 88
 - time scale, 120
 - viewing other types, 10
- Waveform data concepts, 390
- Waveform properties, 209
- Waveform window, 203
- Width of column, 196

- Window, 359
 - color scheme for active data window, 122
 - data, 359
 - locking windows
 - together, 136
 - properties, 120
 - renaming, 359
 - system window, 359
- Word definition dialog box, 159
- Word event, 99
 - triggering on an LA, 99
- Word recognition triggering, 100

X

- X character, 166

Z

- Zoom options, 85
- Zooming a graph, 315
- Zooming a Graph, 85
- Zooming a waveform, 85