Keysight U4431A MIPI M-PHY Protocol Analyzer

User Guide

KEYSIGHT TECHNOLOGIES

Notices

© Keysight Technologies 2014-2015

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

Manual Part Number

U4431-97002

Edition

May 2015

Available in electronic format only

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

Warranty

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

Technology Licenses

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

U.S. Government Rights

The Software is "commercial computer software," as defined by Federal Acquisition Regulation ("FAR") 2.101. Pursuant to FAR 12.212 and 27.405-3 and Department of Defense FAR Supplement ("DFARS") 227.7202, the U.S. government acquires commercial computer software under the same terms by which the software is customarily provided to the public. Accordingly, Keysight provides the Software to U.S. government customers under its standard commercial license, which is embodied in its End User License Agreement (EULA), a copy of which can be found at http://www.keysight.com/find/sweula. The license set forth in the EULA represents the exclusive authority by which the U.S. government may use, modify, distribute, or disclose the Software. The EULA and the license set forth therein, does not require or permit, among other things, that Keysight: (1) Furnish technical information related to commercial computer software or commercial computer software documentation that is not customarily provided to the public; or (2) Relinquish to, or otherwise provide, the government rights in excess of these rights customarily provided to the public to use, modify, reproduce, release, perform, display, or disclose commercial computer software or commercial computer software documentation. No additional government requirements beyond those set forth in the EULA shall apply, except to the extent that those terms, rights, or licenses are explicitly required from all providers of commercial computer software pursuant to the FAR and the DFARS and are set forth specifically in writing elsewhere in the EULA. Keysight shall be under no obligation to update, revise or otherwise modify the Software. With respect to any technical data as defined by FAR 2.101, pursuant to FAR 12.211 and 27.404.2 and DFARS 227.7102, the U.S. government acquires no greater than Limited Rights as defined in FAR 27.401 or DFAR 227.7103-5 (c), as applicable in any technical data.

Safety Notices

CAUTION

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

WARNING

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

Safety Summary

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

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

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

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

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

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

Do Not Remove the Instrument Cover Only qualified, service-trained personnel who are aware of the hazards involved should remove instrument covers. Always disconnect the power cable and any external circuits before removing the instrument cover.

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

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

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

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

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

Safety Symbols

Table 1 Safety Symbol

Symbol	Description
	Direct current
	Alternating current
	Both direct and alternating current
	Three phase alternating current
	Earth ground terminal
	Protective earth ground terminal
	Frame or chassis ground terminal
	Terminal is at earth potential
	Equipotentiality
N	Neutral conductor on permanently installed equipment
L	Line conductor on permanently installed equipment
	On (mains supply)
	Off (mains supply)
	Standby (mains supply). The instrument is not completely disconnected from the mains supply when the power switch is in the standby position
	In position of a bi-stable push switch
	Out position of a bi-stable push switch

Symbol	Description
	Equipment protected throughout by DOUBLE INSULATION or REINFORCED INSULATION
	Caution, refer to accompanying documentation
	Caution, risk of electric shock
	Do not apply around or remove from HAZARDOUS LIVE conductors
	Application around and removal from HAZARDOUS LIVE conductors is permitted
	Caution, hot surface
	Ionizing radiation
CAT I	IEC Measurement Category I
CAT II	Measurement Category II
CAT III	Measurement Category III
CAT IV	Measurement Category IV

Compliance and Environmental Information

Table 2 Compliance and Environmental Information

Safety Symbol	Description
	CSA is the Canadian certification mark to demonstrate compliance with the Safety requirements.
	The C-tick mark is a registered trademark of the Spectrum Management Agency of Australia. This signifies compliance with the Australia EMC Framework regulations under the terms of the Radio Communication Act of 1992.
	CE compliance marking to the EU Safety and EMC Directives. ISM GRP-1A classification according to the international EMC standard. ICES/NMB-001 compliance marking to the Canadian EMC standard.
	KC certification mark to demonstrate compliance with the South Korean EMC requirements. **South Korean Class A EMC declaration** This equipment is Class A suitable for professional use and is for use in electromagnetic environments outside of the home.

Table 3 Environmental Information

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

Contents

1 Introduction

The U4431A MIPI M-PHY Analyzer module is an acquisition module that captures and decodes data as per MIPI M-PHY standards including UniPro, UFS, SSIC, MPCIe, and CSI-3. Using this module, you can get insight into the M-PHY protocol stack.

It allows you to test and troubleshoot the data link, network, transport, and application layers of the M-PHY protocol stack.

You can use multiple U4431A modules in a chassis to test and debug multi-bus systems. You can also time-correlate M-PHY busses with MIPI D-PHY CSI-2 and DSI-1, MPCIe, DDR and HDMI busses by using multiple Keysight AXIe based modules in a time-correlated setup.

For the display and analysis of the captured data, the U4431A module presents the data in a Protocol Viewer, Waveform Viewer, and a Listing Data View.

This module needs to be installed in a Keysight AXIe chassis (for example, the M9502A 2 slot chassis). When a controller PC is connected to the AXIe chassis via an external PCIe / USB interface and a cable, the Keysight Logic and Protocol Analyzer application (running on the controller PC) lets you configure, control, and use the U4431A module.

NOTE

You can also use the U4431A module for transmitting stimulus and capturing bidirectional data, that is, capturing the data that it transmits as stimulus as well as the data that it receives from a DUT. For this usage scenario, you need the **Keysight Command Line Packet Generator (CLPG)**, which is available as a licensed software option (U4431A-613) of the U4431A module,

For detailed information on CLPG and how to use it for stimulus transmission and capture, refer to the ***Keysight MIPI M-PHY Command Line Packet Generator User Guide*** installed with the CLPG software at the following location.

C:\Program Files\Keysight Technologies\Logic Analyzer\Help\pdfs

Protocols Supported

The U4431A module currently supports capture and decode for the following protocols:

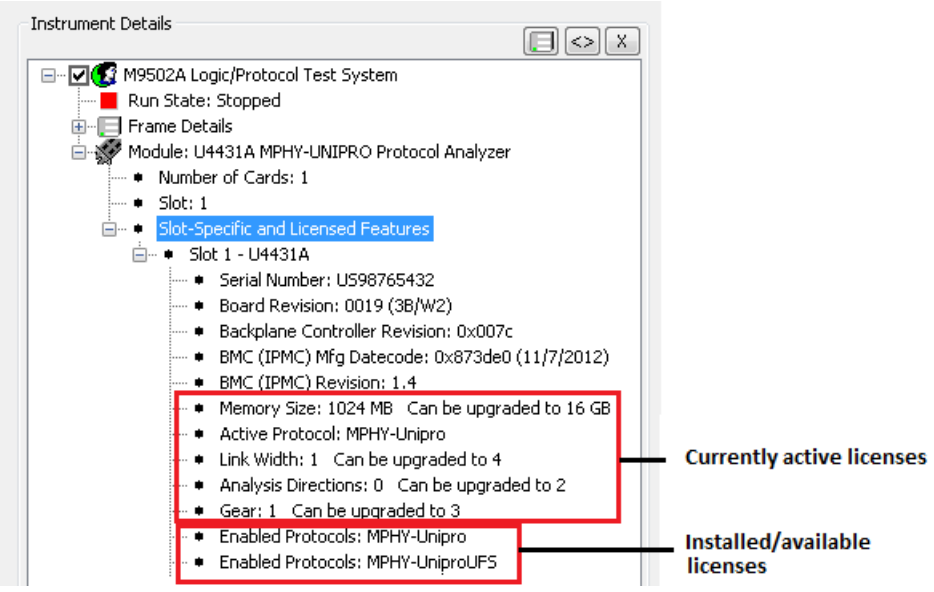
- M-PHY 2.0
- Unified Protocol (UniPro) 1.41.00 and 1.6
- Universal Flash Storage (UFS) v1.1
- SSIC
- CSI-3
- MPCIe

By default, the base protocol supported on the U4431A module is M-PHY. As and when needed, you can add other supported protocol capabilities to your U4431A module by purchasing and installing appropriate U4431A licenses. Refer to the U4431A module data sheet (part number 5991-2544EN.pdf) available at www.keysight.com/find/mphy_analyzer to know about various licensed options available for the module.

NOTE

You can view which licenses are currently active/in use on your U4431A module and which licensed options are installed/available for use. For this, you can use the Keysight Notification Center icon. This icon is displayed as a green dot on the taskbar (lower, right-hand corner of your desktop) if the Keysight Logic and Protocol Analyzer GUI is installed. If the GUI is open and running, the Notification Center icon changes to .

1. Right-click the Notification Center icon and select **Show Instrument Details** from the displayed menu.
2. In the **Instrument Details** dialog box, expand the Module: U4431A Protocol Analyzer node.
3. Note the currently active and available licensed options.



U4431A Features

A U4431A module:

- Can acquire protocol level data as well as raw signal level data.
- Can transmit stimulus to a DUT and also capture the DUT data if the licensed software option Command Line Packet Generator (CLPG) is installed.
- Provides upto 16 GB of memory that you can divide and allocate for capturing raw signal level and protocol level data.
- Supports preempting of UniPro data.
- Supports noise filtering for PWM gears.
- Syncs up to the DUT's link configurations based on the settings that you define.
- Can track DUT's link configurations automatically.
- Supports descrambling of scrambled data for SSIC data acquisition.
- Provides a number of viewers to view the acquired data.
- Allows you to set the following attributes independently for the Transmit and Receive sublinks.
  - Link width
    - x1, x2, x3 and x4 for UniPro/UFS/CSI-3
    - x1, x2, and x4 for SSIC (Same link width is applicable for both sublinks in case of SSIC.)
    - x1, x2, and x4 for MPCIe
  - Burst mode - PWM or HS Gear per sublink.
    - PWM Gear 1 to 7 and HS Gear 1, 2, and 3 for UniPro/UFS/CSI-3
    - PWM Gear 1 and HS Gear 1, 2, and 3 for SSIC and MPCIe
- Sublink based enabling and disabling of raw signal level and protocol level (UniPro, CSI-3, SSIC, MPCIe, and UFS) data capture.
- Sublink based division and allocation of memory between raw and protocol level data acquisition.
- Allows you to set the following attributes at the link level (applicable for both sublinks of the link).
  - Data rate series applicable for the link in the High Speed (HS-Mode).
  - Link Synchronization
  - HS Gear profile (only for SSIC). Same HS Gear is used for both sublinks in case of bidirectional link configuration.
  - Link width profile (applicable for SSIC and MPCIe). Same link width is used for both sublinks in case of bidirectional link configuration.
  - PWM Filtering (for UniPro)

NOTE

The U4431A module allows cross-triggering with an external oscilloscope and making time-correlated measurements (using markers).

You can display data captured by the oscilloscope in the Waveform and Listing display windows. This external oscilloscope correlation and data display feature is also referred to as View Scope. To know more about this feature and how to use it with the U4431A Analyzer, refer to the External Oscilloscope Time Correlation and Data Display Online Help.

Related Documents

Besides using this U4431A User Guide, you can also access the following documents that provide related information about the U4431A module.

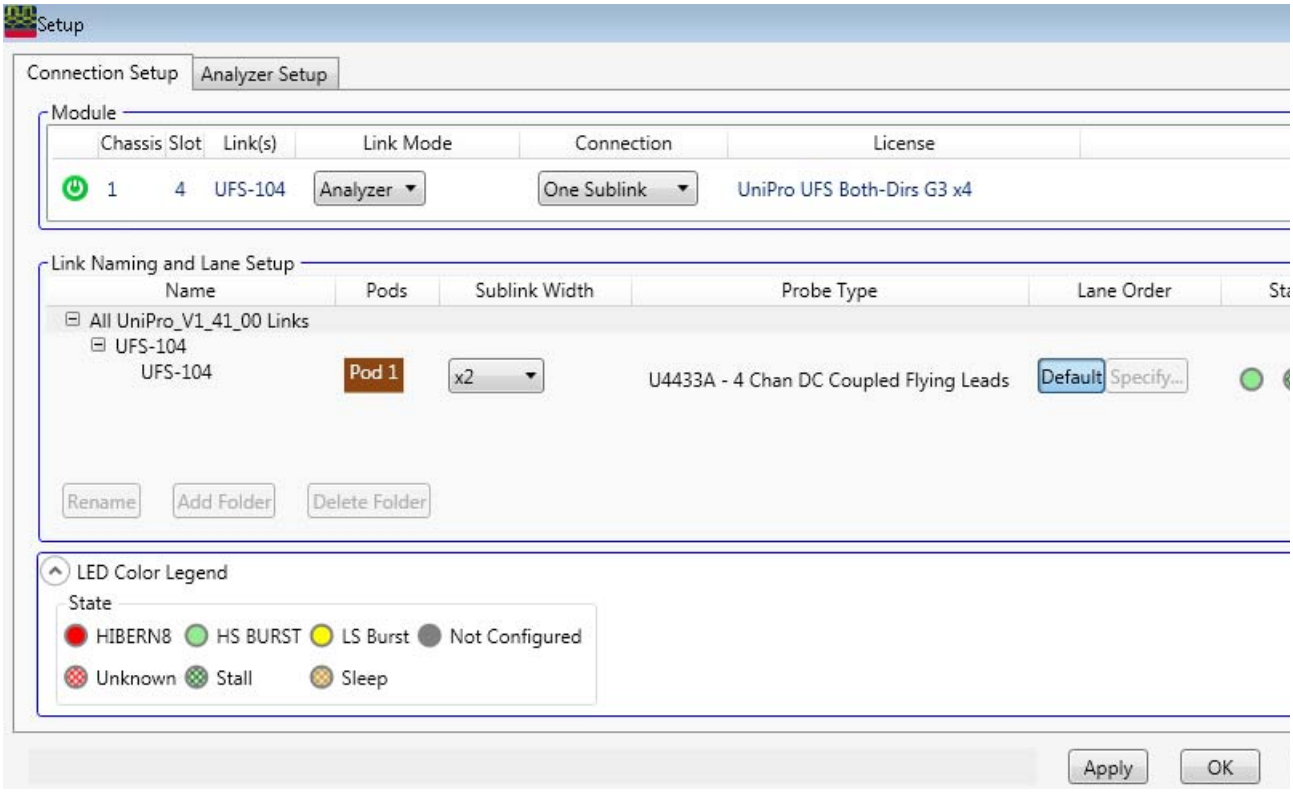
- **MIPI M-PHY Protocol Analyzer Hardware and Probing Guide** - This guide provides information on the acquisition probing options available for use with the U4431A module. This guide is available on www.keysight.com/find/mphy_analyzer and is also installed with the Logic and Protocol Analyzer software at <*logic Analyzer Install location*>*help**pdfs*. It describes how to make probing connections from the DUT to the Keysight U4431A module for various probing situations.
- **AXIe Based Logic Analysis and Protocol Test Modules Installation Guide** - This guide provides information on the Keysight AXIe chassis, the U4431A module, how to set up the chassis, module, and host computer and how to obtain and install the associated software components. This guide is available on www.keysight.com/find/mphy_analyzer and is also installed with the Logic and Protocol Analyzer software at <*logic Analyzer Install location*>*help**pdfs*.
- **MIPI M-PHY Protocol Analyzer and Exerciser Online Help** - This online help describes how to configure and use the U4431A module to capture M-PHY data. This online help is integrated with the Logic and Protocol Analyzer software and is accessible from its Help menu.
- **Context-sensitive help** - A context-sensitive HTML help page is available with each window and dialog box of the U4431A module on clicking the Help button displayed within the GUI element.
- **Keysight MIPI M-PHY Command Line Packet Generator User Guide** - This guide describes how to use the Keysight Command Line Packet Generator tool for stimulus generation on an M-PHY link and for capturing bidirectional M-PHY data. This tool is a licensed software option for use with the Keysight U4431A MIPI M-PHY Protocol Analyzer module. This guide gets installed with the CLPG software at <*logic Analyzer Install location*>*help**pdfs*.

2 Configuring U4431A Connection Settings

By default, you can set up a single direction (Tx or Rx) link between the U4431A module and DUT for data capture.

- If you have the Bidirectional (Tx and Rx) license installed, you can set up a dual-simplex link for capturing Tx as well as Rx data. The module allows you to use one or more unidirectional lanes for each direction. You can choose different number of lanes and lane properties for each direction except in case of an SSIC setup in which same link width is used for both sublinks.
- If you have the Command Line Packet Generator (CLPG) license installed, you can set up a Data Generation + Capture link, thereby sending stimulus from U4431A to a DUT and capturing the DUT data.

You set up a link by configuring the module's connection setup in the Keysight Logic and Protocol Analyzer application after completing the hardware setup for the U4431A module in the AXIe chassis and connecting it to the DUT using appropriate probing hardware. You use the **Connection Setup** tab of the U4431A module's **Setup** dialog to configure its connection setup.



The connection setup details that you specify in this tab tells the Logic and Protocol Analyzer software how the U4431A module is connected to the DUT for data capture.

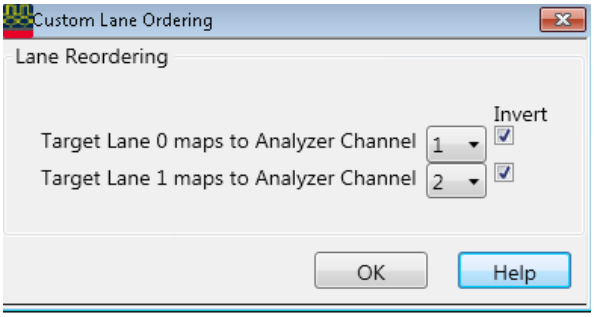
To configure U4431A connection setup

1 In the Logic and Protocol Analyzer GUI, access the Setup dialog box of the U4431A module by clicking **Setup** > **Setup** from the GUI's menubar.

The **Connection Setup** tab is displayed in the Setup dialog box.

2 Configure connection settings in this tab. The tab has the following fields.

Field	Description
Module - This group box displays read-only and editable fields for the U4431A module hardware.	
Chassis	Read-only field that displays the AXIe chassis number in which you installed the U4431A hardware module.
Slot	Read-only field that displays the slot number of the AXIe chassis in which you installed the U4431A hardware module.
Link	Displays the type of link between the U4431A module and DUT based on the protocol family that you selected at the time of starting up the Logic and Protocol Analyzer GUI session.
Link Mode	Currently, U4431A supports the following connection modes. ▪ **Analyzer** - In this mode, the U4431A module can only be used as an analyzer to passively probe and acquire data from DUTs. This mode doesn't allow the usage of the U4431A module as an Exerciser to stimulate the DUT. In this mode, U4431A can acquire both protocol level and raw signal level data simultaneously. ▪ **Data Gen + Analyzer** -.This mode is applicable only when you have installed the Keysight Command Line Packet Generator (CLPG), which is a licensed software option (U4431A-613) of the U4431A module. Selecting this mode ensures that the U4431A link is configured as a packet generator for stimulus transmission as well as an analyzer for data capture. For more information on CLPG and stimulus transmission, refer to the *Keysight MIPI M-PHY Command Line Packet Generator User Guide* installed with the CLPG software at C:\Program Files\Keysight Technologies\Logic Analyzer\Help\pdfs.
Connection	Allows you to select the type of connection you want to set up between the U4431A module and DUT for data capture. The following two connection options are available. ▪ **One Sublink** - This is the default option. It allows you to set up a single unidirectional link (Tx or Rx) between the U4431A module and DUT. ▪ **Both Sublinks** - This is a licensed option. It allows you to set up a bidirectional link (Tx as well as Rx) between the U4431A module and DUT. When you select this option, you get options in the Link Naming and Lane Setup section to set up a Tx and an Rx sublink independently of each other. ***Note***: If you have selected the Data Generator + Analyzer link mode for U4431A, the **One Sublink** option means that the sublink will only be used for capturing data transmitted from DUT. The data transmitted from the U4431A module will not be captured. The **Both Sublinks** option in this case means that the sublink on Pod 1 will be used for capturing data transmitted from DUT and the sublink on Pod 2 will be used for capturing the data transmitted as stimulus by the U4431A module. For more information on Data Generation + Analysis scenario, refer to the *Keysight MIPI M-PHY Command Line Packet Generator User Guide* installed with the CLPG software at C:\Program Files\Keysight Technologies\Logic Analyzer\Help\pdfs.
License	Displays the type of U4431A license(s) currently installed on the system.
Link Naming and Lane Setup	
Name	Displays the name of the M-PHY link that you set up. For a bidirectional link, the names of both Tx and Rx sublinks are displayed. If needed, you can rename these links.
Pods	The U4431A module has two pods labeled as Pod 1 and Pod 2 on its front panel. These pods are used to connect the module to the DUT using probe cables. For a single sublink (Tx or Rx), Pod 1 is used and displayed in this field. For Both sublinks (Tx and Rx), Pod 1 and Pod 2 are used (one for each direction). Ensure that you set up the probing hardware configuration appropriately matching the pod used and displayed for the link. For detailed information on probing, refer to the *U4431A MIPI M-PHY Analyzer - Hardware and Probing Guide.* The guide is available on www.keysight.com/find/mphy_analyzer and also installed with the Keysight Logic and Protocol Analyzer software.

Field	Description
Sublink Width	Select the maximum link width capability that you want to define for the sublink (Tx or Rx). For instance, if you select x4 in this listbox, then you can set x1 to x4 as the initial or current link width of this sublink in the Analyzer Setup tab. This listbox provides options to select a link width from x1 to x4 individually for each sublink. However, some of the options in this listbox are not available if you do not have the appropriate link width license installed. By default, only x1 is available as the link width. Other options are licensed. **NOTES** - Ensure that you make appropriate hardware and probing setup for the U4431A module to match the link width capability you want to set in the Sublink Width field. - For a bidirectional SSIC setup, the same link width is applicable for both sublinks.
Probe Type	Detects and displays the probing option that you have used in the U4431A hardware setup to probe the DUT. The following probing options are available for use with the U4431A module: ▪ U4433A Differential ZIF Flying Lead probe ▪ U4432A SMA probe For detailed information on probing, refer to the *U4431A MIPI M-PHY Analyzer - Hardware and Probing Guide*. The guide is available on www.keysight.com/find/mphy_analyzer and also installed with the Keysight Logic and Protocol Analyzer software.
Lane Order	The **Lane Order** option lets you perform the ordering of the physical probe channels of the U4431A module with the logical data lanes probed by each of these channels. You can either retain the *Default* lane ordering which means channel 0 of the module maps to logical lane 0 and so on. If you want to change this default mapping of channels with logical lanes, then select the *Custom* option from Lane Order and click *Specify* to display the *Custom Lane Ordering* dialog box. In this dialog box, select the module's channel with which you want to map a target logical lane. You can also set the polarity inversion property for each data lane using the *Custom Lane Ordering* dialog box. The polarity of a data lane is not inverted when the positive and negative sides of the probe channel's differential pair are connected to the positive and negative sides of the signal in the DUT. 
State Activity	Displays the current status of the four data lanes LEDs located on the front panel of the U4431A module. The following color coding is used for these LEDs to indicate the state activity on the associated lanes. ▪ **Red** - This means that the lane is in the ultra-low power state (HIBERN8) without loss of configuration information. ▪ **Blinking Red** - This means that the lane state is unknown. This can happen when the U4431A module is in the process of syncing up to the current link configurations. ▪ **Green** - This means that the lane is transmitting HS-BURST in HS-MODE. ▪ **Blinking Green** - This means that the lane is in the STALL state - the power saving state (periods of inactivity) when operating in HS-MODE. ▪ **Yellow** - This means that the lane is transmitting LS-BURST in Low Speed-MODE. ▪ **Blinking Yellow** - This means that the lane is in the SLEEP state - the power saving state (periods of inactivity) when operating in Low Speed-MODE. ▪ **Off** - This means that the lane is not configured/available. Based on the link width that you select in the U4431 Connection Setup tab of the Logic and Protocol Analyzer GUI, the number of lanes on the U4431A module are used. The LEDs of only the used lanes glow. For example, if you are using the x2 link width, then the LEDs of only two lanes being used will glow and the rest of the LEDs will be off. You can also refer to the LED Color Legend section at the bottom of the tab to know what each LED color represents.
LED Color Legend	This section displays a description for various colors of the data lanes status LEDs on the front panel of the U4431A module. Each LED color represents a specific state activity on the associated lane. Refer to the *State Activity* field description in this table to know more about the state activity on lanes.

3 Switching M-PHY Protocols

If you have a single (Unipro, Unipro+UFS, Unipro+CSI-3, or SSIC) protocol license for the U4431A module, then by default, the Logic and Protocol Analyzer GUI is started using that licensed protocol option when you launch this GUI.

If you have purchased multiple protocol licenses for the U4431A module, then the Logic and Protocol Analyzer GUI is started using the currently selected protocol option. In situations when multiple protocol licenses are available, you can choose the protocol license option with which you want to launch the Logic and Protocol Analyzer GUI. This feature allows you to switch among the Unipro, Unipro+UFS, Unipro+CSI-3, MPCIe, and SSIC analysis when using the U4431A module with these multiple protocol license options.

You use the U4421A-U4431A-Protocol-Selector.bat script to select Unipro, Unipro+UFS, Unipro+CSI-3, MPCIe or SSIC protocol when accessing the Logic and Protocol Analyzer GUI.

NOTE **The U4421A-U4431A-Protocol-Selector.bat script puts any active Logic and Protocol Analyzer GUI sessions to the offline mode during protocol change. Therefore, you must stop any ongoing analysis work and save your captured data/configurations before you start using this script.**

To select M-Phy protocols (Unipro, UFS, CSI-3, MPCIe or SSIC)

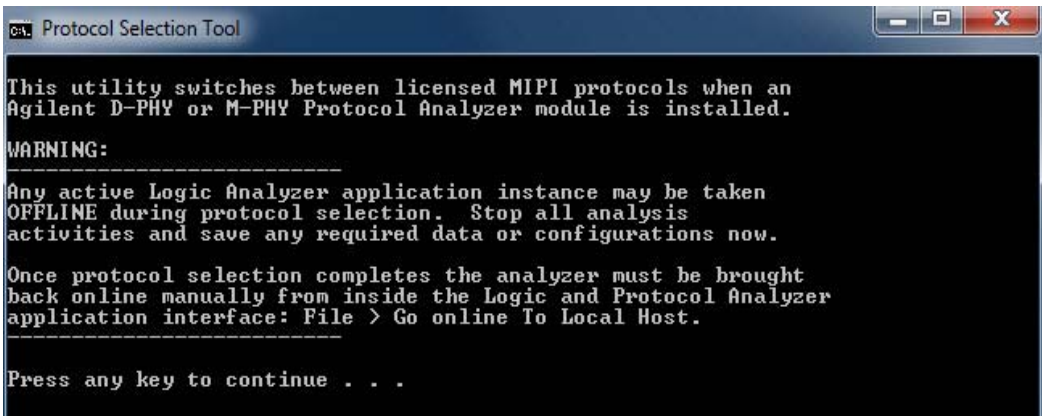
1 Double-click the U4421A-U4431A-Protocol-Selector.bat script to launch it. The script is installed at the following location when you install the Keysight Logic and Protocol Analyzer software (version 5.80.0000 or later).

C:\Program Files (x86)\Keysight Technologies\Logic Analyzer\Contributed Files (for 64 bit OS installs)

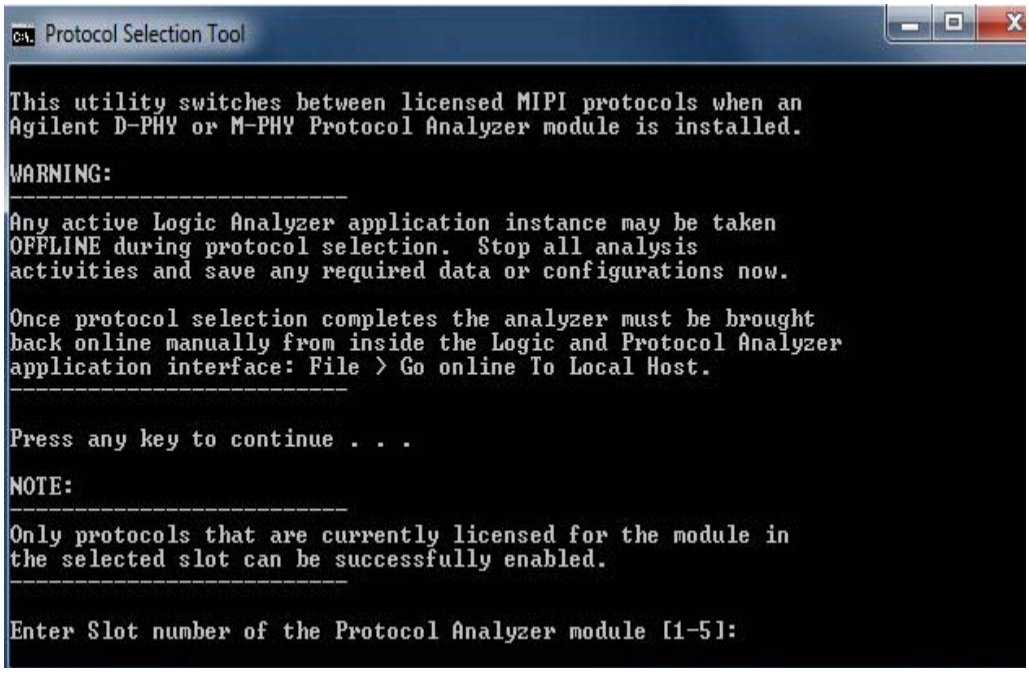
or

C:\Program Files\Keysight Technologies\Logic Analyzer\Contributed Files (for 32 bit OS installs)

2 The script starts running in the Command window with warning messages displayed. Press any key to continue.



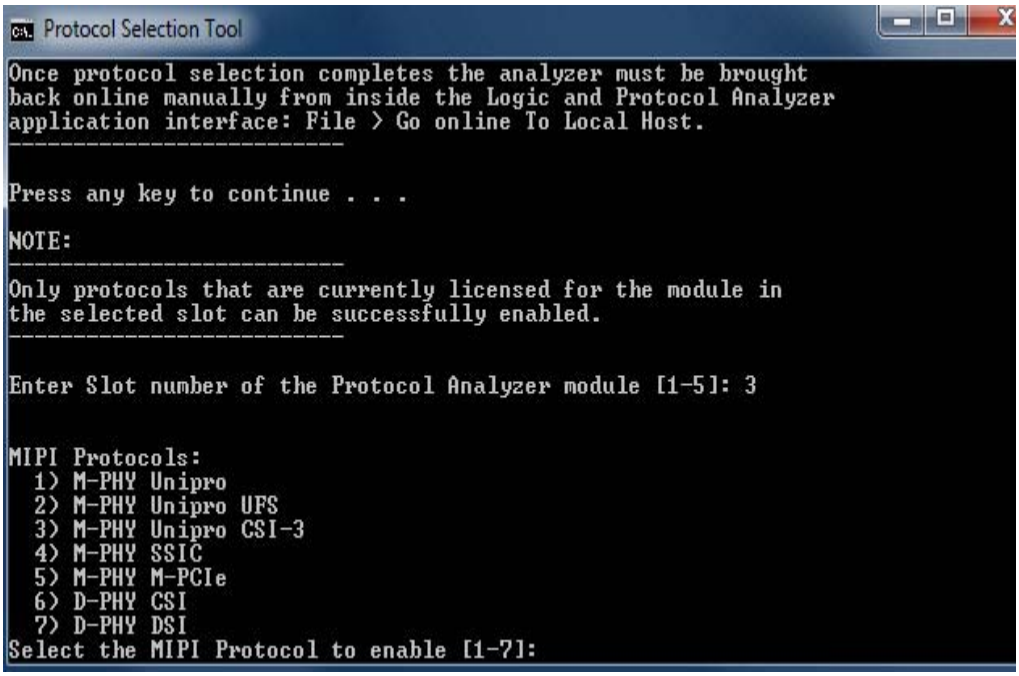
3 Specify the slot number in which you installed the U4431A module for which you want to select the protocol. Press <Enter>.



NOTE If an invalid slot is used or an input other than the number 1 to 5 is entered (for example- any string), the script will run as usual, but the background operation that the script performs will not succeed. The script will crash in case a string with a space is entered.

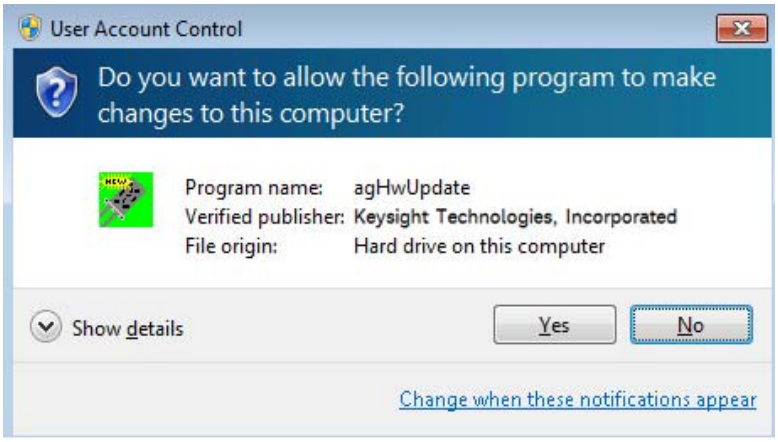
4 Specify the MIPI protocol that you want to select for the module. Specify M-phy Unipro to select the Unipro protocol, M-phy Unipro UFS to select the Unipro+UFS protocol, M-phy Unipro CSI-3 to select the Unipro+CSI-3 protocol, and M-phy SSIC to select the SSIC protocol. Press <Enter>.

NOTE UFS and CSI-3 protocol licenses are available in combination with the Unipro license.



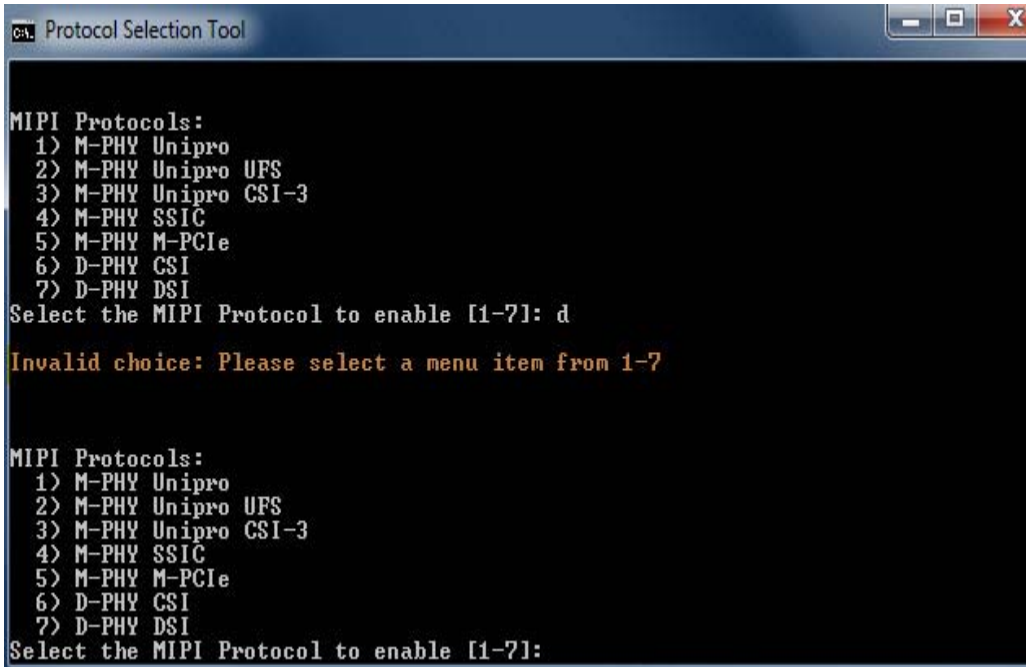
NOTE The CSI and DSI options are available for the U4421A MIPI D-PHY module and these options will be ignored for the U4431A module.

5 The **User Account Control** dialog box is displayed. Click **Yes** to proceed.



The script runs to perform the protocol change as per your selections. The protocol change is successfully completed if you have specified the correct slot number for the U4431A module and appropriate license is available for the selected protocol.

If an invalid option is entered (other than the number 1 to 7), the script fails to execute and keeps on prompting until a valid choice is made.



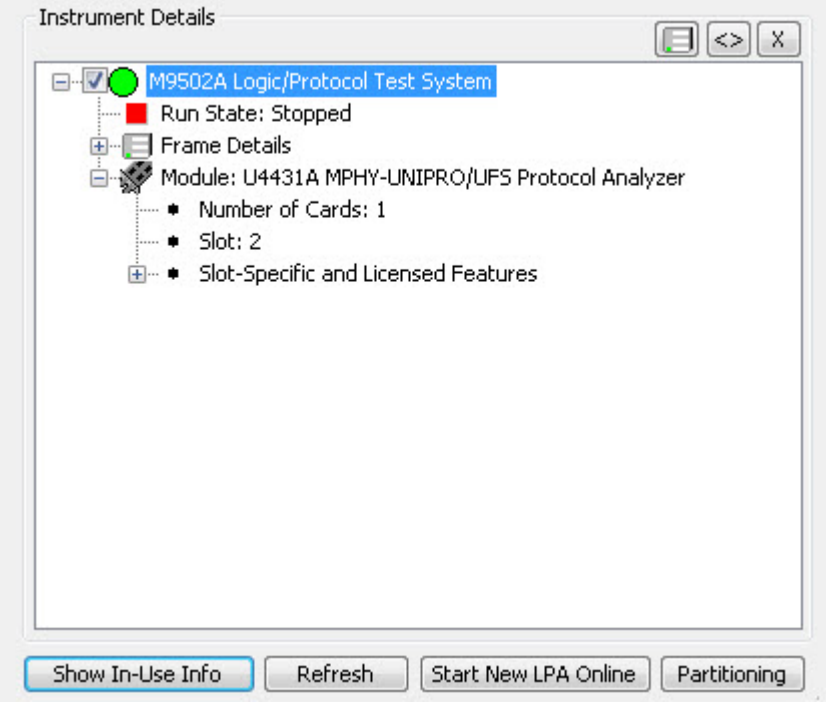
6 When the protocol change completes, you need to bring any open offline sessions of the Logic and Protocol Analyzer GUI to the Online mode by clicking File > Go online to Local Host option in the GUI's menubar. If there are no existing open sessions of the GUI, launch the GUI in the Online mode.

The Logic and Protocol Analyzer GUI should now launch with the selected protocol.

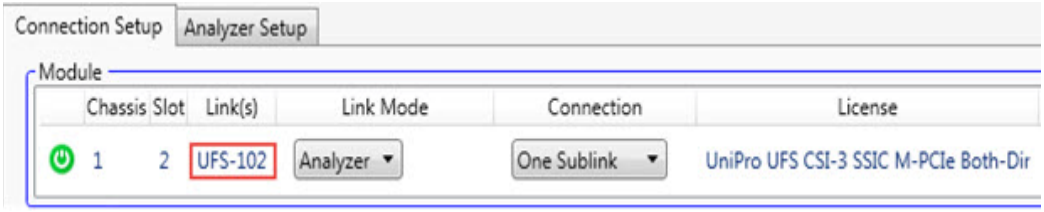
To verify if protocol change is successful

- Using the Keysight Notification Center icon
  1. Double-click the **Keysight Notification Center** icon displayed in Window's taskbar.
  2. In the **Instrument Details** dialog box, expand the U4431A module option for which you changed the protocol.

     The protocol currently selected for the module is displayed.



- Using the Logic and Protocol Analyzer GUI
  1. Launch the Logic and Protocol Analyzer GUI in the Online mode.
  2. Click **Setup > Setup** from the GUI's menubar to access the **Setup** dialog box of the U4431A module for which you changed the protocol.
  3. In the **Connection Setup** tab, the currently selected protocol (Unipro, UFS, CSI-3, MPCIe or SSIC) for the module is displayed.



4 Capturing M-PHY Data

This chapter provides information on how to configure the U4431A module as an analyzer to capture M-PHY data.

Before you Start

Before you start capturing data, ensure that the following tasks are performed.

- You have connected the U4431A module to the DUT using the appropriate acquisition probe in the required configurations.
- You have installed the Keysight Logic and Protocol Analyzer GUI, version 5.80.0000 or higher.
- You have configured the U4431A module's connection setup in the Keysight Logic and Protocol Analyzer GUI.
- You have specified the sync related settings in the Analyzer Setup tab. The U4431A module uses these settings to sync up to the DUT's link configurations before data capture.

Supported Probes

The following probes are supported for use with the U4431A module.

- **U4433A Differential ZIF Flying Lead probe** - Can probe individual signals from multiple randomly located points on the target system. For detailed probing information and how to make connections with the U4431A module and DUT, refer to the *U4431A MIPI M-PHY Analyzer Hardware and Probing Guide.* The guide is available on www.keysight.com/find/mphy_analyzer and also installed with the Keysight Logic and Protocol Analyzer software.
- **U4432A SMA probe** - Provides coaxial SMA connections for each of the lines of the M-PHY bus. You can use the U4432A SMA probe in the following two scenarios with the U4431A module.
  - **For Data Capture -** For detailed probing information and how to make connections with the U4431A module and DUT, refer to the *U4431A MIPI M-PHY Analyzer Hardware and Probing Guide.* The guide is available on www.keysight.com/find/mphy_analyzer and also installed with the Keysight Logic and Protocol Analyzer software.
  - **For Packet Generation as well as data capture -** You can also use the U4432A SMA probe with the Command Line *Packet Generator (CLPG) option (U4431-613)* of the U4431A module to transmit stimulus and receive DUT data. You can find the details of this usage scenario in the ***Keysight MipiMphy Command Line Packet Generator User Guide*** which is installed with the Keysight Command Line Packet Generator software.

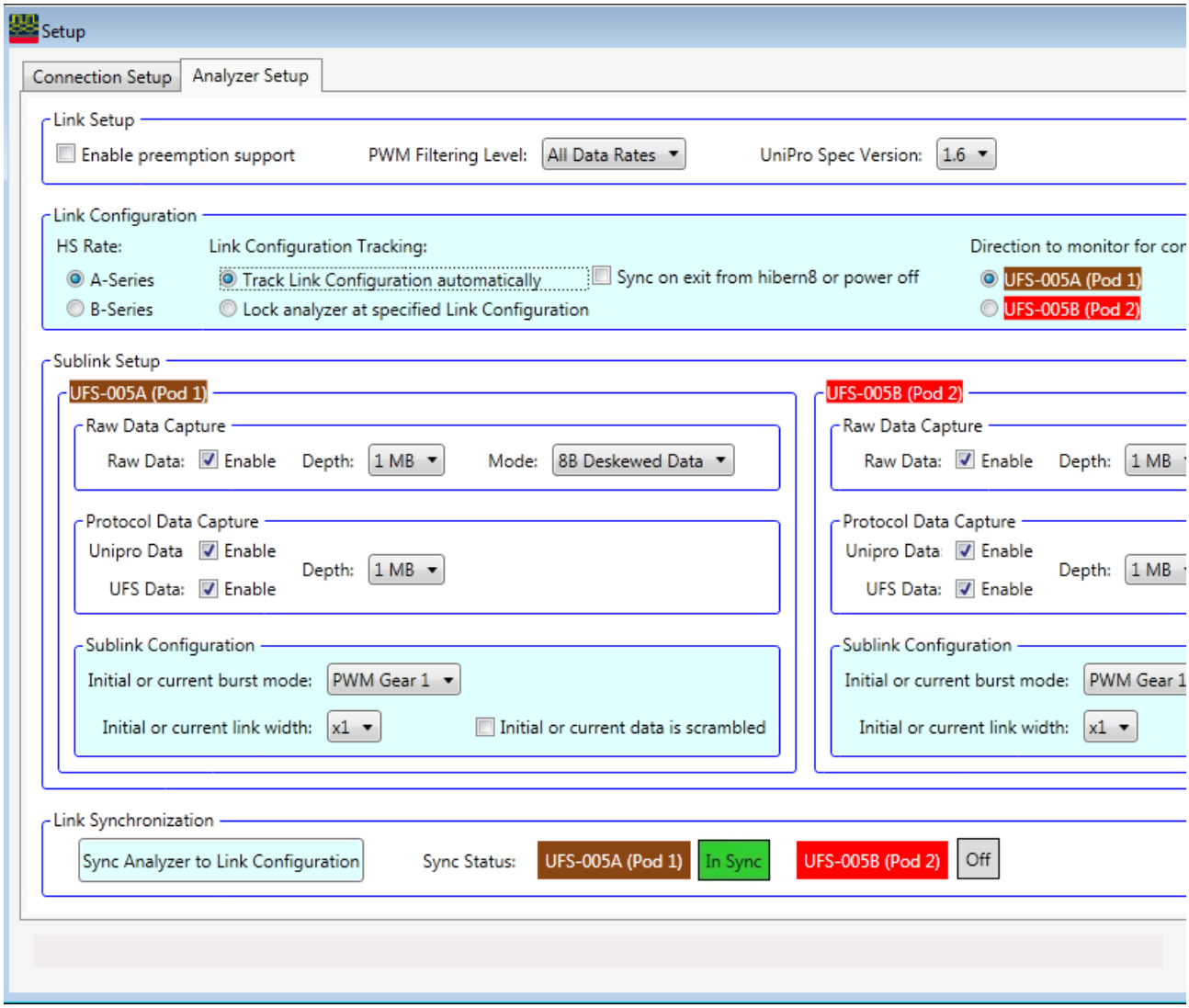
Configuring Data Capture and Sync Settings

You can configure the data capture settings at the link and sublink(s) levels using the **Analyzer Setup** tab in the **Setup** dialog box of the Logic And Protocol Analyzer GUI.

The U4431A module performs the sync process before it starts data capture to sync up to the DUT's link configurations such as link width, burst mode, link speed, and HS rate series. To enable the U4431A module to perform this sync process, you define link/sublink settings that the module should use during this sync process and also instruct the U4431A module on how it should perform this sync process. You use the **Analyzer Setup** tab to define these settings.

NOTE The U4431A module does not start capturing data until it completes the sync process.

The following screen displays the capture settings fields available for a bidirectional UniPro/UFS setup.



The **Analyzer Setup** tab has the following fields that you can modify to configure the capture and sync related settings.

Some fields are license-specific and are displayed only when an appropriate U4431A license is installed.

NOTE

If you configured the U4431A module's link mode as Data Generation + Analyzer and Connection as Both Sublinks, then:

- the settings you configure in the Pod 1 Sublink Setup section are applicable for capturing data transmitted from DUT

- the settings you configure in the Pod 2 Sublink Setup section are applicable for capturing data transmitted from U4431A.

For detailed information on this usage scenario, refer to the Keysight MIPI M-PHY Command Line Packet Generator User Guide installed with the Command Line Packet Generator (CLPG) software at the following location.

C:\Program Files\Keysight Technologies\Logic Analyzer\Help\pdfs

Field	Description
Link Setup	
Enable Preemption Support *(Applicable only for a UniPro setup)*	Allows you to enable or disable the preemption support for the link. The preemption support is based on the UniPro specifications. Therefore, this option is applicable and visible only when you install the UniPro specific license on the system. When this support is enabled, the U4431A module identifies and supports preemption in the captured data, for instance, preemption for higher priority frames into lower priority data frames. For preemption, U4431A supports the nesting limited to level 1. With preemption enabled, you can view the preempted frames in the captured trace displayed in the Protocol Viewer.
Data on link is scrambled *(Applicable for SSIC and MPCIe setup)*	Allows you to instruct the U4431A module on whether or not the data on the link is scrambled. Depending on the protocol license that is either SSIC or MPCIe whichever is currently applicable, the scrambling is based on the SSIC or MPCIe specifications. Therefore, this option is applicable and visible only when you install the SSIC or MPCIe specific license on the system. If you select this checkbox, the U4431A module expect scrambled data on the link and enables descrambling of the scrambled data. If you do not select this checkbox, the U4431A module does not expect scrambled data and therefore descrambling is not enabled and the data is presented as is. You can view the descrambled data in the captured trace using the Waveform Viewer. To know how to view scrambled data, refer to the topic "Viewing Deskewed and Descrambled Data" on page 72.
PWM Filtering Level	Allows you to instruct the U4431A module on whether or not it should implement the noise filtering when capturing PWM signals which are more prone to noise. This listbox also allows you to set the amount of noise filtering to be applied for PWM gears based on the actual bit rate being transmitted by the DUT. You can select from the following filtering options. ▪ **No Filtering** - This option means that there is no noise filtering applied for PWM gears. ▪ **All Data Rates** - This option means that the U4431A module applies the maximum possible filtering even when the DUT is transmitting at the maximum data rate from the data rate range defined for all PWM gears. This is the default setting and is suitable for most of the situations. ▪ **Rates < = 75% Max** - This option means that the U4431A module applies more filtering than applicable for the All Data Rates option. However, for this option, the DUT should transmit at less than or equal to 75% of the maximum data rate from the data rate range defined for all PWM gears. ▪ **Rates < = 25% Max** - This option means that the U4431A module applies more filtering than applicable for the Rates < = 75% Max option. However, for this option, the DUT should transmit at less than or equal to 25% of the maximum data rate from the data rate range defined for all PWM gears. ***NOTE***: PWM Filtering is applicable only to lane 0 of each sublink. The U4431A module does not perform PWM filtering for lanes 1-3. Therefore, to ensure that you get good data in case of multilane PWM configurations, enable terminations on the link when the link is in the PWM mode. This can usually be done by issuing a power mode request.

Field	Description
HS Profile *(Applicable only for an SSIC setup)*	This option is displayed only when you install the SSIC license for the U4431A module. The option is enabled only when you select the **Track Link Configuration Automatically** option in this tab. The HS Profile option has the following two components: ▪ **Gear** - Allows you to set the HS Gear profile for the U4431A module. This setting is applicable for both sublinks in case of a bidirectional link configuration. The U4431A module can dynamically change to this profile in response to the changes in the probed link's configurations. For instance, you can set the initial burst mode as PWM Gear 1 in the Initial or current burst mode listbox and set the HS Profile to HS Gear 2. This will ensure that initially, PWM Gear 1 is used as the burst mode and then later HS Gear 2 is used in response to the probed link's configuration changes. ▪ **Width** - This is a read-only field that displays the link width that you configured for the U4431A module in the Connection Setup tab. This link width profile is applicable for both sublinks in case of a bidirectional link configuration.
UniPro Spec Version *(Applicable only for a UniPro setup)*	Allows you to select the version of the UniPro specifications based on which you want the U4431A module to decode and present the captured data. For instance, if you select the 1.6 version, you get an option to instruct the U4431A module on whether or not the data on a sublink is scrambled. The listbox is visible only when you install the UniPro specific license.
Link Configuration	
HS Rate	Select the data rate series at which you want the U4431A module to capture data in the High Speed (HS-Mode). The selected rate series is applicable for both sublinks in the HS-Mode. As per the M-PHY specifications, the following two Data Rate series are available for selection: ▪ A-Series ▪ B-Series The data rate applicable for various gears in these series is as per the data rate defined in the M-PHY specifications.
Link Configuration Tracking	Allows you to configure how you want to the U4431A module to track and act for the changes in the probed link configurations such as link speed and link width. ▪ **Track Link Configuration Automatically** - Select this option to ensure that the U4431A module automatically tracks the probed link's configuration changes such as changes in sublink speed and width. On enabling this option, the U4431A module automatically changes its link configurations in response to the changes in the probed link's configurations. Such automatic tracking and adjustments in link configurations are then viewable in the data captured and displayed in the Protocol Viewer. ▪ **Lock Analyzer at Specified Link Configuration** - Select this option to ensure that the link configurations such as sublink speed and sublink width of the U4431A module are fixed to the values that you currently selected in the Sublink Setup group box. On selecting this option, the U4431A module does not change its link configurations automatically in response to the changes in the probed link's configurations. When locked to a specific configuration, the U4431A module will capture data specific to only that configuration.
Sync on exit from hibern8 or power off	Select this checkbox to instruct the U4431A module to sync up to the DUT configurations on DUT's exit from the hibern8 or power off state. This option is disabled and therefore fixed if you select the Lock Analyzer at Specified Link Configuration option. This option is enabled only when you have instructed the U4431A module to track and respond to the changes in the probed link configurations automatically. Selecting this checkbox is particularly useful when the U4431A module is in the process of syncing to the DUT's configurations and you want to convey the hibern8 or power off state of DUT to the U4431A module. When you select this checkbox, the U4431A module gets to know about the hibern8 or powered off state of DUT and then consequently starts ignoring all data from the DUT until it detects the DUT's exit from HIBERN8. To ensure that the dynamic changes to this checkbox take effect, you also need to click the *Sync Analyzer to Link Configuration* button after selecting/deselecting this checkbox. ***NOTE***: When the U4431A module completes its sync process, it can automatically track the DUT's entry and exit from hibern8.
Direction to monitor for configuration tracking	This option is available only when you have chosen to analyze both Tx and Rx sublinks (Both Sublinks option in the Connection Setup tab). This option allows you to select the sublink that you want the U4431A module to monitor for tracking the link configuration requests. ***NOTE***: This field is not available for a "Both Sublinks" configuration if you have configured the U4431A module in the Packet Generator + Analyzer link mode. In this link mode, the U4431A module always monitors the sublink on Pod 1 (used for DUT data capture) for tracking the link configuration requests. The sublink on Pod 2 is not used for this purpose in any scenario.
Sublink setup	

Field	Description
Raw Data Capture - The U4431A module can capture both protocol level and raw signal level data simultaneously. The raw data capture allows you to view the time-correlated 8b/10b data or 8b Deskewed data that underlies a protocol. This data can be displayed in the Waveform or Listing viewer, providing insight into how a packet is formed at the physical layer. **Note**:: When you configure the U4431A module in the Data Generator + Analyzer mode, the raw data capture is not supported for capturing the data transmitted from the U4431A module. Therefore, in such a scenario, the Raw Data Capture section is disabled for the Pod 2 sublink setup. The Raw Data Capture group box contains the following fields to enable and set the raw data capture setting for a sublink.	
Raw Data	You can choose to enable or disable the raw signal level data capture for the applicable sublink. You can instruct the U4431A module to capture the raw signal level data by selecting the **Enable** checkbox displayed with the **Raw Data** field.
Depth	You can divide the total memory depth that is licensed and available for the U4431A module between the configured sublink(s) of the link. For each sublink, you further divide the allocated memory between the protocol level data acquisition and raw signal level data acquisition (if enabled). From the **Depth** listbox, select the memory depth that you want to allocate to the acquisition of the raw signal level data for the sublink. This listbox is disabled if you deselect the **Enable** checkbox displayed with the **Raw Data** field. You can allocate a value ranging from 64 KB to 4 GB as the raw data capture memory depth.
Mode	Select the mode of raw signal level data acquisition. You can select from the following three options: ▪ **8b Data** - Select this option if you want to display 8b symbol data. Data will not be deskewed in this case. ▪ **10b data** - Select this option if you want to display 10b raw data. Data will not be deskewed/descrambled in this case. ▪ **8b Deskewed Data** - Select this option if you want the acquired data to be deskewed and aligned across lanes. ▪ **8B Descrambled and Deskewed Data** - This option is applicable for SSIC and MPCIe setup. Select this option if you are expecting scrambled data on the sublink and want the U4431A module to descramble and deskew the acquired data. On selecting this option, the module will, however, descramble the acquired data only if the Data on link is scrambled checkbox is selected. ▪ **8B Descrambled Data** - This option is applicable only for an SSIC setup. Select this option if you are expecting scrambled data on the sublink and want the U4431A module to descramble the acquired data. On selecting this option, the module will, however, descramble the acquired data only if the Data on link is scrambled checkbox is selected.
Protocol Data Capture - The U4431A module can capture both protocol level and raw signal level data simultaneously. The protocol data capture allows you to view the protocol level data in the Protocol viewer, providing an insight into packet trace and packet debug. The Protocol Data Capture group box contains the following fields to enable and set the protocol data capture setting for a sublink.	
UniPro Data *(Applicable only for a UniPro setup)*	You can choose to enable or disable the decoding and extraction of the UniPro protocol level data for the applicable sublink. You can instruct the U4431A module to decode and extract the UniPro protocol level data by selecting the **Enable** checkbox displayed with the **UniPro Data** field. If this checkbox is not selected, the UniPro protocol level data is not available for viewing in the Protocol Viewer tool of the Keysight Logic and Protocol Analyzer application. This field is displayed only when you install the UniPro specific license for the U4431A module.
UFS Data *(Applicable only for a UFS setup)*	You can choose to enable or disable the decoding and extraction of the UFS protocol level data for the applicable sublink. You can instruct the U4431A module to decode and extract the UFS protocol level data by selecting the **Enable** checkbox displayed with the **UFS Data** field. If this checkbox is not selected, the UFS protocol level data is not available for viewing in the Protocol Viewer tool of the Keysight Logic and Protocol Analyzer application. This field is displayed only when you install the UniPro+UFS license for the U4431A module.
CSI-3 Data *(Applicable only for a CSI-3 setup)*	You can choose to enable or disable the decoding and extraction of the CSI-3 protocol level data for the applicable sublink. You can instruct the U4431A module to decode and extract the CSI-3 protocol level data by selecting the **Enable** checkbox displayed with the **CSI-3 Data** field. If this checkbox is not selected, the CSI-3 protocol level data is not available for viewing in the Protocol Viewer tool of the Keysight Logic and Protocol Analyzer application. This field is displayed only when you install the UniPro+CSI-3 license for the U4431A module.
SSIC Data *(Applicable only for an SSIC setup)*	You can choose to enable or disable the decoding and extraction of the SSIC protocol level data for the applicable sublink. You can instruct the U4431A module to decode and extract the SSIC protocol level data by selecting the **Enable** checkbox displayed with the **SSIC Data** field. If this checkbox is not selected, the SSIC protocol level data is not available for viewing in the Protocol Viewer tool of the Keysight Logic and Protocol Analyzer application. This field is displayed only when you install the SSIC license for the U4431A module.
MPCIe Data *(Applicable only for MPCIe setup)*	You can choose to enable or disable the decoding and extraction of the MPCIe protocol level data for the applicable sublink. You can instruct the U4431A module to decode and extract the MPCIe protocol level data by selecting the **Enable** checkbox displayed with the **MPCIe Data** field. If this checkbox is not selected, the MPCIe protocol level data is not available for viewing in the Protocol Viewer tool of the Keysight Logic and Protocol Analyzer application. This field is displayed only when you install the MPCIe license for the U4431A module.

Field	Description
Depth	You can divide the total memory depth that is licensed and available for the U4431A module between the configured sublink(s) of the link. For each sublink, you further divide the allocated memory between the protocol level data acquisition and raw signal level data acquisition (if enabled). From the **Depth** listbox, select the memory depth that you want to allocate to the acquisition of the protocol level data for the sublink. This listbox is disabled if you deselect the **Enable** checkboxes displayed in the Protocol Data Capture section. You can allocate a value ranging from 64 KB to 4 GB as the protocol data capture memory depth.
Sublink Configuration	
Initial or current burst mode	Allows you to select the burst mode that you want the U4431A module to initially use for the sublink during the process of syncing to the DUT's link configurations. Once initially set, you can later use this listbox to dynamically change the burst mode of the sublink as and when you need. For such dynamic changes to take effect, you also need to click the Sync Analyzer to Link Configuration button after setting the burst mode. In this listbox, by default, you get options to set: ▪ HS Gear 1 or PVM Gear 1 to 7 for a UniPro/UFS/CSI-3 sublink. ▪ HS Gear 1 or PVM Gear 1 for SSIC and MPCIe sublinks. The HS Gear 2 and 3 are licensed options and are displayed only when you install the appropriate license. If you select a PVM Gear, then the minimum and maximum range of bit rates defined for that PVM gear in the specifications are applicable for the sublink. If you select an HS Gear, then the data rate defined for that gear in the selected HS Rate series is applicable for the sublink.
Initial or current link width	Allows you to select the link width that you want the U4431A module to initially use for the sublink during the process of syncing to the DUT's link configurations. Once initially set, you can later use this listbox to dynamically change the link width of the sublink as and when you need. For such dynamic changes to take effect, you also need to click the **Sync Analyzer to Link Configuration** button after setting the link width. In this listbox, by default, you get the option to set x1 as the link width for a sublink. The x2 to x4 are licensed options and are displayed only when you install the appropriate license. ***Notes*** ▪ The link width options displayed in this listbox also depend on the link width capabilities that you defined in the **Sublink Width** field in the **Connection Setup** tab. For instance, if you set the link width capabilities of a sublink as x2 in the Connection Setup tab, then only x1 and x2 options are displayed in the Initial or Current link width listbox. ▪ For SSIC and MPCIe, only **x1** is displayed as the link width option in case you select the **PWM Gear 1** burst mode.
Initial or current data is scrambled *(Applicable only for a UniPro setup)*	This checkbox is displayed only when you select the **1.6** option from the **UniPro Spec Version** listbox for a UniPro setup. This checkbox allows you to instruct the U4431A module on whether or not the data on the sublink is scrambled initially during the process of syncing to the DUT's link configurations. Once initially set, you can later use this checkbox to dynamically instruct the module on scrambled data on the sublink. For such dynamic changes to take effect, you also need to click the Sync Analyzer to Link Configuration button after setting the scrambling. When this checkbox is selected, the U4431A module enables descrambling for the scrambled data on the link.
Link Synchronization	
Sync Analyzer to Link Configuration	Click this button to synchronize the U4431A analyzer module to the currently set link configurations such as current link width and current link speed. On clicking this button, the current status of the sublink(s) in terms of their synchronization to current link configurations is displayed. This option is particularly useful when you dynamically changed the sublink configurations using the *Initial or current burst mode* or *Initial or current link width* listboxes and want to synchronize the U4431A module to these changes.
Sync Status	Displays the current status of the U4431A sublink(s) in the context of its synchronization to the currently set link configurations. A sublink can have one of the following link synchronization status: ▪ **In Sync** - Indicates that the sublink is in sync to the current link configurations. ▪ **Off** - Indicates that the sublink is not connected/active. ▪ **Syncing** - Indicates that the U4431A sublink is currently in the process of syncing to the current link configurations. If it is taking too long to sync, it may indicate that the U4431A module is not able to sync to the currently set link configurations. In such a situation, you may want to check if the link configurations that you are trying to set are matching the DUT's configurations/capabilities. The Sync Status is updated: ▪ when you click the *Sync Analyzer to Link Configuration* button. ▪ when you change one of the settings in the Setup dialog box that impacts the link configuration and then click *Apply* or *OK*. ▪ or automatically when there is a change in the probed link configurations and you have instructed the U4431A module to automatically track and sync to such changes.

After you have configured the capture settings, save the settings in a Logic Analyzer configuration (.ala) file. To do this, click File > Save, select a location and name for the configuration file, select Standard Configuration (.ala) from the Save as type listbox and then click Save.

Setting up Triggers

Trigger - Overview

You can trigger the U4431A module to start storing the captured data in its memory when the specified trigger condition is met.

Using triggers, you can isolate events of interest in traffic. You can use triggers to detect errors at each layer of the protocol. You can set triggers for:

- Specific UniPro/UFS/SSIC/CSI-3/MPCIe packet types (based on the protocol license(s) you have installed)
- Physical layer triggers - Currently Crc, disparity, and symbol errors are supported.
- Occurrence counter
- Global counter
- Timer

You can set a *Simple* or an *Advanced* trigger.

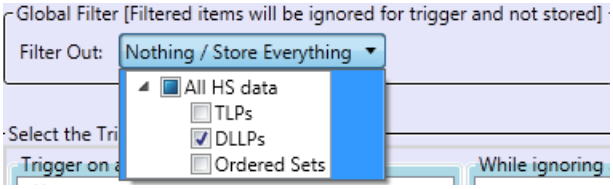
- Simple trigger - A simple trigger allows you to quickly set up a trigger without getting involved into setting up multi-level sequenced steps for the trigger. In this type of trigger, you can include one or more trigger events whose occurrence causes the module to trigger the storage of the captured data.
- Advanced trigger - An advanced trigger allows you to set up a complex multi-level sequenced trigger. In an advanced trigger, you can include multiple steps. In each step, you can define one or more trigger events and the action that should be performed with the occurrence of that event.

Trigger Settings

Based on the type of trigger (Simple or Advanced) that you are setting up, a trigger setup may be comprised of a trigger position, type, mode, store qualification, condition(s), and action. All these components of a trigger are set up using the fields displayed in the *Trigger* dialog box.

You access the Trigger Dialog box by selecting **Setup>Trigger...**. from the *Keysight Logic and Protocol Analyzer* GUI's menu bar.

The following table has descriptions of all the fields available in this dialog box for setting up a simple or an advanced trigger. (See "Triggering Based on Packet Types, "To Set up a Simple Trigger, and "To Set up an Advanced Trigger for more information.)

Field	Description
Link tabs	The top of the Trigger dialog box allows you to add tabs that let you set up separate triggers for different MIPI M-PHY links. You can add tabs using the + icon and then apply these tabs to different links that are set up in the Connection Setup tab of the Setup dialog box.
Applies to	Displays the MIPI M-PHY links that are set up in the Connection Setup tab of the Setup dialog box. Selecting one or more links from the **Applies to** drop-down listbox applies the trigger settings in the current tab page to the data captured on the selected links.
Trigger Position	Defines the trigger position within the acquisition memory of the U4431A module. The selected trigger position on the slider sets the pre and post trigger memory ratio. By default, the available acquisition memory is equally divided between the pre and post trigger memory.
Favorite Triggers	Displays a drop-down menu for: ▪ opening a previously saved trigger setup. ▪ saving the currently configured trigger setup in a Trigger Setup (.trg) file. ▪ viewing and accessing a list of recently accessed trigger setup files. ***Note***: The favorite trigger setups referred in this drop-down menu are different from the favorites list that appear in the left-side events pane. In the left pane, the list of favorites refer to the favorite trigger events that you added, edited, and saved to the favorites list using the Event Editor dialog.
Clear	Clears the current trigger settings that you configured in the current tab and restores the default settings.
Global Filter [Filtered items will be ignored for trigger and not stored]	The **Global Filter** option is visible only for SSIC and M-PCIe protocols. In the **Global Filter** section, you can select an option from the **Filter Out** list-box to specify whether or not to store certain type of packets after triggering. You can also choose whether or not to consider the filtered items for trigger. By default, all the packets are stored. If you don't want to store certain types of packets after triggering, select the required option from the **Filter Out** list-box. For example, if you don't want to store DLLPs after triggering, select the DLLPs checkbox in the **Filter Out** list-box. The DLLPs will not be considered for the trigger. 
For setting up a Simple Trigger - The following fields are applicable for a simple trigger.	
Simple Trigger	Select the **Simple Trigger** radio button. Selecting this radio button displays the fields relevant for setting a simple trigger in the **Trigger** dialog box. These fields are described below.
List of events	In the left pane of the Trigger dialog box, an expandable list of events is displayed based on the protocol family (M-PHY, UniPro, or UFS) currently active for the U4431A module. The events in this list are organized based on categories such as: ▪ Types of packet - Displayed on the basis of the protocol license(s) you have installed. ▪ Physical Layer triggers - The transmission of an erroneous packet, for instance, with Crc, symbol, or disparity error sets the trigger. ▪ Arming - The U4431A module can receive triggers from another module installed in the AXIe chassis or another device with which it is connected via the Trigger in Connector on the AXIe chassis. You can select one or more of these events to act as trigger condition(s).
Trigger Mode	▪ Select the **Trigger on Packets or Ordered Sets** radio button to ensure that the U4431A module is triggered on the occurrence of the trigger event(s) included in the *Select the Trigger(s) to use* section. ▪ Select the **Trigger when Stop button pressed** radio button to ensure that the U4431A module is triggered to store the captured data when you click the Stop button to stop the data capture. On selecting this option, the trigger events are not applicable. Therefore, the Select the Trigger(s) to use section is disabled.

Field	Description
Select the Trigger(s) to use	▪ **Trigger on any of these events** - You can drag events from the list of events displayed in the left pane and drop these events in this section. The U4431A module is triggered on the occurrence of any of the events included in this section. ▪ **While ignoring any of these events** - You can drag events from the list of events displayed in the left pane and drop these events in this section. The U4431A module is **not** triggered on the occurrence of any of the events included in this section. To edit events included in the above-mentioned sections, click the underlined event name. This displays the *Event Editor* dialog box to let you edit the event properties or add the event to the list of favorite events. To remove events from the above-mentioned sections, click the "X" displayed to the left of the event name.
For setting up an Advanced Trigger- The following fields are applicable for an advanced trigger.	
Advanced Trigger	Select the **Advanced Trigger** radio button. Selecting this radio button displays the fields relevant for setting an advanced trigger in the **Trigger** dialog box. These fields are described below.
List of events	In the left pane of the Trigger dialog box, an expandable list of events is displayed based on the protocol family (M-PHY, UniPro, or UFS) currently active for the U4431A module. The events in this list are organized based on categories such as: ▪ Type of packet - Displayed on the basis of the protocol license(s) you have installed. ▪ Physical Layer triggers - The transmission of an erroneous packet, for instance, with Crc, symbol, or disparity error sets the trigger. ▪ Timers, counters, and flags - Sets trigger on a timeout or the expiry of a counter. The U4431A module can also receive triggers from another module installed in the AXIe chassis or another device with which it is connected via the Trigger in Connector on the AXIe chassis. The *Arm In from* event is used in such a situation. ▪ Comment - Allows you to add comments to your advanced trigger setup. You can select one or more of these events to act as trigger condition(s).
Select the Trigger(s) to use	By default, this section displays a single step with an 'If - then' specification for the trigger condition and subsequent action. If required, you can add more steps (maximum eight) in this section to set up a complex multi-level sequenced trigger. This section has the following fields: ▪ The Step button allows you to add or delete steps from the sequence. ▪ The If/Else if buttons let you insert additional "If" clauses in the same step or delete an additional "if" clause from the step. ▪ To add multiple conditions / actions for a step, drag the required event from the list of events in the left pane and drop it to the step box. ▪ The event chevron buttons let you insert, delete, or logically group (or negate) events. ▪ The action chevron buttons let you insert or delete actions.

To Set up a Simple Trigger

1 In the *Keysight Logic and Protocol Analyzer* GUI's Overview window, click the M-PHY module and select **Setup>Trigger...**. from the drop-down menu. Alternatively, click the icon displayed for the M-PHY module.

The **Trigger** dialog box is displayed.

2 From the **Applies to** listbox, select the MIPI M-PHY link(s) to which you want to apply the trigger settings.

3 If required, use the **Trigger Position** slider to change the default trigger position (50%) for the allocation of pre and post trigger memory.

4 Select the **Simple Trigger** radio button. All simple trigger related fields are then displayed.

5 Select the **Trigger Mode**. On selecting the **Trigger on Packets or Ordered Sets** radio button, the **Select the Trigger(s) to use** section is displayed in which you can add trigger events. For the **Trigger when Stop button is pressed** option, trigger events are not applicable. Therefore, you cannot add any trigger events on selecting this option.

6 If you don't want to store certain packet types, select the packet category from the **Filter Out** listbox in the **Global Filter** section.

7 Drag and drop the desired trigger event(s) from the event list in the left pane to the **Trigger on any of these events** section on the right.

8 To change an added trigger event's properties, click that event's underlined name in the **Trigger on any of these events** section.

9 To delete an added trigger event from the **Trigger on any of these events** section, click the "X" to the left of the event name.

10 From the event list in the left pane, drag the required events that you want to exclude and drop these in the **While ignoring any of these events** section.

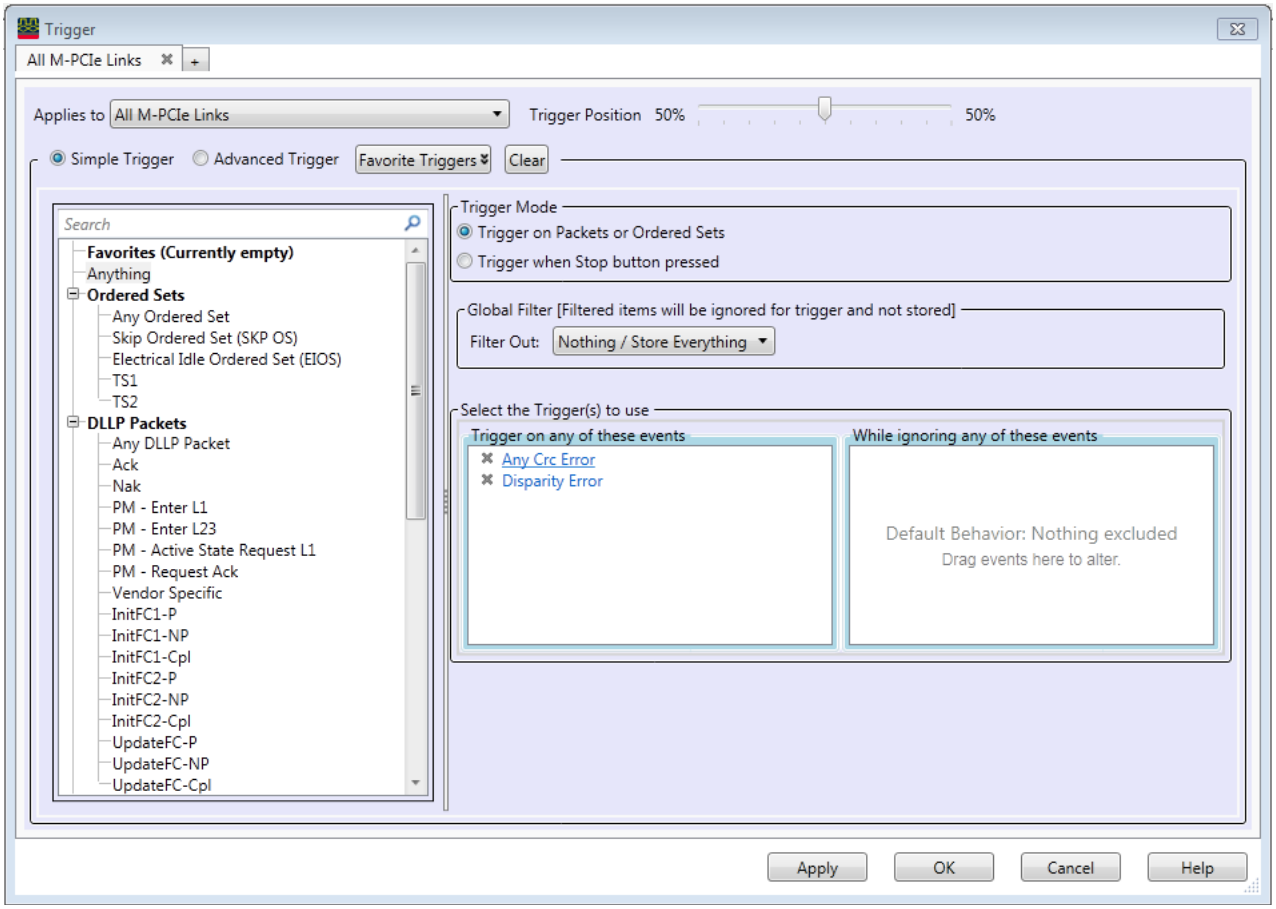
11 Click **Apply**.

12 To save the currently configured trigger setup in a Trigger Setup (.trg) file, click the **Favorite Triggers** button and then select the **Save** option from the displayed drop-down menu.

13 Click **OK**.

Trigger Example

The following trigger setup aims at triggering the U4431A module on the transmission of *M-PCIe* packet with Crc or disparity error.



To Set up an Advanced Trigger

1 In the *Keysight Logic and Protocol Analyzer* GUI's Overview window, click the M-PHY module and select **Setup>Trigger...**. from the drop-down menu. Alternatively, click the 

icon displayed for the M-PHY module.

The **Trigger** dialog box is displayed.

2 From the **Applies to** listbox, select the MIPI M-PHY link(s) to which you want to apply the trigger settings.

3 If required, use the **Trigger Position** slider to change the default trigger position (50%) for the allocation of pre and post trigger memory.

4 Select the **Advanced Trigger** radio button. All advanced trigger related fields are then displayed.

5 As per your specific requirements:

a Edit the default step (Step 1) to select the trigger condition and action in the "If-Then" clause.

b Add more "If-Then" clauses to a step by clicking the If/Else If button in the step.

c Add more than one And/Or condition to an "If" clause of a step by clicking the  button for events. You can also negate a condition by selecting the "Insert Not at beginning of row" option from the  drop-down menu.

d Add more than one action to "Then" clause of a step by clicking the  button for actions.

e Add more steps before or after a step by clicking a Step button.

6 If you don't want to store certain packet types, select the packet category from the **Filter Out** listbox in the **Global Filter** section.

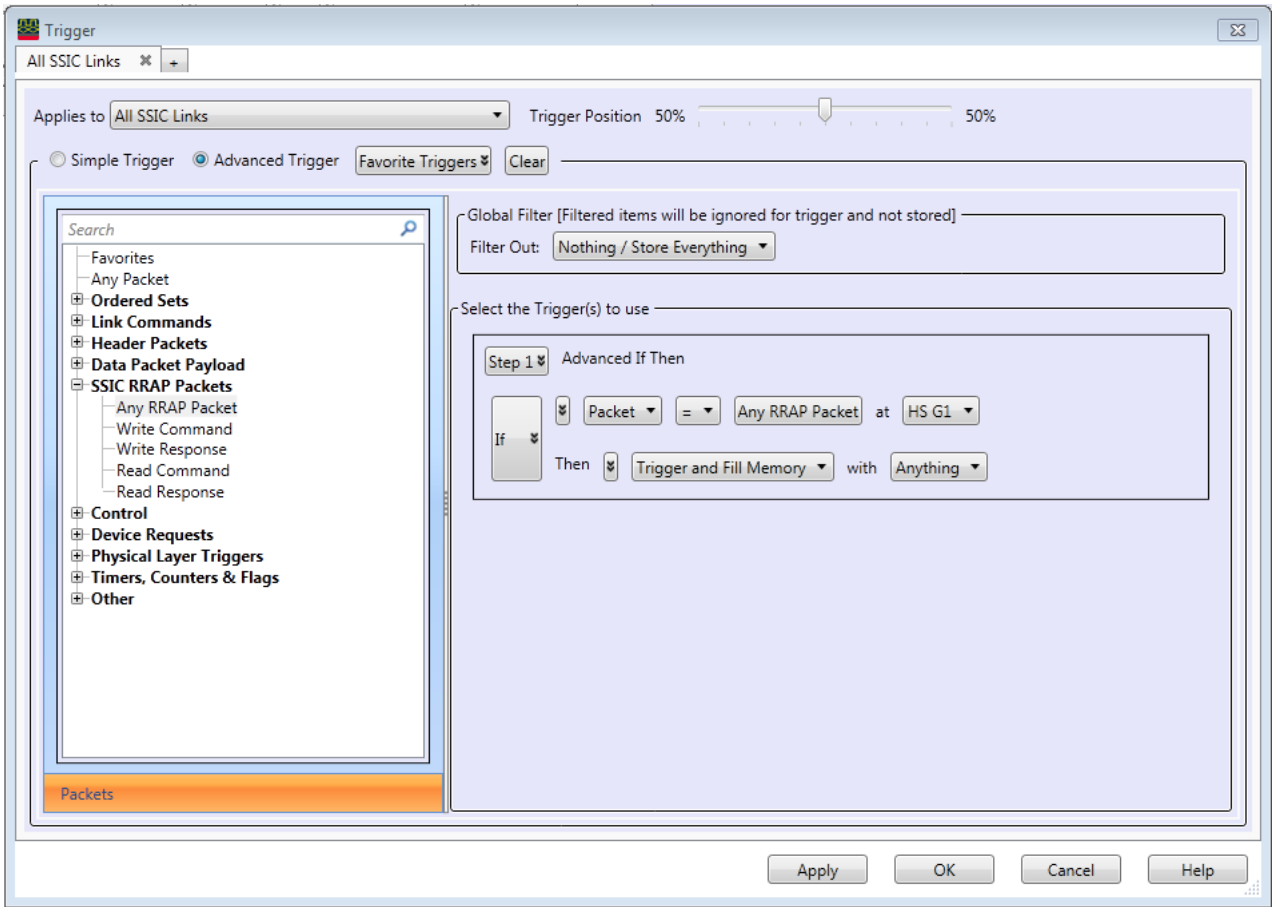
7 Click **Apply**.

8 To save the currently configured trigger setup in a Trigger Setup (.trg) file, click the **Favorite Triggers** button and then select the **Save** option from the displayed drop-down menu.

9 Click **OK**.

Trigger Example 1

The following trigger setup aims at triggering the U4431A module on the transmission of any RRAP SSIC packet at HS Gear 1 speed.

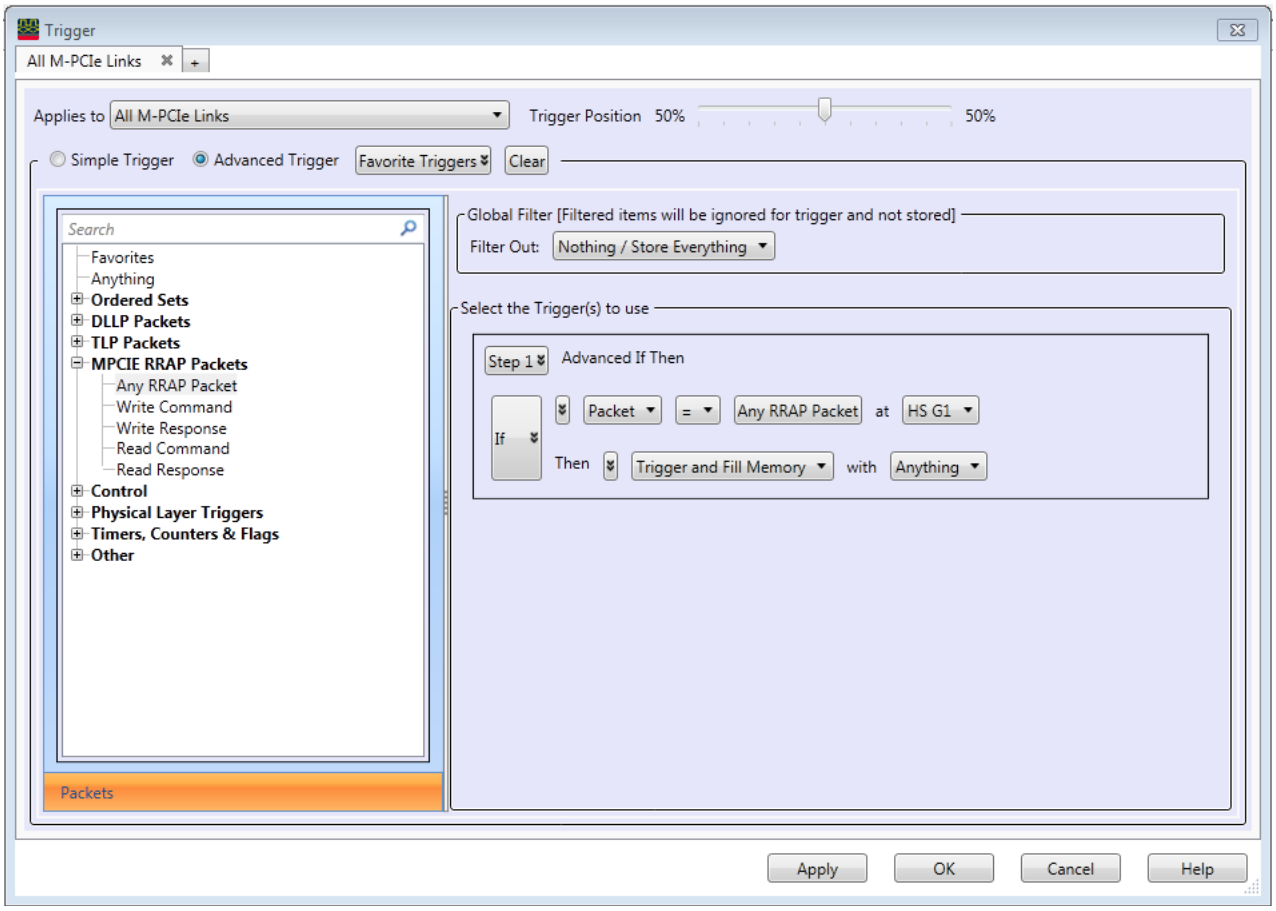

Trigger
All SSIC Links
Applies to All SSIC Links
Trigger Position 50%
50%
Simple Trigger
Advanced Trigger
Favorite Triggers
Clear
Search
Favorites
Any Packet
Ordered Sets
Link Commands
Header Packets
Data Packet Payload
SSIC RRAP Packets
Any RRAP Packet
Write Command
Write Response
Read Command
Read Response
Control
Device Requests
Physical Layer Triggers
Timers, Counters & Flags
Other
Packets
Global Filter [Filtered items will be ignored for trigger and not stored]
Filter Out: Nothing / Store Everything
Select the Trigger(s) to use
Step 1
Advanced If Then
If
Packet
=
Any RRAP Packet
at
HS G1
Then
Trigger and Fill Memory
with
Anything
Apply
OK
Cancel
Help

Trigger Example 2

The following trigger setup aims at triggering the U4431A module on the transmission of any RRAP M-PCIe packet at HS Gear 1 speed.

NOTE

In case of MPCIe, setting a trigger on **Start of Burst** packet would ensure that the U4431 module is triggered on the transmission of **Start of Burst** as well as **TS1 with SOB** packets. Similarly, during the Search process if you search for any **Start of Burst** packets the search results would display even **TS1 with SOB** packets also.

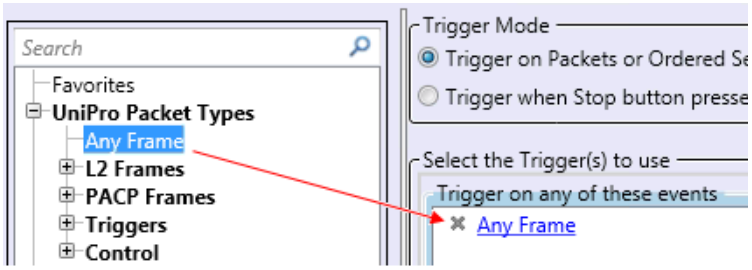
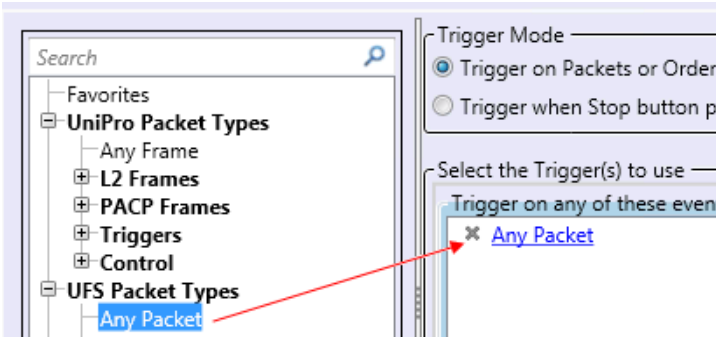


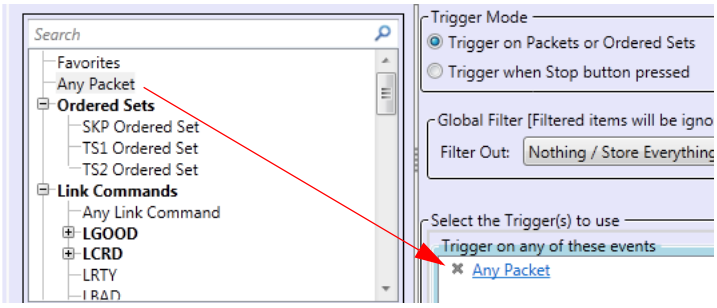
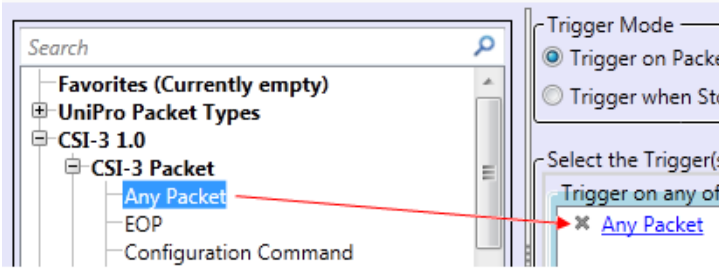
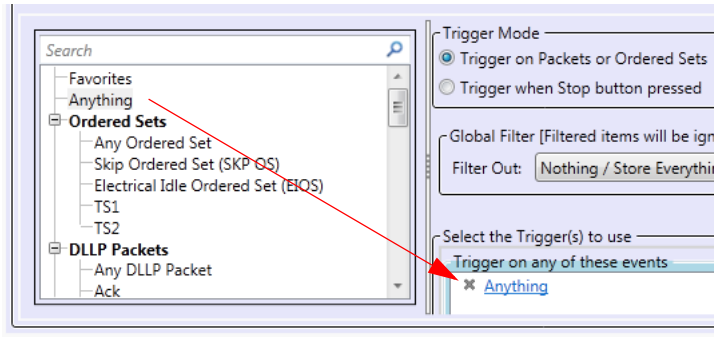
Triggering Based on Packet Types

You can configure the U4431A module to trigger on any or specific packet types. The packet types displayed in the simple and advanced trigger dialogs depend on the license(s) that you have installed. For instance, if you have the SSIC license installed, the SSIC-specific packet types are displayed. In case you have the UniPro and UFS license installed, the packet types for both UniPro as well as UFS are displayed.

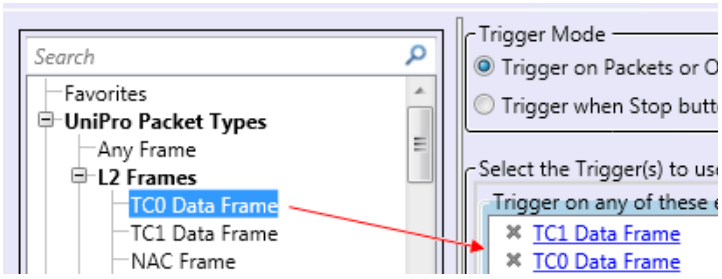
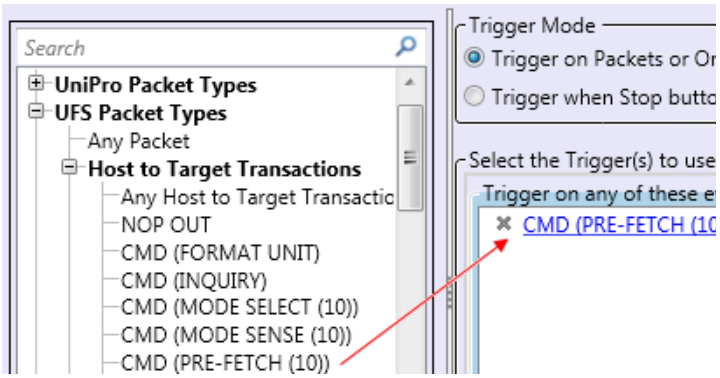
In a simple trigger

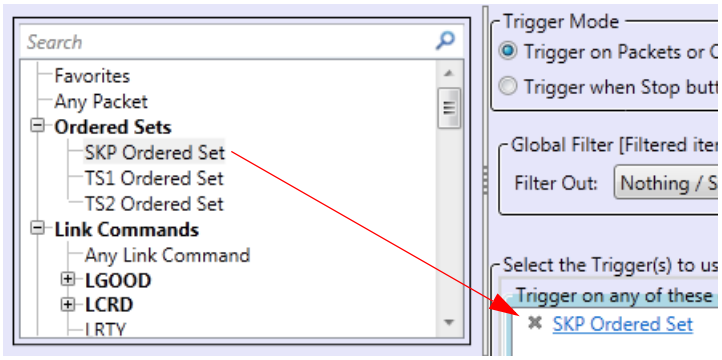
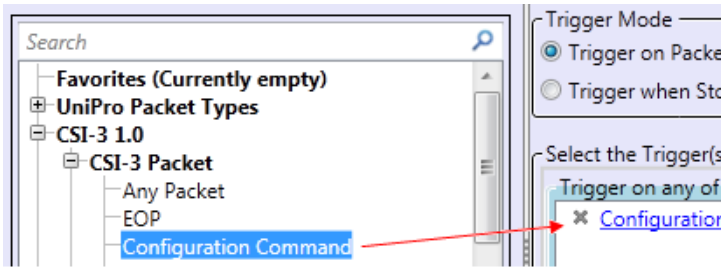
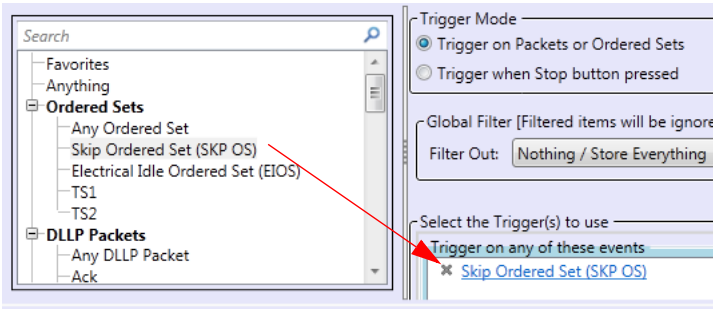
To trigger on “any packet” of the applicable protocol

Protocols	Description	
For UniPro	Drag and drop the Any frame option from the **UniPro Packet Types** list in the left pane of the **Trigger** dialog.	
For UFS	Drag and drop the Any Packet option from the **UFS Packet Types** list in the left pane of the Trigger dialog.	

Protocols	Description	
For SSIC	Drag and drop the **Any Packet** option from the left pane of the Trigger dialog.	
For CSI-3	Drag and drop the **Any Packet** option from the **CSI-3 1.0** list in the left pane of the Trigger dialog.	
For M-PCIe	Drag and drop the **Anything** option from left pane of the Trigger dialog.	

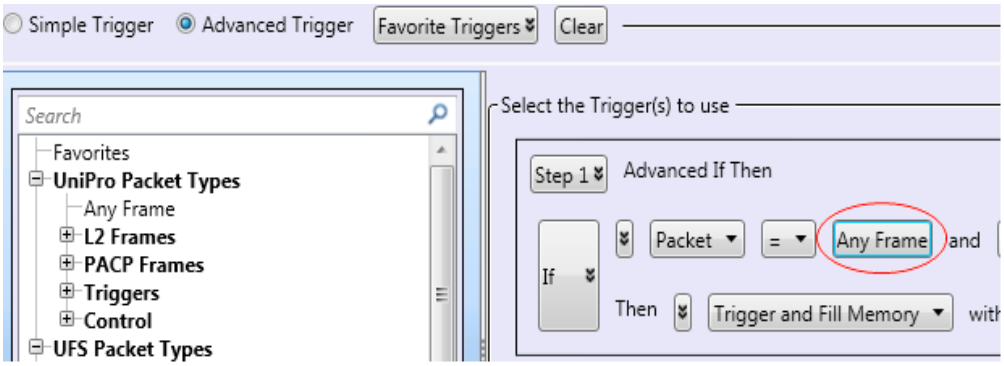
To trigger on specific packet(s) of the applicable protocol

Protocol	Description	
For UniPro	Drag and drop specific packet type(s) from the **UniPro Packet Types** list in the left pane of the Trigger dialog.	
For UFS	Drag and drop specific packet type(s) from the **UFS Packet Types** list in the left pane of the Trigger dialog.	

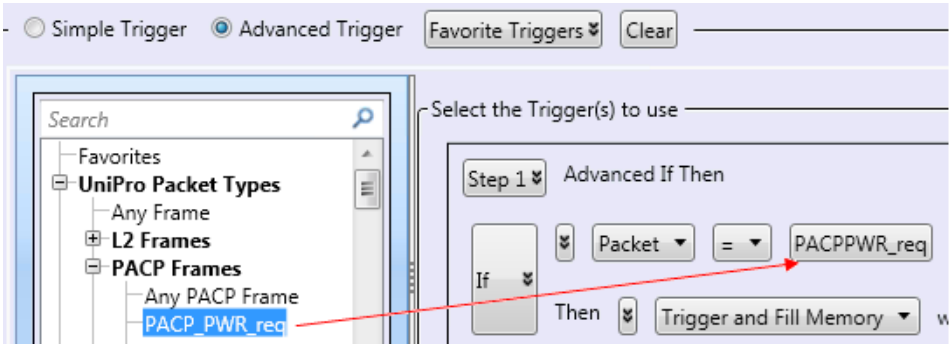
Protocol	Description	
For SSIC	Drag and drop specific packet type(s) from the left pane of the Trigger dialog.	
For CSI-3	Drag and drop specific packet type(s) from the **CSI-3 1.0** list in the left pane of the Trigger dialog.	
For MPCIe	Drag and drop specific packet type(s) from the left pane of the Trigger dialog.	

In an advanced trigger

If you have the UniPro and UFS license, by default, a step added in an advanced trigger has the Any Frame option selected as the packet type. Any Frame, in this case represents any UniPro frame.



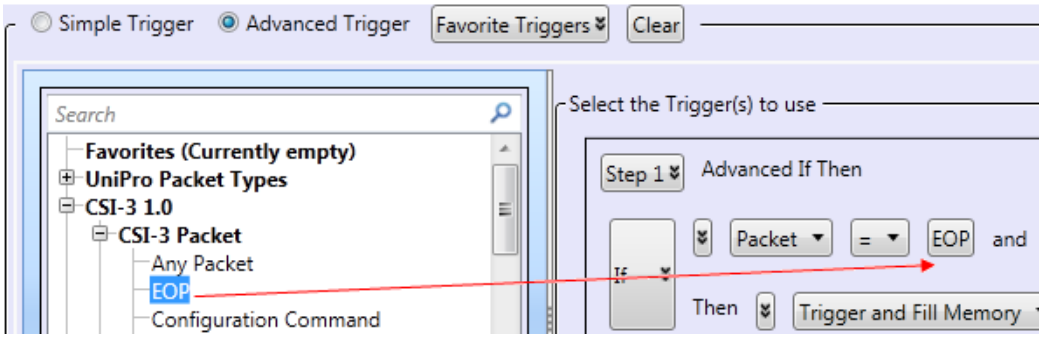
In an advanced trigger step, if you want to configure a trigger on a specific UniPro frame or a specific UFS packet, simply drag and drop that frame/packet from the **UniPro Packet Types**/ **UFS Packet Types** lists in the left to the **Any Frame** option in the step.



For SSIC, by default, a step added in an advanced trigger has **Any packet** selected as the packet type. Any packet, in this case represents any SSIC packet. If you want to trigger on a specific SSIC packet, you can drag and drop any SSIC packet from the left to the **Any Packet** option in the step.

For MPCIe, by default, a step added in an advanced trigger has Anything selected as the packet type. Anything, in this case represents any MPCIe packet. If you want to trigger on a specific MPCIe packet, you can drag and drop any MPCIe packet from the left to the Anything option in the step.

If you have the UniPro/CSI-3 license, by default, a step added in an advanced trigger has **Any frame** selected as the packet type. Any Frame in this case represents any UniPro frame. If you want to trigger on a specific UniPro frame or a specific CSI-3 packet, simply drag and drop that frame/packet from the **UniPro Packet Types**/ **CSI-3** lists in the left to the Any Frame option in the step.



Using Timers in Advanced Triggers

You can use timers in events and actions of sequence step(s) of an advanced trigger setup.

In a trigger setup, timers are used to check the amount of time that has elapsed between events. For example, if you want to trigger on an End of Burst packet that occurred within 800 ns of a Start of Burst packet, you can use a timer.

When using timers in a U4431 trigger setup, following are some points to remember:

- Timers do not start automatically. You need to start a timer by using the Start Timer action in the trigger setup before you can use the timer to check time condition in an event statement. (Refer to the timer example given below in this topic)
- Starting a timer does not reset the timer. It only starts/restarts the timer from its existing count. Therefore, you should reset the timer before starting it to get accurate triggering results. You use the Reset Timer action to accomplish this. (Refer to the timer example given below in this topic)
- The U4431 module provides two timers, Timer 1 and 2. Ensure that you use the correct timer in both event and action.

To insert a timer in an event statement

1 Drag the **Timer** event from the events list on the left of the Trigger dialog and drop it to the event statement on the right.

2 Select the timer operator (>= or <) and timer value.

To insert a timer action

1 Drag the **Timer** event from the events list on the left of the Trigger dialog and drop it to the action statement on the right.

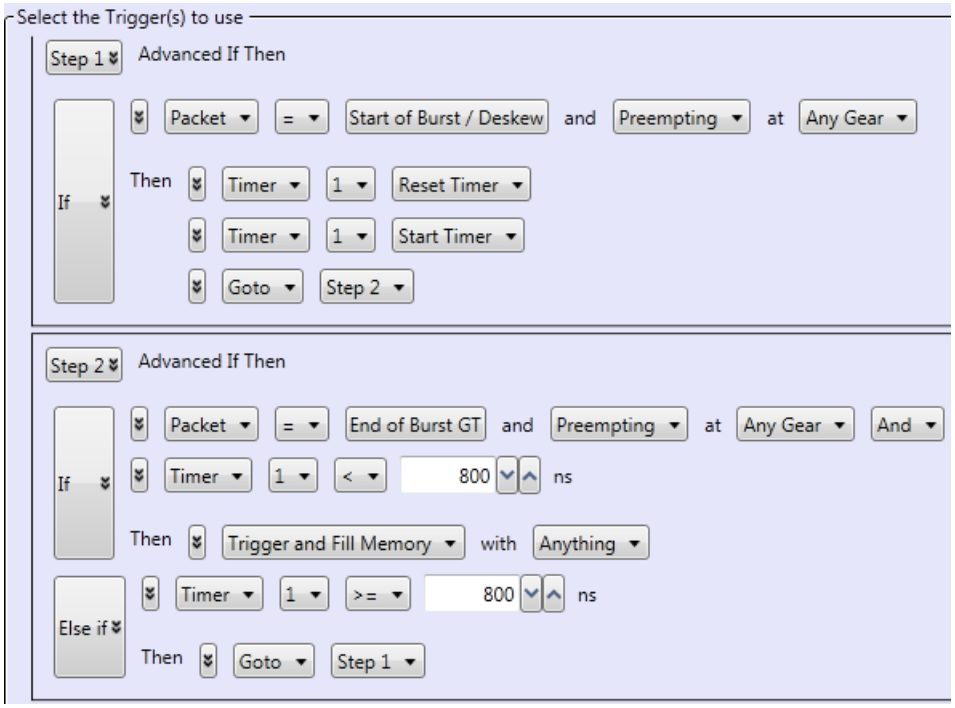
2 Select the timer action **Start Timer, Reset Timer** or **Stop Timer**.

Timer Example

The following example illustrates the usage of a timer to trigger on an End of Burst packet that occurred within 800 ns after a Start of Burst packet.

In step 1, the Timer 1 is first reset and then started when the event condition (Start of Burst packet) is met. In step 2, the Timer 1 is tested to check if an End of Burst packet occurred within 800 ns. If the condition is met, the module is triggered to start capture else the timer keeps running till it exceeds 800 ns without triggering.

On exceeding 800 ns without triggering, the Else If statement is executed to take the sequential flow of the trigger back to step 1 to again start looking for a Start of Burst packet.



Starting and Stopping the Data Capture

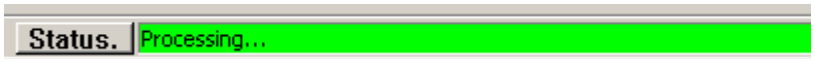
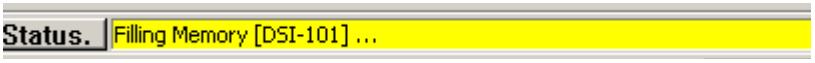
Once the capture setup and trigger setup are ready, you can start capturing the M-PHY data.

To start the data capture, choose **Run/Stop>Run** from the Logic and Protocol Analyzer GUI's menubar. Alternatively, click the  icon from the toolbar.

For more information, see the *Running/Stopping Measurements* topic in the *Logic and Protocol Analyzer Online Help*.

On starting data capture, the Status section at the bottom of the Logic and Protocol Analyzer GUI is updated to reflect the current status of the data capture procedure.

The status moves from the Processing, Running, Waiting, and finally to Filling Memory state.

Capture Status	Description
Processing	The module is processing the data capture settings to initiate data capture. 
Running	The U4431A module is now in the Running state to capture M-PHY data. 
Waiting	The U4431A module is waiting for the trigger condition to meet to start data capture. 
Filling Memory	The trigger condition is met. The U4431A module is now triggered to start storing the captured data in its memory. 

To stop the data capture, click the  icon from the toolbar.

5 Viewing and Analyzing Captured Data

This chapter provides information on how to view and analyze the captured M-PHY data for testing and debugging purposes using various viewers available in the Keysight Logic and Protocol Analyzer GUI.

Overview

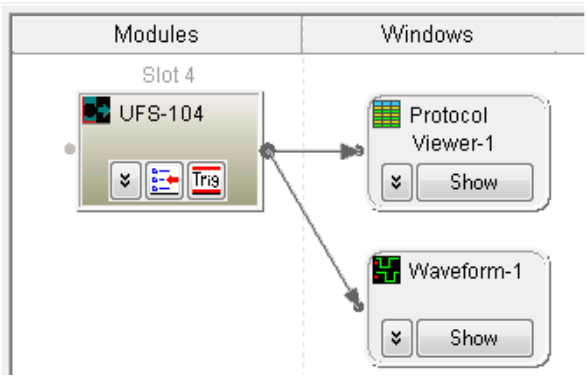
After you have captured M-PHY data using the U4431A module, you can view and analyze this data in the Keysight Logic and Protocol Analyzer GUI in the following two modes:

- Online - In the Online mode, the U4431A hardware is still connected and needed for viewing the captured data.
- Offline - For the Offline mode, you can save the captured data and configurations in a Logic Analyzer configuration file (.ala extension). You can later load this file in the Logic and Protocol Analyzer GUI to view and analyze data offline (without the U4431A module connected).

You can also export the captured data to CSV format files for offline viewing and analysis in applications other than the Logic and Protocol Analyzer GUI.

For viewing the captured data in online or offline mode, various viewers are available in the Logic and Protocol Analyzer GUI. Each viewer has its own set of features for the presentation of captured data and is useful for specific situations.

Viewer	Usage	Notes
Protocol Viewer	To view protocol level packet data. Packet details are organized and viewable in different forms using different views available in Protocol Viewer	An instance of Protocol Viewer is automatically added and available in the Logic and Protocol Analyzer GUI when you create a new session of the U4431A module.
Waveform Viewer	To view both packet as well as raw signal data.	An instance of Waveform Viewer is automatically added and available in the Logic and Protocol Analyzer GUI when you create a new session of the U4431A module.
Listing	To view raw signal level data	If needed, you can add instance(s) of this viewer to the U4431A module by selecting *New Window* > *Listing* option in the Overview pane.



In the above screen, a Protocol Viewer and a Waveform Viewer are already added on starting a new session. A Listing viewer can be added manually.

The topics that follow describe how to use each of the available viewers for viewing and analyzing captured data.

NOTE The information about viewers in the following topics is specific to viewing and analyzing M-PHY data. To get general information about a viewer, its fields, or how to use it, refer to the topics in **Reference > Windows** section of the Logic and Protocol Analyzer Online Help.

Viewing M-PHY Packet Data Using the Protocol Viewer Display

NOTE

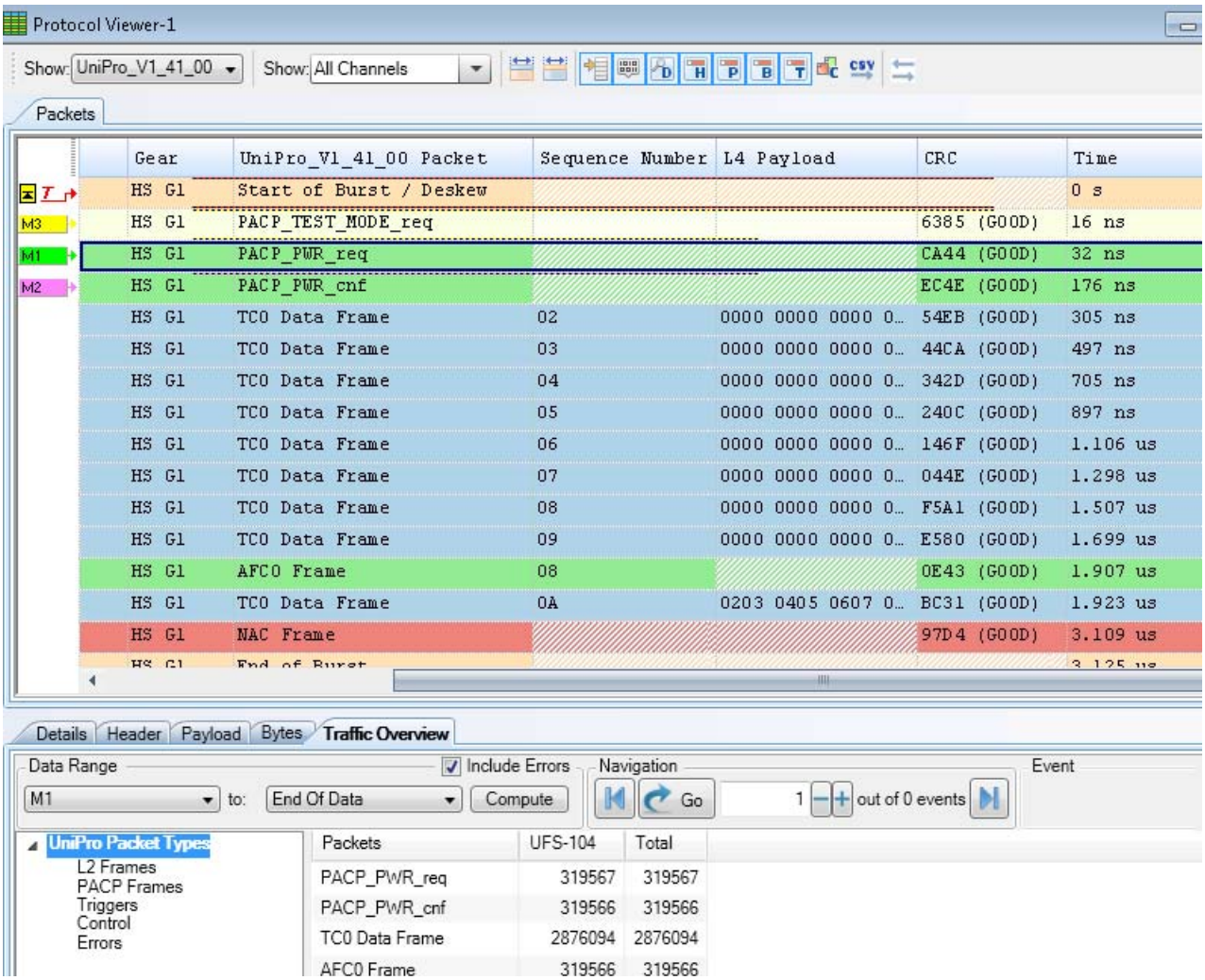
The information about the Protocol Viewer in this topic is specific to viewing and analyzing M-PHY packets. To get general information about the Protocol viewer, its fields, or how to use it, refer to the following topics in the Logic and Protocol Analyzer Online Help.

- Reference > Windows > Protocol Viewer Display Window

- Analyzing the Captured Data > Analyzing Packet Data

The Protocol Viewer window provides various ways of viewing the captured M-PHY packet data. You can view summarized as well as detailed packet information at the same time within the upper and lower panes in this window.

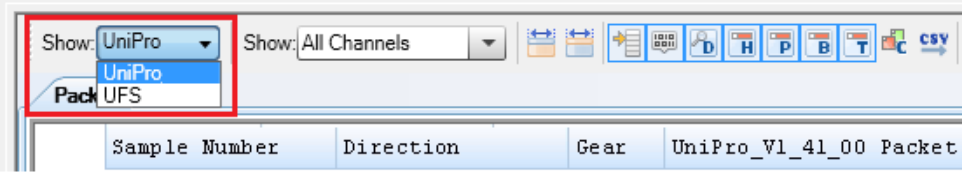
The upper pane of the Protocol Viewer displays a summarized list of captured M-PHY packets. The Protocol Viewer window is customized for the protocol family being decoded. For an M-PHY packet, it displays the M-PHY related decoded fields. In the following screen, the captured packet details specific to the UniPro protocol are displayed.



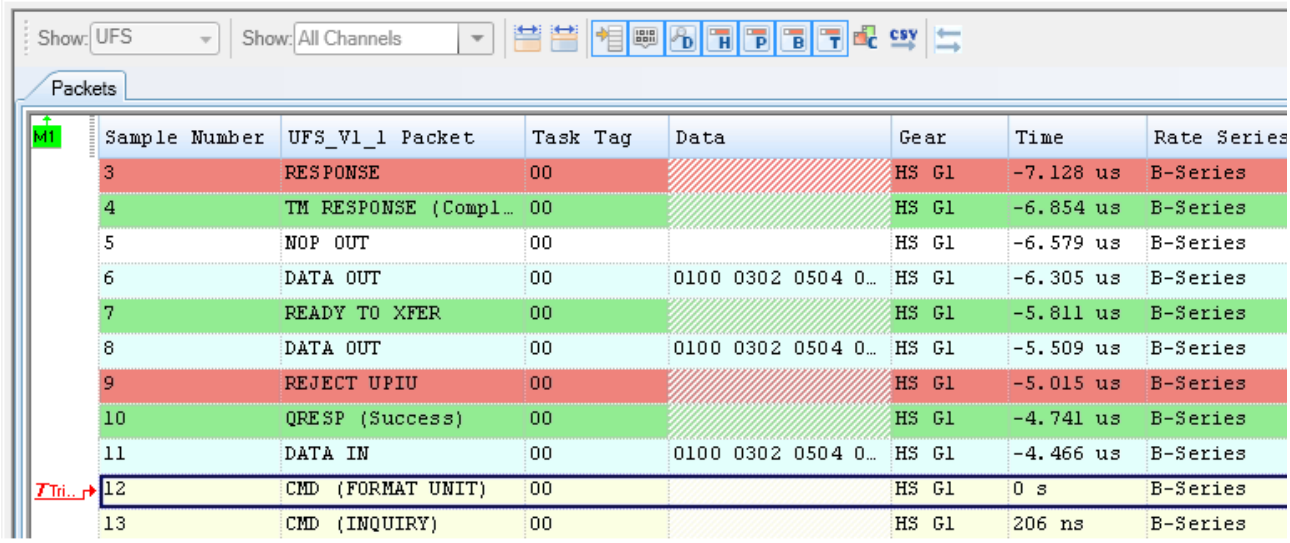
Viewing Packet Data Specific to a Protocol

The U4431A module supports acquisition and decoding of the M-PHY, UniPro, UFS, SSIC, MPCIe, and CSI-3 protocols depending on which licenses you have installed. From the captured data, if required, you can view packet data specific to a U4431A supported protocol in the Protocol Viewer.

The Show listbox at the top in the Protocol Viewer displays the currently active licensed protocol options.



Selecting a protocol option from this listbox displays the captured data specific to only that protocol in the Protocol Viewer window. In the following screen, selecting the UFS protocol from the Show listbox displays the UFS specific captured data.

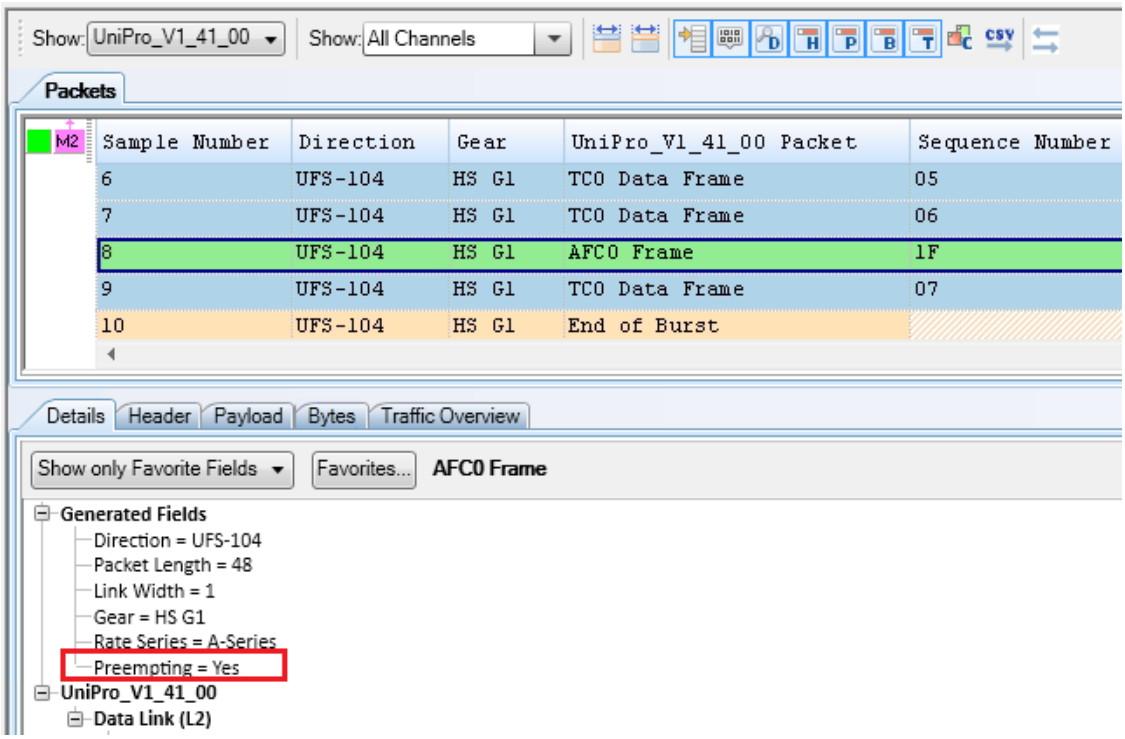


Viewing Preempted Data

If you enabled preemption support for the U4431A module in the Analyzer Setup tab, you can view the captured frames that have been preempted by the module in the Protocol Viewer. If the preemption support is not enabled for the module, then for all the captured frames, the Preempting field is set to No to indicate that the frames have not been preempted.

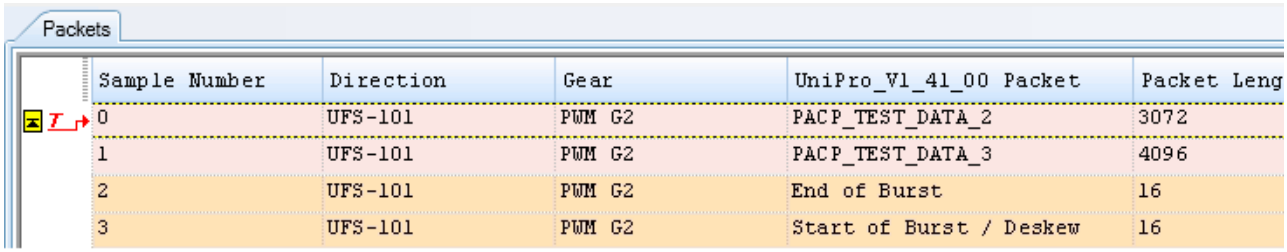
The preemption support is not applicable for SSIC and MPCIe setups.

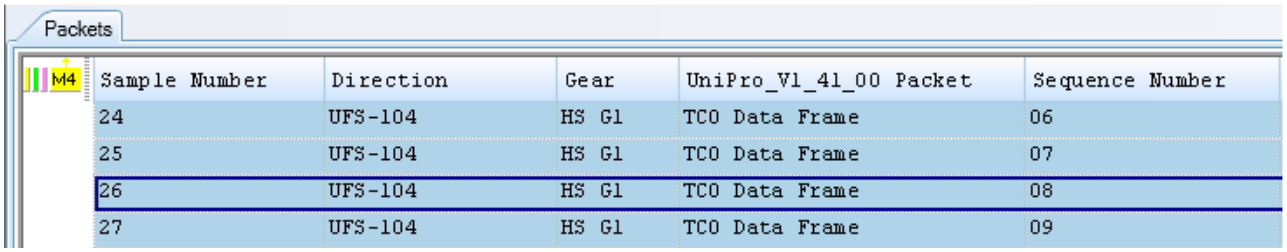
The following screen displays a sample preempted frame with the Preempting field set to Yes to indicate that the frame has been preempted.



Identifying HS and PWM Data Transmissions

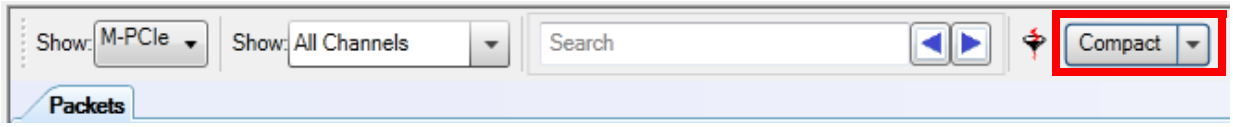
From the captured trace, you can identify the transmission in HS or PWM modes. The **Gear** column in the upper pane of the Protocol Viewer displays the burst mode and gear applicable for each packet in the captured trace.





Compacting the Display of Training Sequence Packets

By default, the captured training sequence packets are listed individually in the upper pane of the Protocol Viewer. There are situations when there are numerous training sequence packets of a particular type, say TS1, exchanged continuously over a period of time. In such situations, the listing of packets in Protocol Viewer can become clogged with these training sequences thereby requiring you to scroll significantly to view other types of packets. To overcome this, you can use the **Compact** tool that compacts the display of training sequence packets into sets in Protocol Viewer. This tool is available in the Protocol Viewer toolbar as displayed in the screen below.

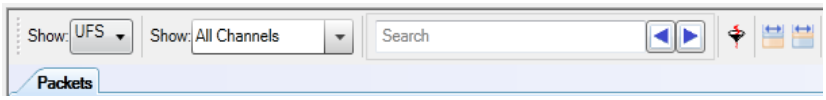


NOTE

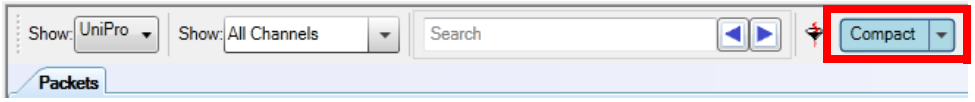
When using the U4431A module, the Compact Tool is available for the following protocols:

- UniPro
- SSIC
- M-PCIe

When you select a protocol such as UFS and CSI-3 from the **Show** listbox in the **Protocol Viewer**, the **Compact** button gets disabled as the Compact tool is not supported for this protocol.



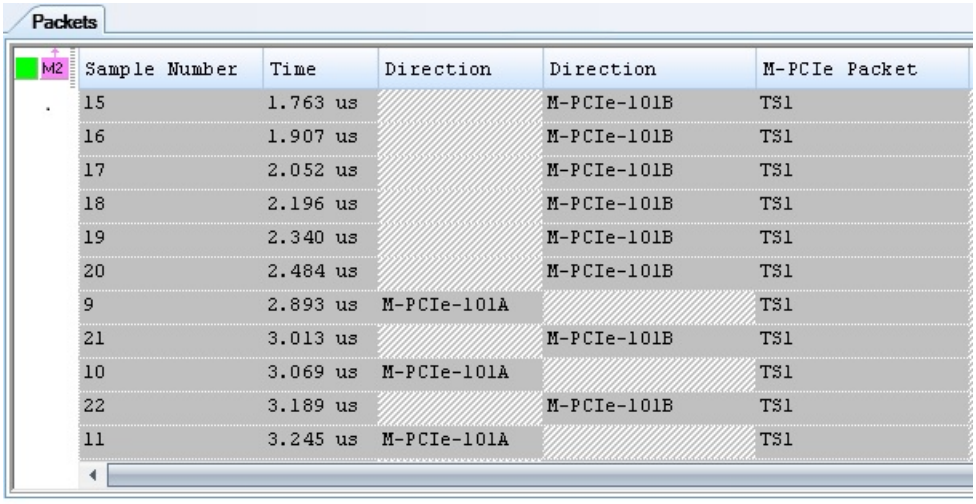
However, when you select a protocol such as UniPro for which the Compact tool is supported, the **Compact** button is displayed as Enabled.



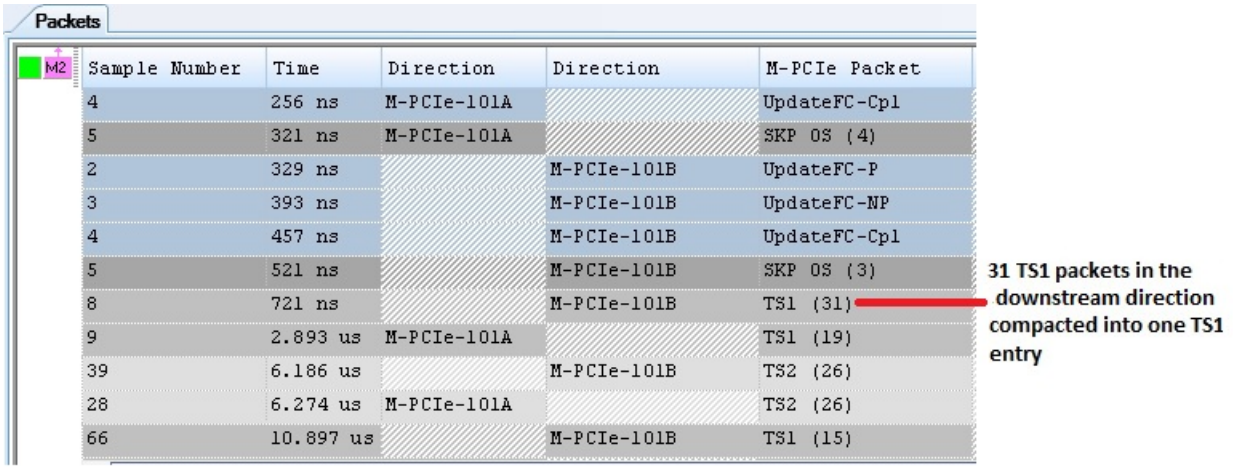
On enabling Compact, the training sequence packets of the same type, transmitted in the same direction in a time sequence are compacted into sets thereby reducing the packet entries in the Protocol Viewer listing. The compacted view also provides you a clearer view of the transitions such as TS1 to TS2 occurring in the exchange of training sequence packets.

Example - Compacted Display

The screens below display the packets in the Protocol Viewer before and after the Compact tool is used. Notice that all the packets are listed individually before compact is enabled. After enabling compact, the TS1 and TS2 packets are compacted as specific sets per direction and are listed as one entry per set in the Protocol Viewer. The number of packets that have been compacted into one entry/set is represented through the numeric value displayed with the entry/set.



Before Enabling Compact



After Enabling Compact

To configure the Compact setup

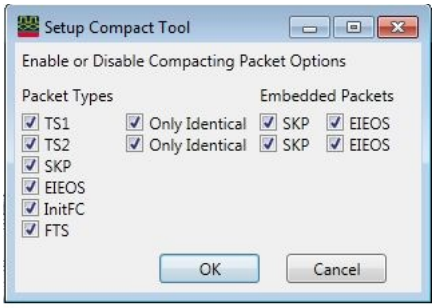
1 Click the drop-down arrow displayed with the **Compact** button and then select **Setup Compact**. The **Setup Compact Tool** dialog box is displayed.

2 Select the packet types that you want to compact into sets.

For **M-PCIe**,

- You can compact TS1, TS2, SKP, EIEOS, InitFC, and FTS packet types.

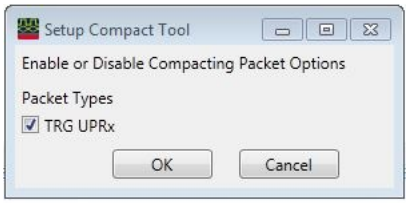
- For TS1 and TS2 packets, you can select whether or not only identical or all TS1 and TS2 packets should be compacted. Selecting the **Only Identical** checkbox compacts only identical TS packets (that is, packets with same field values). This allows you to view when a field of a TS packet changes compared to the previous TS packet.
- You can also choose whether or not the SKP and EIEOS packets embedded within the TS1 and TS2 packets should be compacted with the applicable TS1/TS2 packets. If you do not select these check boxes, the SKP and EiEOS packets appear as individual packets after compact.



M-PCIe

For **UniPro**,

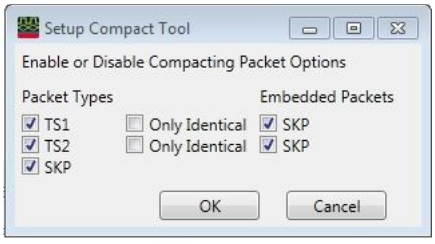
- You can compact TRG UPRx packet types.



UniPro

For **SSIC**,

- You can compact TS1, TS2 and SKP packet types.
- For TS1 and TS2 packets, you can select whether or not only identical or all TS1 and TS2 packets should be compacted. Selecting the **Only Identical** checkbox compacts only identical TS packets (that is, packets with same field values). This allows you to view when a field of a TS packet changes compared to the previous TS packet.
- You can also choose whether or not the SKP packets embedded within the TS1 and TS2 packets should be compacted with the applicable TS1/TS2 packets. If you do not select these check boxes, the SKP packets appear as individual packets after compact.



SSIC

3 Click **OK**.

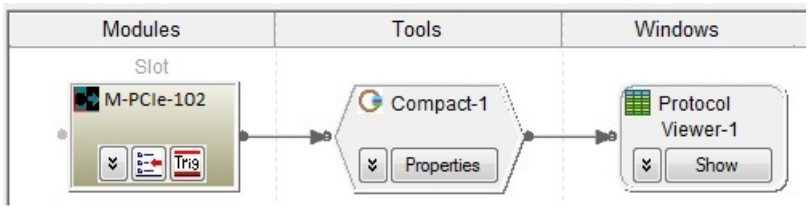
The compact setup is now configured as per your selections. When you enable compact, the packets will be compacted as per this configured setup. If the compact is already enabled, the changed setup configuration will be automatically reflected in the compacted packets display.

To enable the compact display

1 Click the drop-down arrow displayed with the **Compact** button and then select **Enable Compact**.

NOTE

When you enable the compact display for the first time, an instance of the Compact tool is added to your setup in the Overview window. Do not delete this tool's instance as this tool is used in the compact process.



To turn off the compact display

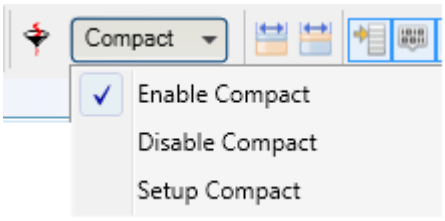
1 Click the drop-down arrow displayed with the **Compact** button and then select **Disable Compact**.

All packets are then listed individually.

NOTE

Clicking the Compact button flips the current display of packets from Compact to Not Compact and vice versa.

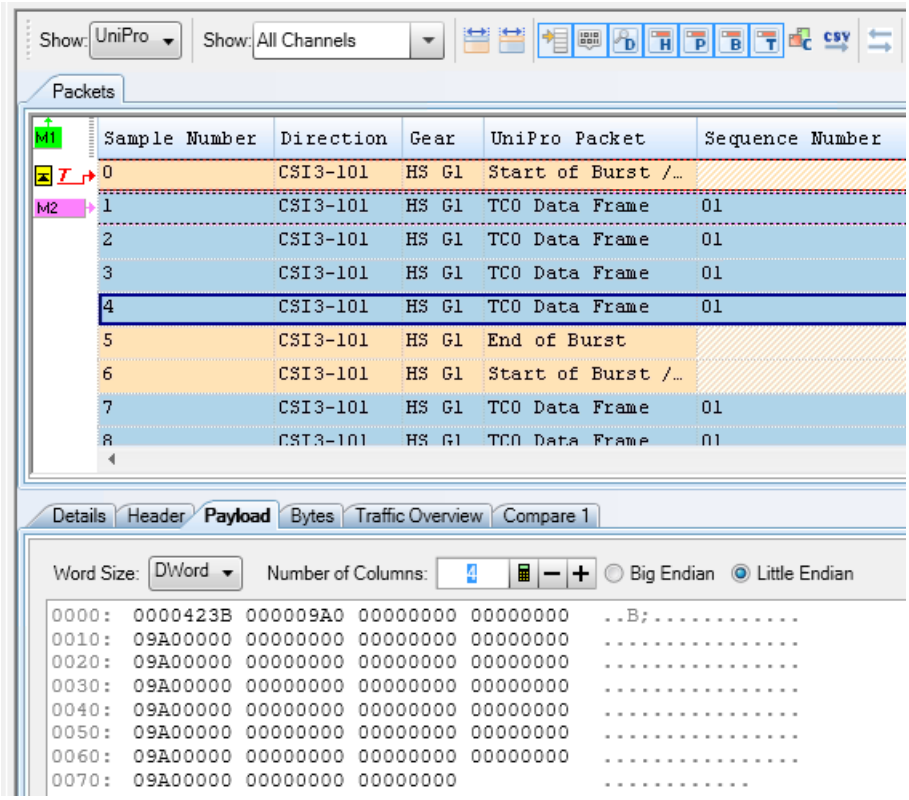
The Compact button's drop-down menu displays the current selection mode in terms of whether or not compact is currently enabled.



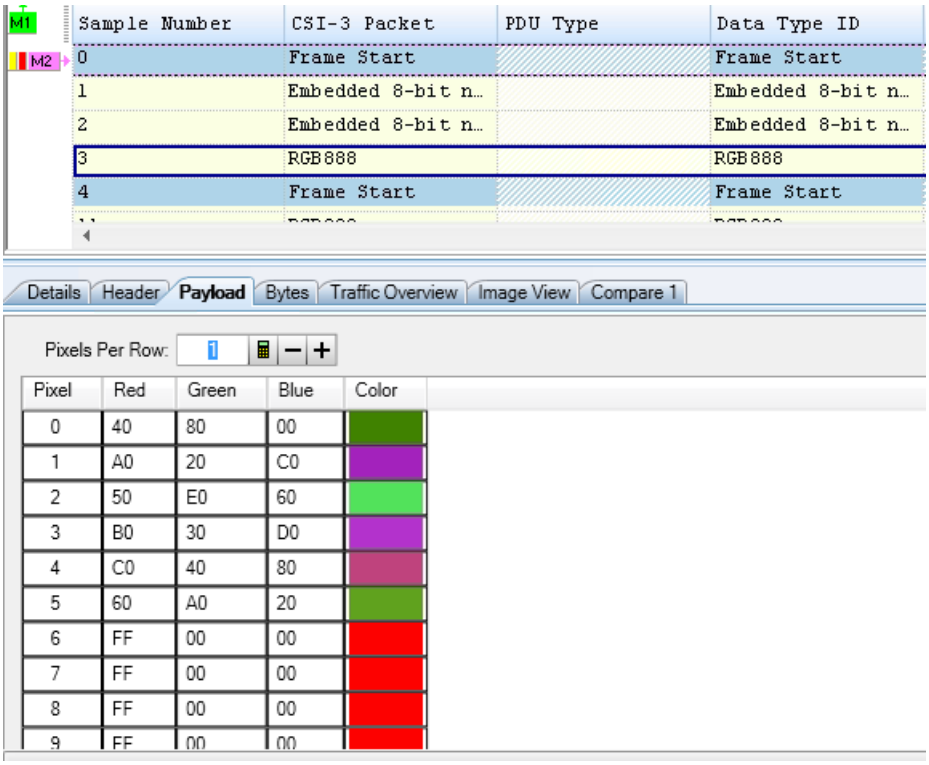
Viewing the Decoded Payload for a Packet

The **Payload** tab displays the decoded payload for the packet currently selected in the upper pane of the Protocol Viewer.

The following screen displays the payload of a UniPro TC0 Data Frame.

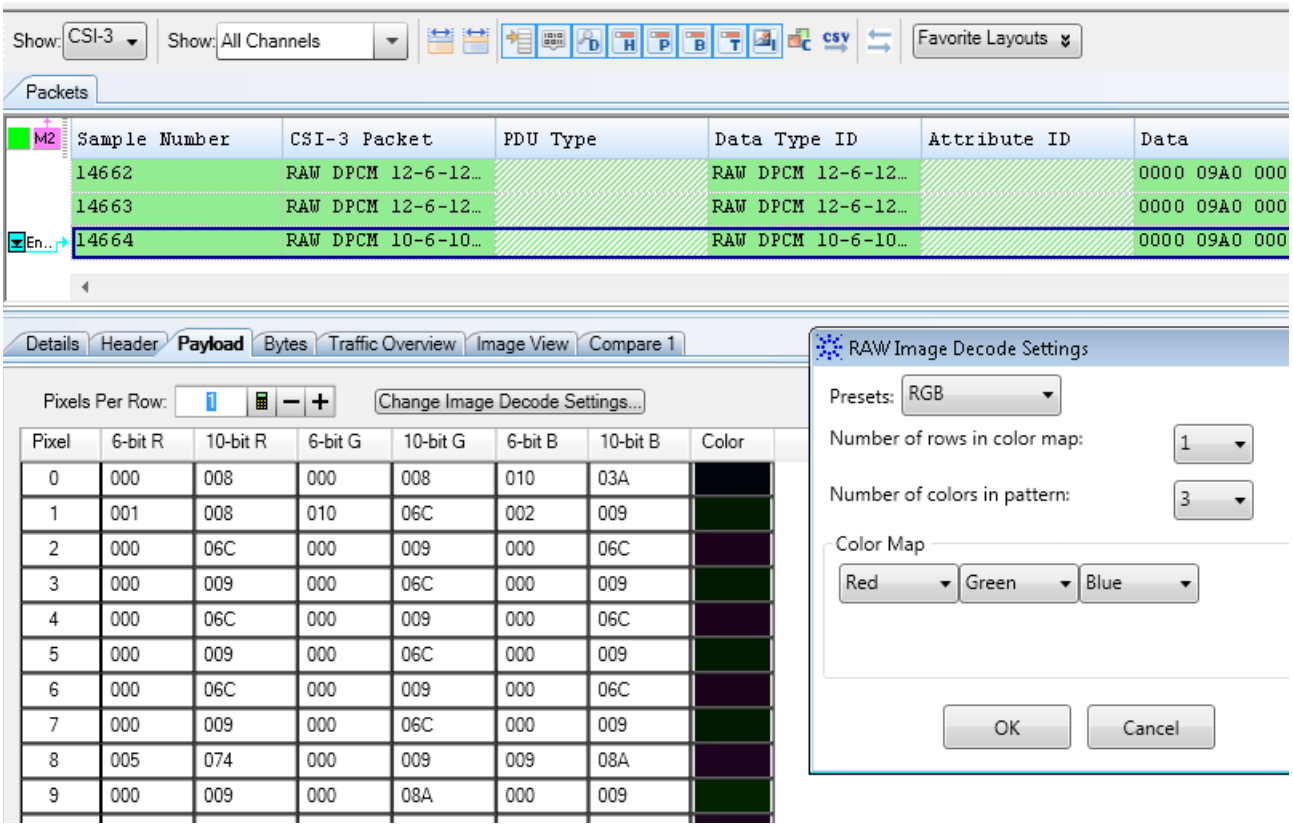


For CSI-3 image packets, the Payload tab displays the image's pixel bytes data. The payload of an RGB format CSI-3 image packet is shown below.



Changing Decode Settings for CSI-3 RAW Image Payload Data

For CSI-3 RAW image formats, the Payload tab also provides the **Change Image Decode Settings** button. By default, RGB presets are used for decoding the payload of CSI-3 RAW image formats. However, you can use the **Change Image Decode Settings** button to select different decode settings for the image payload data. Clicking this button displays the **Raw Image Decode Settings** dialog box.



For RAW formats, a number of preset options such as RGB, YMC, CMY, and Bayer decoding are available in the Raw Image Decode Settings dialog box. Based on the decode settings that you select in this dialog box, the image's payload data is recalculated in the Payload tab. The changed settings are also applicable while extracting an image in the **Image View** tab.

Viewing the Byte-wise Transmission of M-PHY Data

The **Bytes** tab in the lower pane of the Protocol Viewer provides a byte-wise view of the header and payload data of a packet.

The bytes data in this tab is spread across rows to display the bytes data for header followed by the bytes data for payload (if applicable).

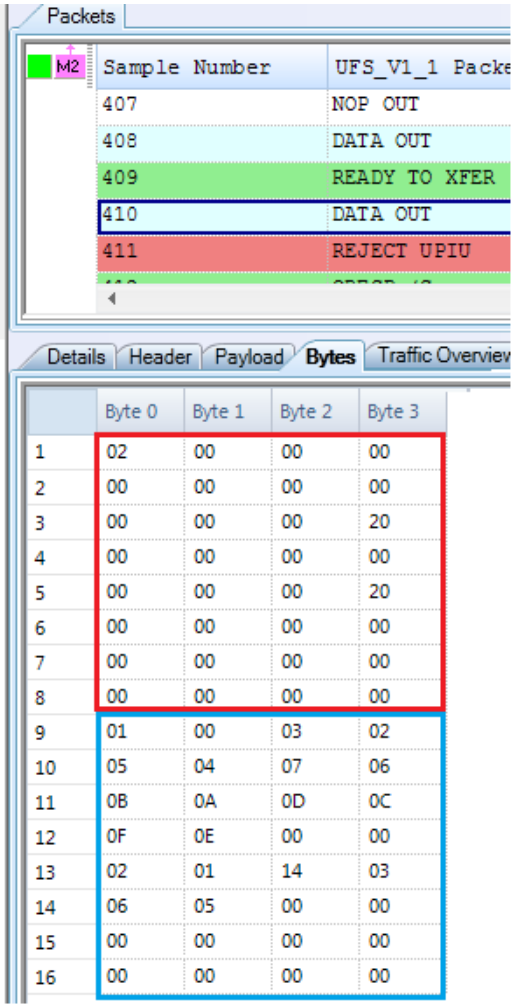
When you select a packet listed in the upper pane of Protocol Viewer, its raw data spread across bytes used for its transmission is displayed in the Bytes tab.

NOTE

The data displayed in the Bytes tab does not represent the lane-wise view of bytes. This tab does not show the bytes spread across the lanes on which the bytes were transmitted.

If you want to view the lane-wise transmission of raw data, you use the Waveform Viewer display. In this Viewer, you can see the raw data spread across the lanes used for its transmission. For this data to be acquired and displayed in Waveform Viewer, ensure that the **Raw Data** checkbox is selected in the **Analyzer Setup** dialog box prior to data acquisition. To know more, refer to the topic "Viewing the Lane-wise Transmission of Raw Data" on page 71.

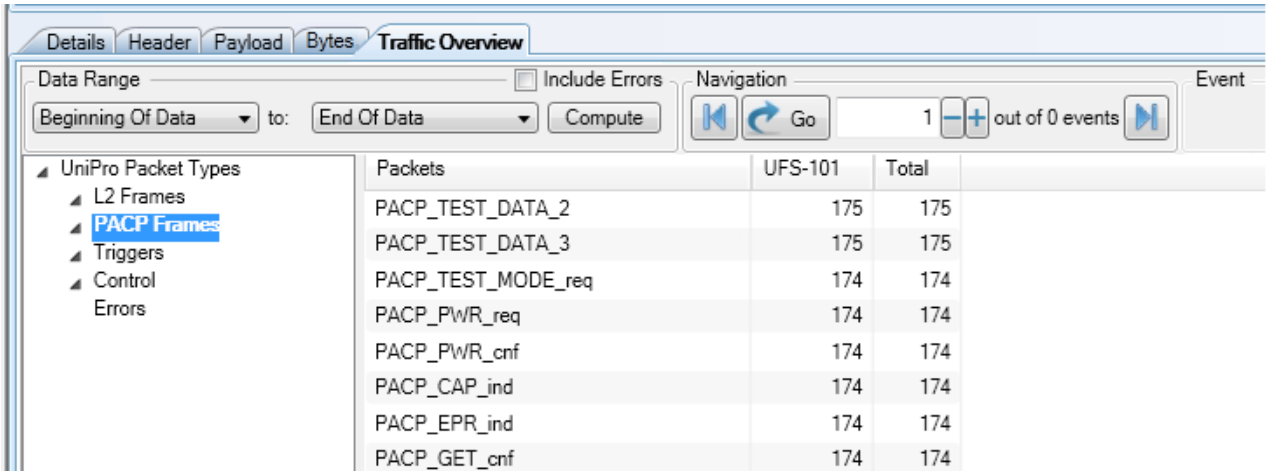
In the following figure, the data of a *DATA_OUT UFS* packet spread across the bytes used for its transmission is displayed. The red and blue highlights in this screen indicate the packet's header and payload data respectively.



NOTE You can toggle the visibility of the Bytes tab using the **Toggle Byte Pane Visibility** toolbar button displayed in the Protocol Viewer toolbar.

Viewing the Captured M-PHY Traffic Statistics

The Traffic Overview tab in the lower pane of the Protocol Viewer provides an overview of the M-PHY traffic listed in the upper pane. It provides a count of various types of captured M-PHY packets categorized on the basis of the type of packet. It also displays the count of M-PHY packet errors.



To view M-PHY traffic statistics

1 Click the **Traffic Overview** tab.

2 In the **Data Range** group box, specify the range of data (from the upper pane) for which you want to compute and display traffic statistics. You can also select markers set in the upper pane for defining the data range.

3 Select the **Include Errors** checkbox if you want a count of errored packets to be included and displayed in the traffic statistics.

4 Click **Compute**.

Traffic statistics are displayed for the selected data range. The left pane lists the packet types for the applicable protocol. Selecting a packet type from this list displays the total number of packets in the data range for that packet type.

5 Use the **Navigation** section in the Traffic Statistics tab to navigate through the packets of a particular type in the upper pane.

a Select a packet type from the right pane of Traffic Statistics.

b In the Navigation section text field, specify the packet occurrence to which you want to navigate in the upper pane.

c Click **Go**.

The specified packet occurrence is highlighted in the upper pane.

NOTE The last packet type "**Errors**" listed in the left pane displays the count for the errored packets categorized on the basis of errors types. This error count is displayed only if you selected the **Include Errors** checkbox before computing traffic statistics.

Extracting Images from the Packet Data

You can use the **Image View** tab in the lower pane of the Protocol Viewer to display images constructed from the captured packet data. The Image View tab lists all the “Start of Frame” locations for images found in the captured data range specified by you. An image can be constructed from a selected “Start of Frame”.

NOTE

The **Image View** tab is available in the lower pane of the Protocol Viewer window only if the following two prerequisites are met

- the CSI-3+UniPro license is installed for the U4431A module.

- CSI-3 is selected as the protocol in the Show listbox at the top of the Protocol Viewer. For other protocols, the Image View is not applicable.

All data formats in the CSI-3 specifications are supported for image construction in the Image View tab.

To extract an image

1 Click the **Image View** tab.

2 In the **Data Range** fields, select the range of the captured data from which the Start of Frame locations are to be displayed.

3 From the **Display Time as** section, you can choose to display the Start of Frame locations with their absolute time or the time relative to their previous frames. The default selection is **Absolute**.

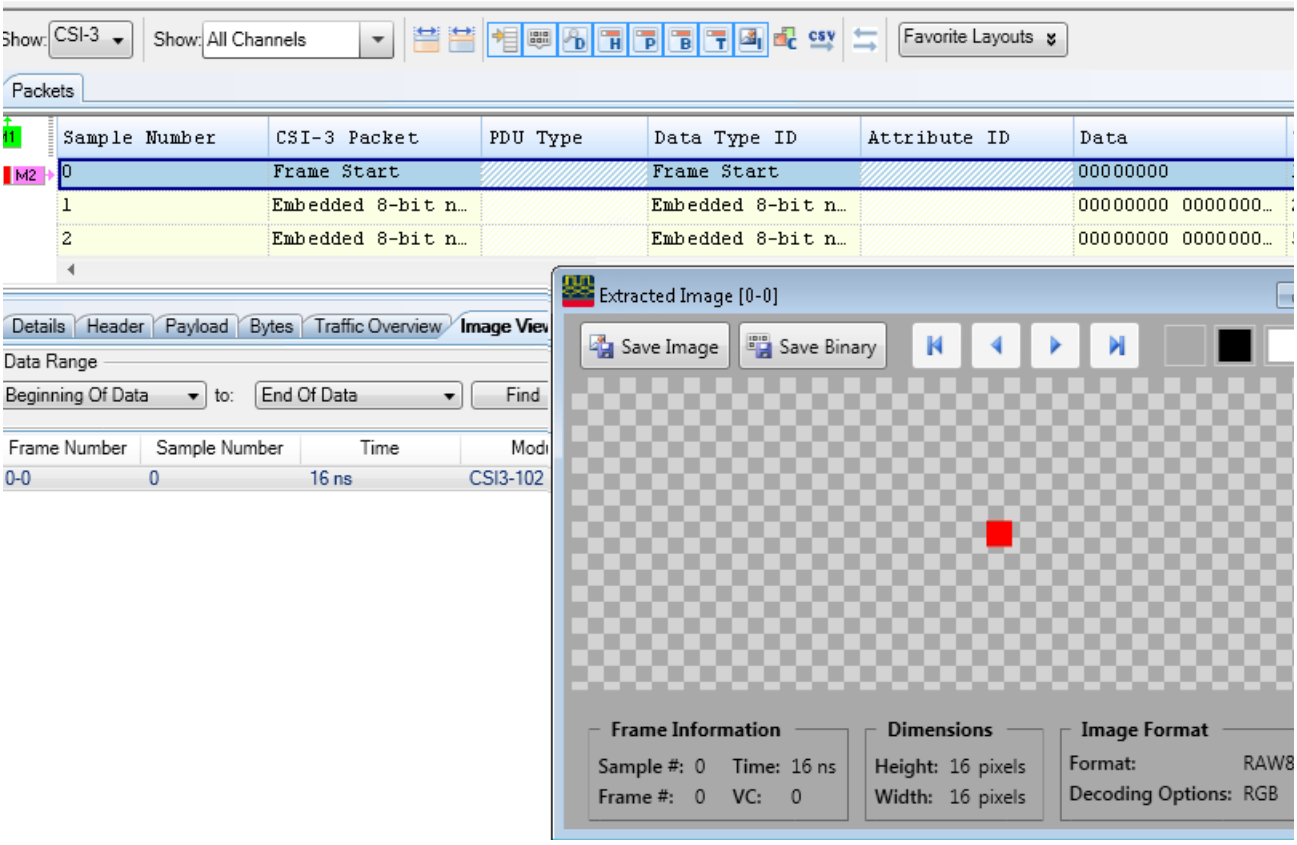
4 Click **Find**.

All the “Start of Frame” locations from the captured data as per the data range specified by you are listed in the Image View tab.

5 Select a start of frame from the displayed list and click the **Show Image** button to get an image constructed from the selected start of frame.

If you have the appropriate license, the image is constructed and displayed in the **Extracted Image** dialog box. The constructed image’s attributes such as width, height, format, and decoding options are also displayed.

A test image constructed from the start of frame 0 (sample number 0) is displayed in the following screen.



NOTE

In the **Extracted Image** dialog box, you can use the  buttons to construct and display an image from the first, next, previous or last Start of Image Frame locations currently listed in the Image View tab.

The Extracted Image dialog box also provides you four background patterns  for the displayed image.

Saving the Extracted Image

Once the extracted image is displayed in the Extracted Image dialog box, the following two options are available for saving this image:

- **Save Binary** - This button is displayed only if the extracted image is in a RAW format. Using this button, you can save the image's pixel bytes in a binary (.bin) file. This allows you to save the raw image payload data that was originally captured by the U4431A module. This raw image payload data excludes any modifications that are made as a result of the color information calculation and decoding the image.
- **Save Image** - This button is available for all image formats. It allows you to save the decoded image as a .bmp file. For an image with a RAW format, this button allows you to save the modified image that resulted from the color information calculations and decoding.

Changing the Decoding Options for the Extracted Image

By default, an image is decoded and displayed in the Extracted Image dialog box as per the image decoding options set in the **Payload** tab. If required, you can change the decoding options for the displayed image to redisplay it as per the changed decoding options.

To change the decoding options of an extracted image

1 Click the **Change the decoding options**  button displayed in the Extracted Image dialog box.

The **Raw Image Decode Settings** dialog box is displayed.

2 Change the decoding options as needed. To know more, refer to the topic "Changing Decode Settings for CSI-3 RAW Image Payload Data" on page 61.

Exporting the list of "Start of Frames" locations to a CSV File

You can also export the list of Start of Frames locations to a CSV file by clicking the **Export list to CSV** button. On clicking this button, a CSV file is generated. The list of "start of frames" displayed for the specified range of captured data is exported to this CSV file. The timestamps of the start of frames list exported to this CSV file are absolute regardless of the time stetting (**Absolute** or **Relative to previous frame**) that you have selected in the Image View tab.

The following screen displays a sample CSV file with the exported start of frames list.

1	Frame Nu	Sample N	Time (ns)	Module
2	0-0	0	0	CSI3-102
3	0-1	5748	1.48E+08	CSI3-102
4	0-2	8639	2.6E+08	CSI3-102

NOTE

If you want to export the protocol data displayed in the upper pane of the Protocol Viewer to a CSV file, you can use the **Export to CSV** toolbar button displayed at the top of Protocol Viewer.

The output CSV file has data matching the fields and their sequencing currently displayed in the upper pane of the Protocol Viewer.

	A	B	C	D	E	
1	Sample N	CSI-3 Packet	PDU Type	Data Type ID	Attribute	Data
2	0	Frame Start		Frame Start		0000 0000
3	1	Embedded 8-bit non Image Da		Embedded 8-bit n		0000 0000 000
4	2	Embedded 8-bit non Image Da		Embedded 8-bit n		0000 0000 000
5	3	RAW8		RAW8		0000 0000 000

Viewing Packet and Raw Signal Data using the Waveform Viewer

NOTE

The information about the Waveform Viewer in this topic is specific to viewing and analyzing M-PHY data. To get general information about the Waveform Viewer, its display, or how to use it, refer to the following topics in the Logic and Protocol Analyzer Online Help.

- Reference > Windows > Waveform Display Window

- Analyzing the Captured Data > Analyzing Waveform Data

You can view the captured M-PHY data (both packet data as well as raw signal data) as a digital waveform in the Waveform Viewer. The captured bus/signal data is displayed in a time based waveform as transmitted on the link. The Waveform view is useful in situations where you want to get an overview of the M-PHY link transitions.

NOTE

The display of raw data in the Waveform Viewer depends on whether or not you enabled the **Raw data capture** feature in the **Analyzer Setup** tab prior to data acquisition.

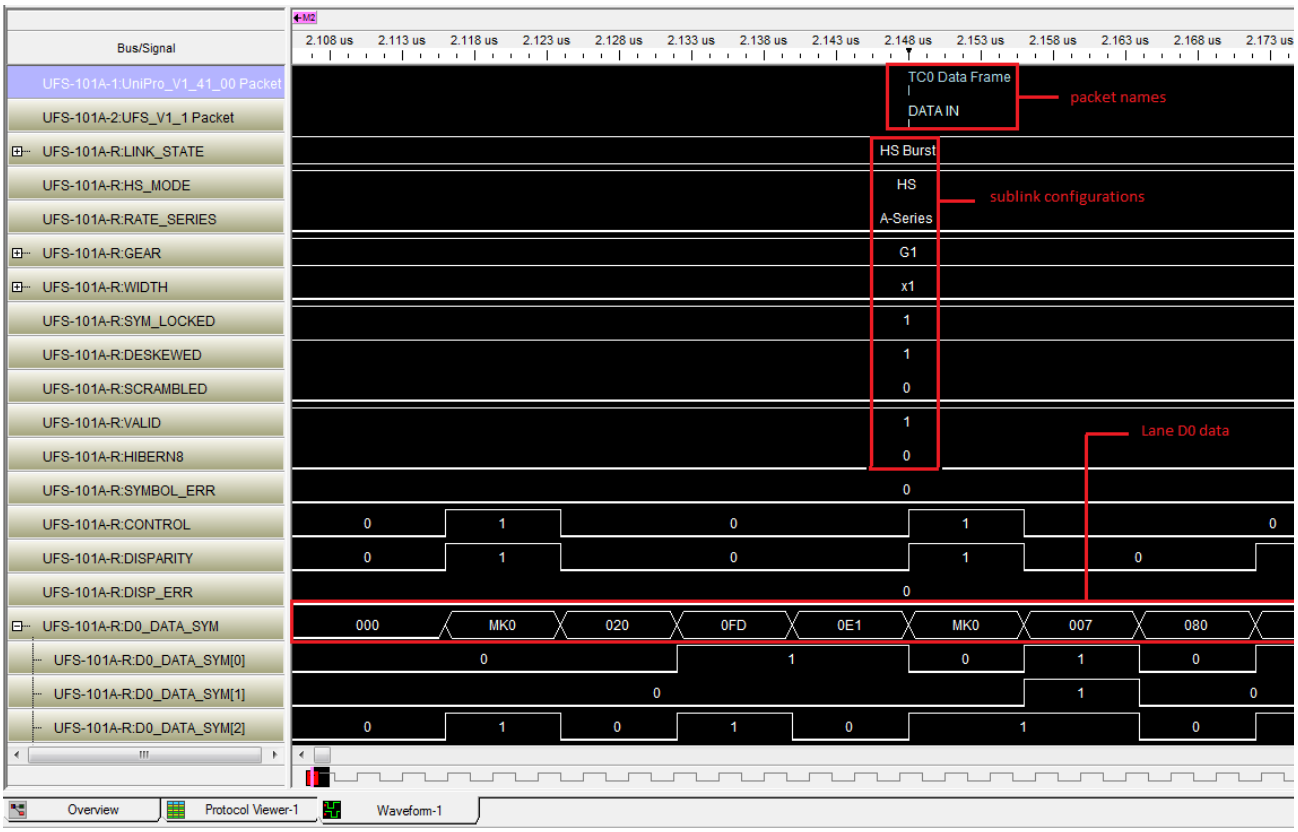
Understanding Waveform Viewer Display for M-PHY Data

A waveform display primarily displays the acquired M-PHY data as follows.

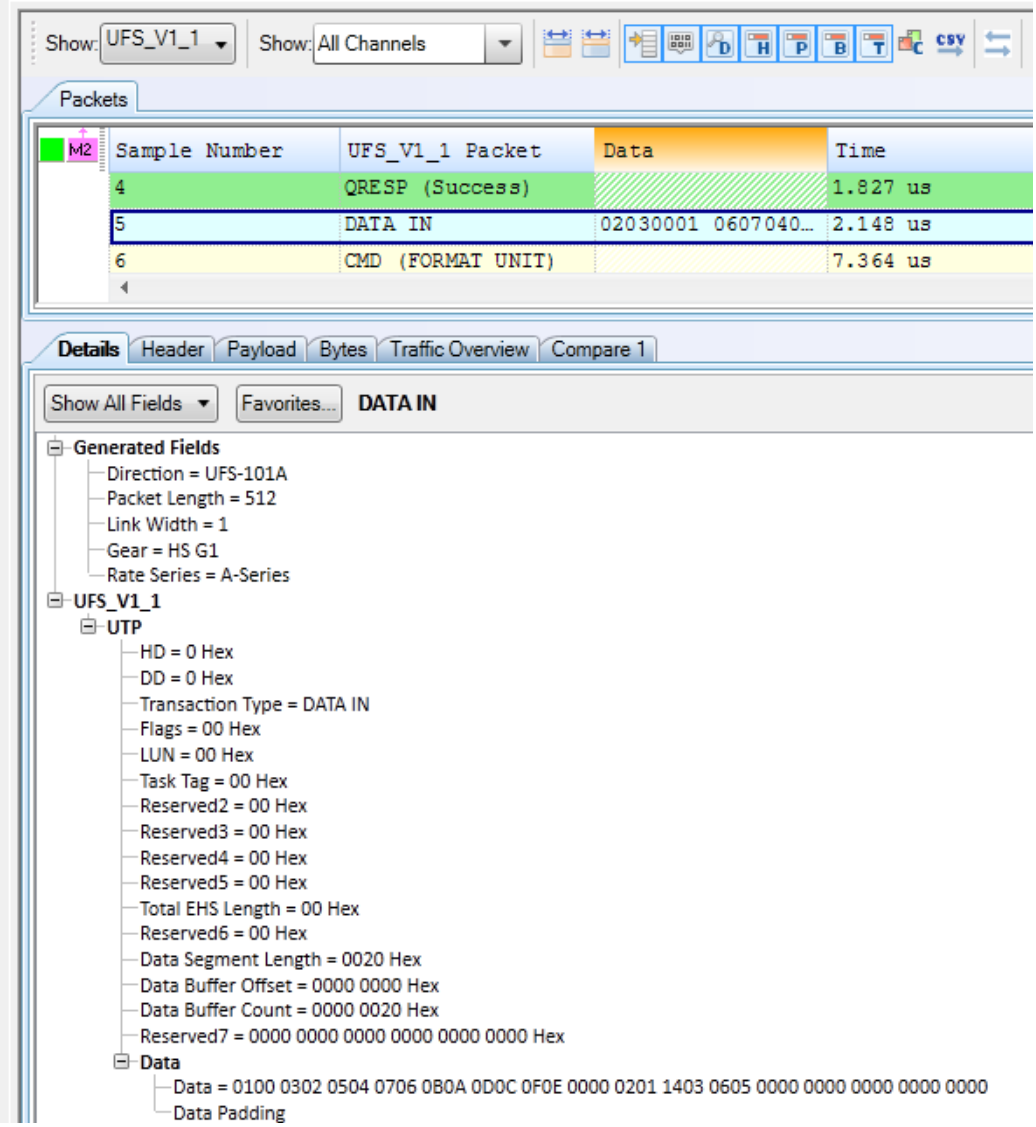
- The first row in the waveform area is the captured M-PHY/UniPro/SSIC packet name. In case of MPCIe, each packet is displayed in the form of a horizontal colored bar. Each packet is identified by a separate color across the timeline and you can correlate the same color both in Waveform and Protocol viewer.
- The second row in the waveform area is the applicable UFS/CSI-3 packet name if the UFS/CSI-3 data is captured. For a bidirectional link, this is followed by the captured M-PHY/UniPro/SSIC packet name in the other direction.
- After the packet name(s), various link configuration settings as detected across the timeline are displayed such as:
  - the current state of link
  - burst mode
  - rate series
  - link width
  - whether or not data is deskewed (1 represents deskewed and 0 represents skewed)
  - whether or not data is scrambled (1 represents descrambling is enabled for scrambled data and 0 represents descrambling is disabled)
  - whether or not there is a symbol error (1 represents a symbol error in the data and 0 represents no symbol error)
  - whether or not there is a disparity error (1 represents a disparity error in the data and 0 represents no disparity error)
- The link configurations are followed by the captured lane-wise signal data values for a sublink. This includes symbol, 8b encoded, and 10b encoded data. For a bidirectional setup, the data for one sublink is followed by the data for the other sublink.

If needed, you can hide a specific information about the captured data from the waveform display by right-clicking on the information row on the left and selecting **Delete Row**.

In the following screen, the data for the **DATA IN** UFS packet (**TC0 Data Frame** UniPro packet) is displayed in the Waveform Viewer.



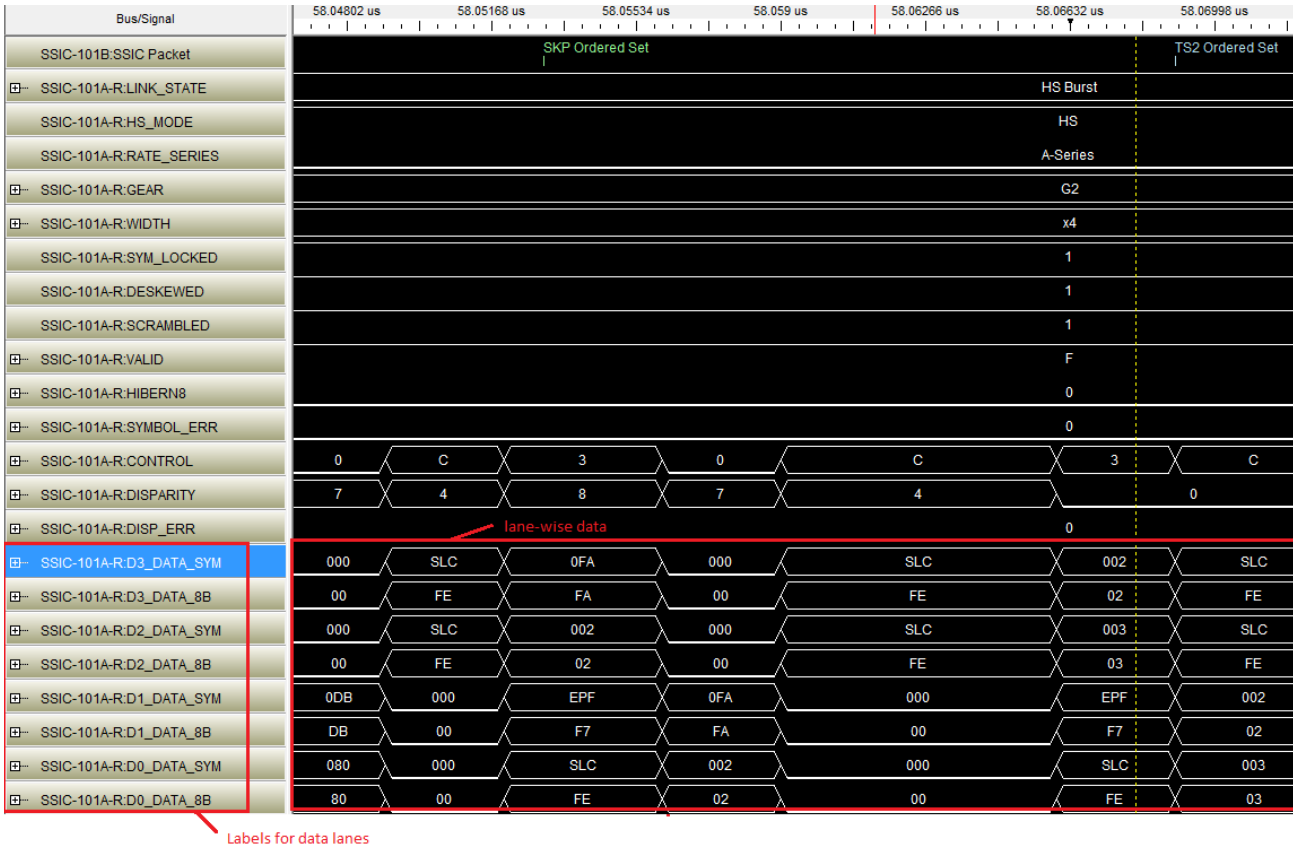
You can correlate the data displayed in the waveform with the relevant packet(s) in the Protocol Viewer. The following screen displays the correlated packet data for the signal data displayed in the above screen.



Viewing the Lane-wise Transmission of Raw Data

In the Waveform Viewer, you can view raw data organized as per the data lanes on which it was transmitted.

For instance, in the following screen, the raw data for SSIC packets is displayed across the four lanes D0 to D3 used for its transmission. In this example, each row of data in the highlighted section of the Waveform Viewer represents the data transmitted on the lane indicated through the row's label.



The lane-wise display of data can be skewed or deskewed based on the Raw Data Capture Mode that you selected in the Analyzer Setup tab prior to data acquisition.

NOTE The display of raw data for each lane in the Waveform Viewer depends on whether or not you enabled the **Raw data capture** feature in the **Analyzer Setup** tab prior to data acquisition.

Viewing Deskewed and Descrambled Data

Deskewing the Acquired Data

By default, the U4431A modulede skews the acquired data because the default selection for the **Raw Data Capture Mode** is **8b Desrambled and Deskewed Data** in case of SSIC and MPCIe and **8b Deskewed Data** in case of UniPRo/UFS/CSI-3.

To enable deskew for the data to be acquired

1 Access the **Analyzer Setup** tab of the Setup dialog box.

2 From the **Raw Data Capture Mode** listbox,

- select the **8b Deskewed Data** option for UniPro/UFS/CSI-3.
- select the **8b Desrambled and Deskewed Data** option for SSIC and MPCIe.

To view deskewed data

1 Display the Waveform Viewer window.

2 Look for the **DESKEWED** row in the Waveform Viewer. Notice that, in the following example, the DESKEWED value is displayed as 1 indicating that the deskewing is enabled. The deskewed data at the start of a burst across the three lanes (D0, D1, and D2) is also highlighted in this example.

Descrambling the Scrambled Data

Descrambling is applicable only for SSIC and MPCIe setups. By default, the U4431A module descrambles the scrambled data on the link because by default, the **Data on link is scrambled** checkbox is selected in the **Analyzer Setup** tab.

To enable descrambling for the scrambled data

1 Access the **Analyzer Setup** tab of the **Setup** dialog box.

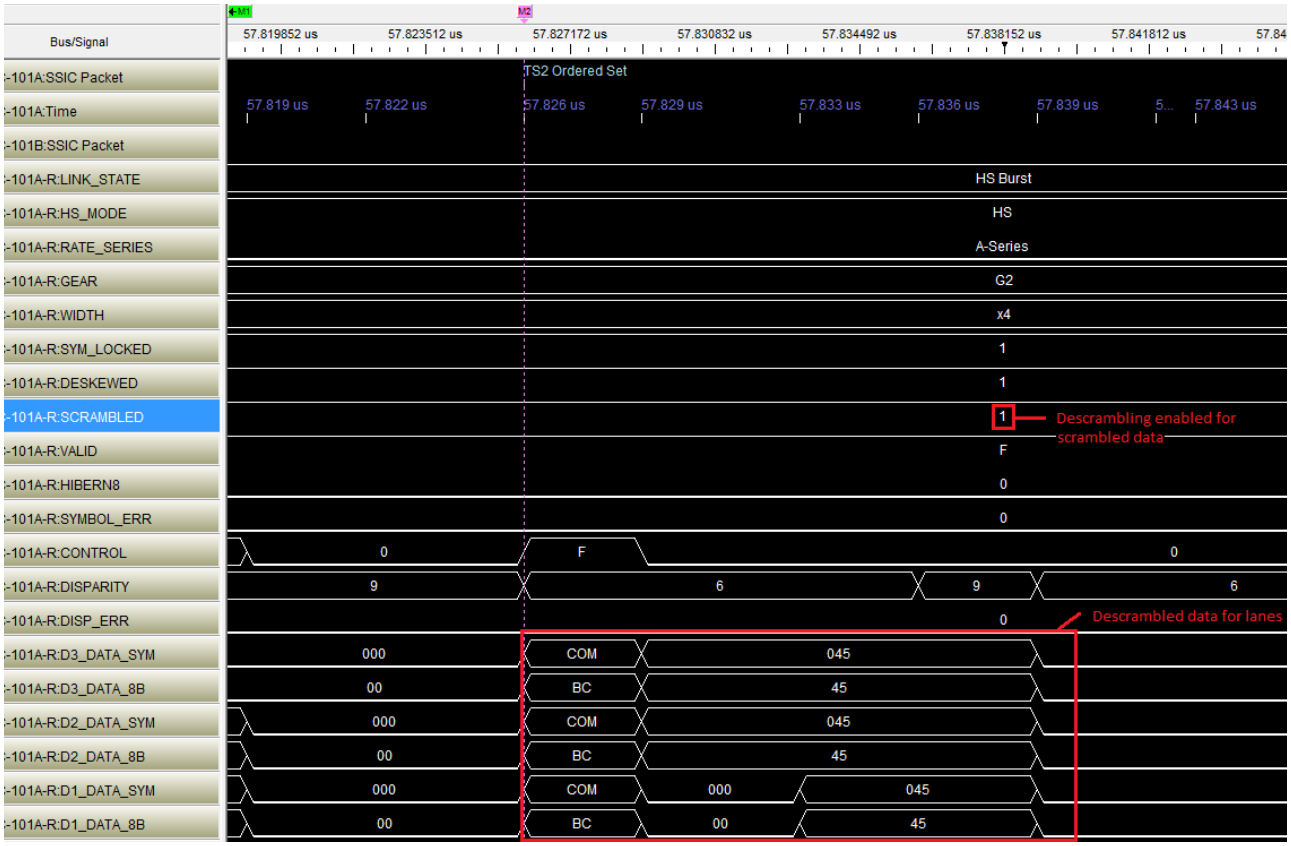
2 Ensure that the following two options are selected:

- the **Data on link is scrambled** checkbox is selected
- the **8b Desrambled and Deskewed Data** option or the **8b Descrambled Data** option is selected from the **Raw Data Capture Mode** listbox.

To view descrambled data

1 Display Waveform Viewer.

2 Look for the **SCRAMBLED** row in the Waveform Viewer. Notice that, in the following example, the SCRAMBLED value is displayed as 1 indicating that the data on the link is scrambled for which descrambling has been enabled. The descrambled data across lanes is also highlighted in red in this example.

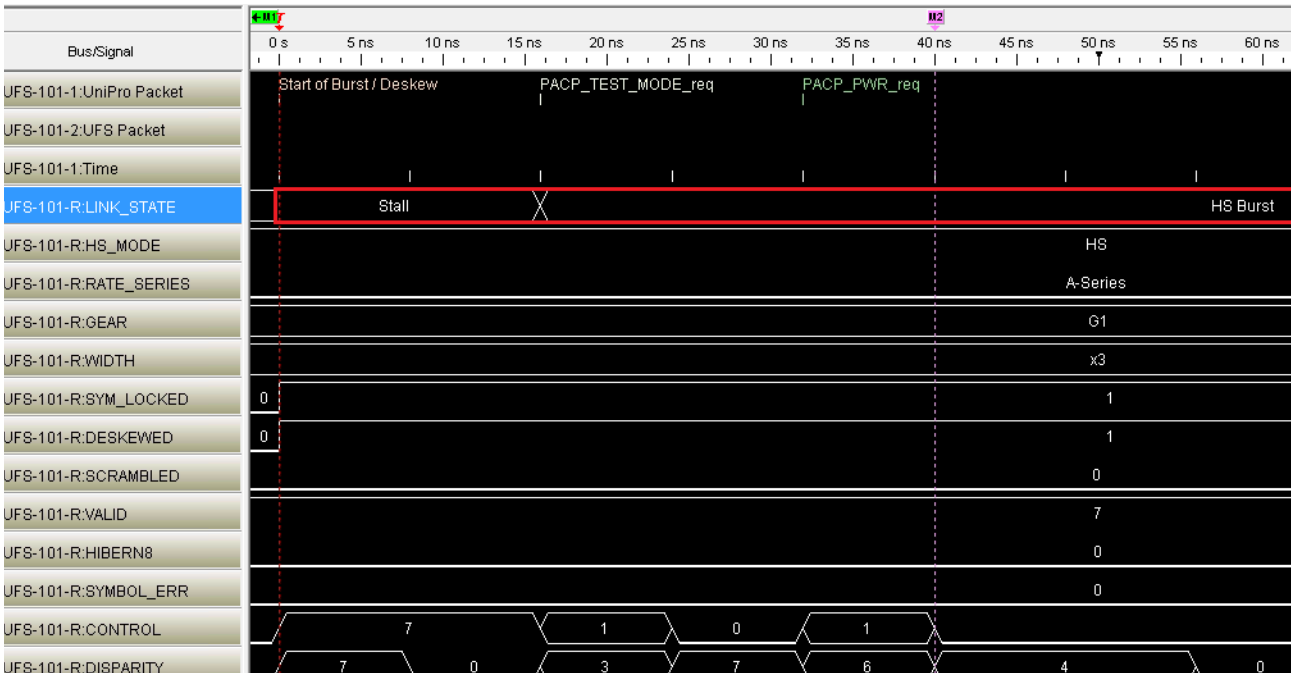


Tracking and Viewing Link States in Captured Data

In the Waveform Viewer, you can track and view the changes in the link's state of the acquired data. The **LINK_STATE** row in the Waveform Viewer displays the link's transitions from one state to another across the timeline of the acquired data.

In case of a bidirectional acquisition, the state of Transmit and Receive sublinks is displayed separately.

In the following screen, the row displaying the link state is highlighted.



Viewing Time Synchronized Protocol Data in Display Windows

For analysis or debugging, you may want to view the captured data in a display window along with its related protocol/signal data in another window such that the displays are time synchronized for ease of mapping and viewing related data. One such situation can be viewing UniPro protocol data in a Protocol Viewer and the related UFS protocol data in another Protocol Viewer. Or, viewing SSIC protocol data in a Protocol Viewer and its related signal data in a Waveform Viewer.

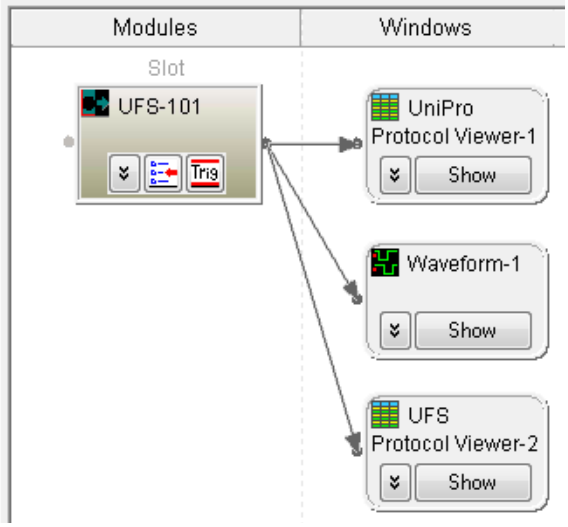
In such situations, you can use the Lockstep windows feature to ensure that when a Protocol Viewer window is scrolled, other lock stepped windows are automatically scrolled as well, such that the same time is centered in each lock stepped display window. This allows you to easily map and view correlated data in multiple display windows.

You can lockstep a Protocol Viewer window with display windows such as another Protocol Viewer, Waveform, Listing, or Compare.

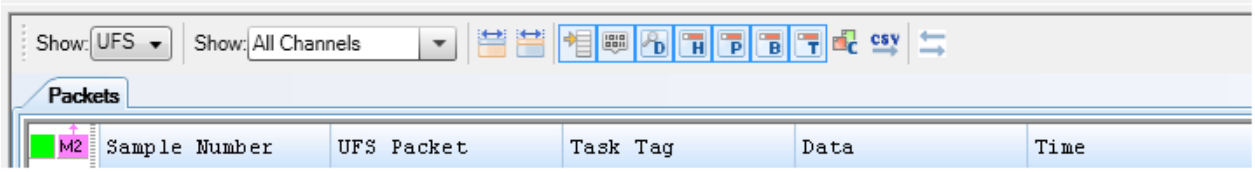
Some examples of using the lockstep feature with the data captured using U4431A are described below.

To lockstep UniPro protocol data with UFS protocol data

1 In the **Overview** window, ensure that you have added two instances of Protocol Viewers, one for the UniPro data and another for UFS data.

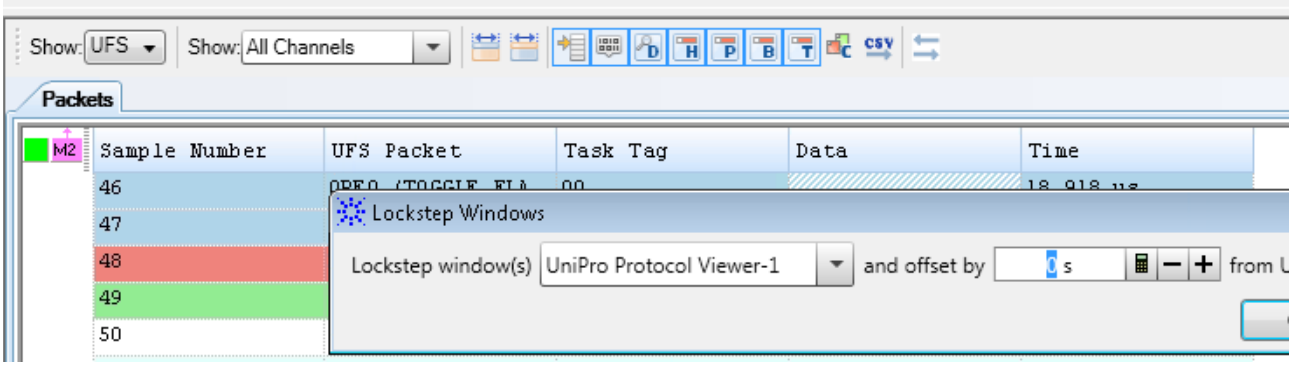


2 Ensure that each of the Protocol Viewer instance shows the data specific to the protocol for which it was created. Use the **Show** button in Protocol Viewer to show data specific to a protocol.



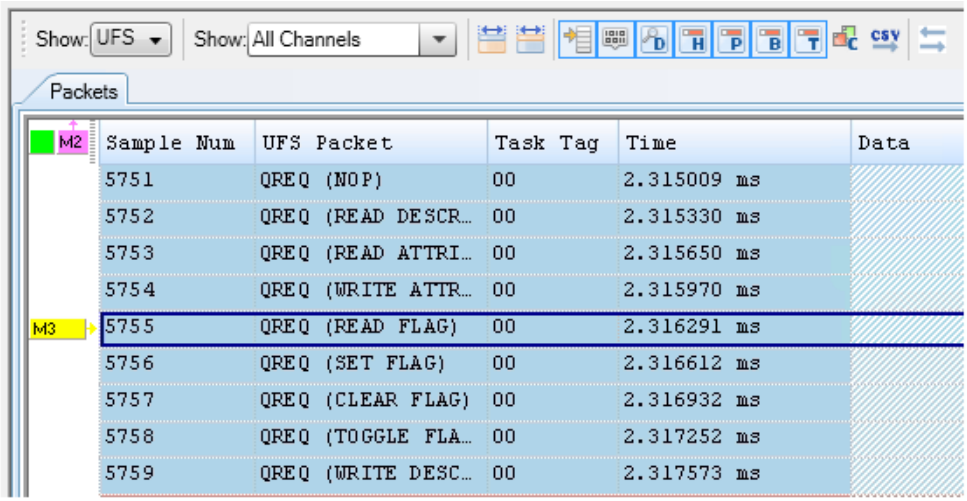
3 From the Protocol Viewer toolbar, click the **Lockstep Windows** toolbar button.

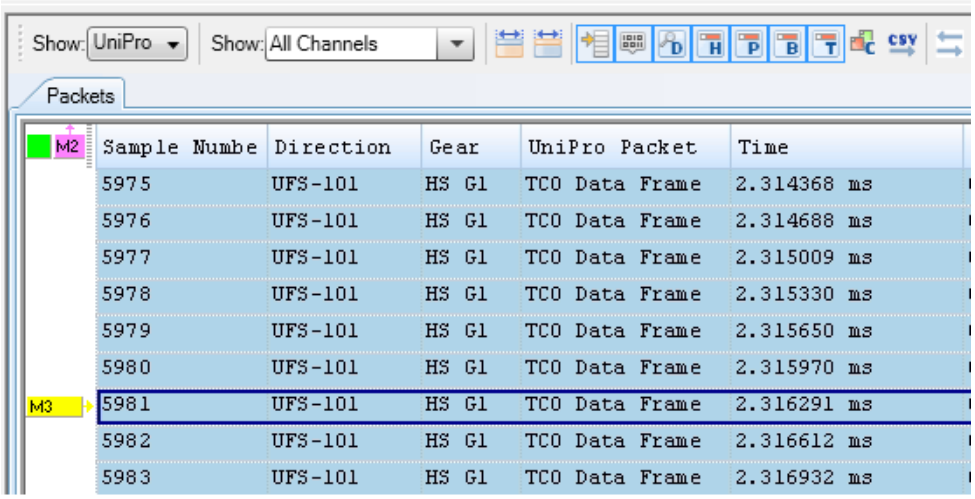
In the **Lockstep Windows** dialog, select the other instance of Protocol Viewer with which you want to synchronize scrolling and specify any time offset from this window. For instance, from the UFS Protocol Viewer instance, select the UniPro Protocol Viewer instance.



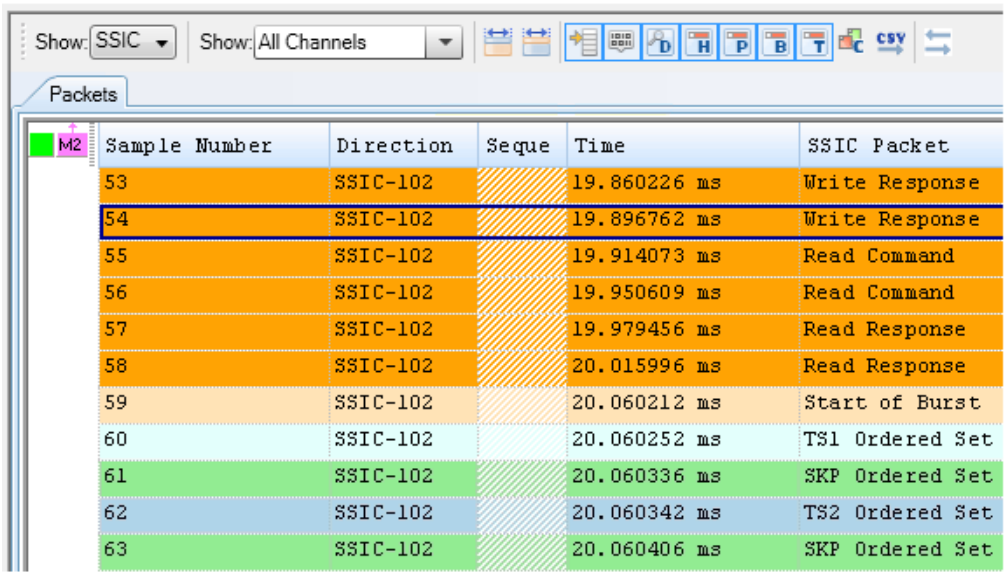
4 Click **OK** to close the **Lockstep Windows** dialog.

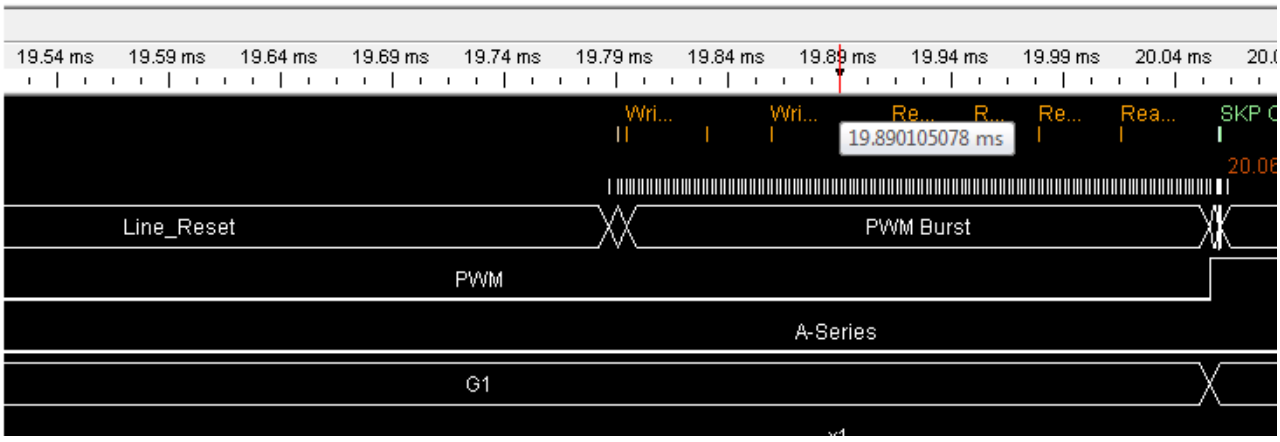
Once the windows are lockstepped, scrolling or moving to a specific marker in the lockstepped window automatically scrolls the other window to the appropriate location.





Similarly, you can lockstep a Protocol Viewer window with a Waveform Viewer window to ensure that when you scroll in Waveform Viewer window, protocol data in Protocol Viewer is scrolled. The following screen shows a synchronized display of SSIC data in Protocol Viewer and Waveform Viewer windows using Lockstep feature.





6 Viewing LTSSM States and State Transitions

The U4431A M-PHY analyzer lets you view LTSSM states and state transitions as detected in the data captured in a trace. Using this data, you can test or debug the DUT's LTSSM functions.

This chapter describes how you can view these LTSSM states.

NOTE The LTSSM feature is available only in SSIC and MPCIe protocols.

NOTE The number of SSIC states is less than MPCIe states.

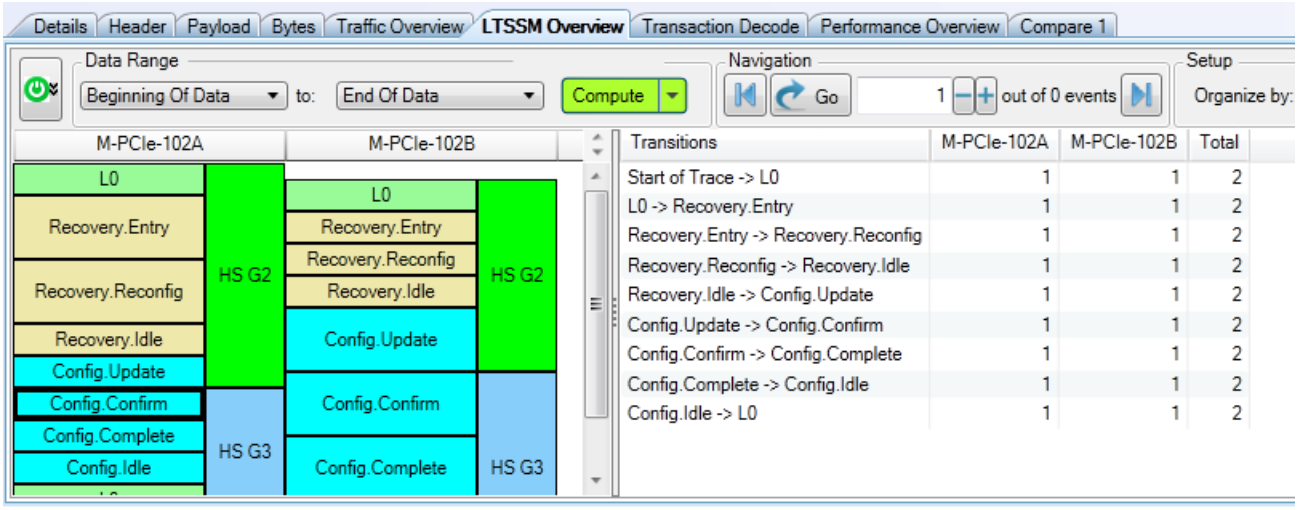
LTSSM Overview

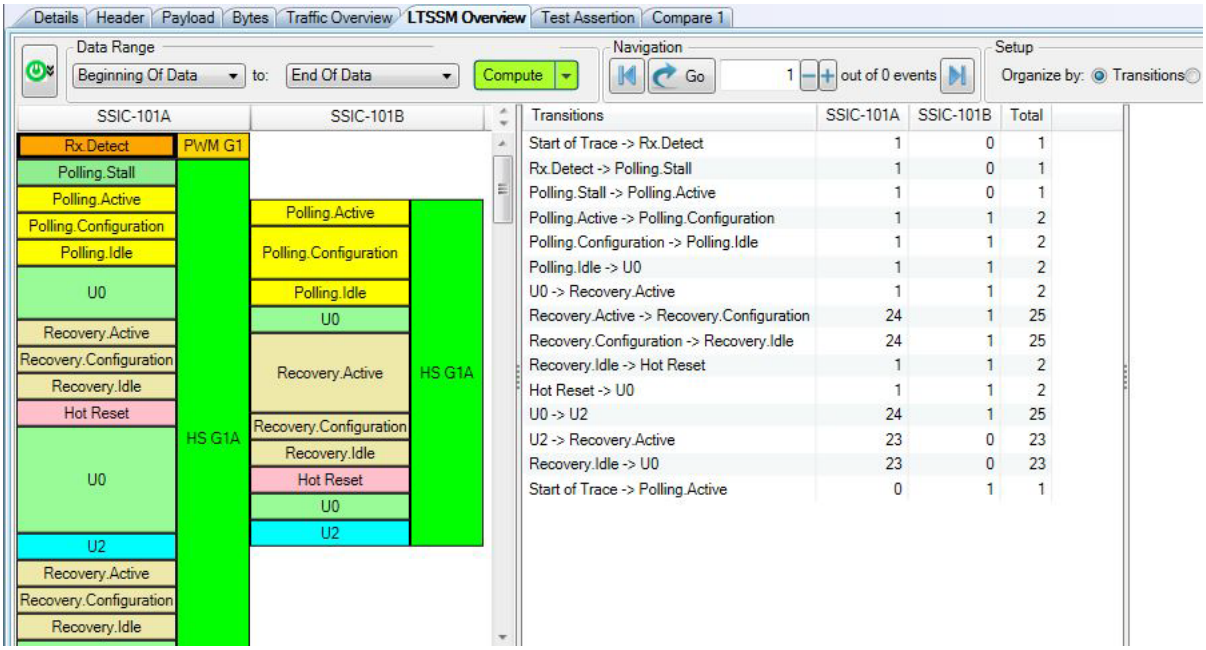
Link Training and Status State Machine (LTSSM) drives and controls the link initialization and training process for a device to enable the normal data exchange between the two devices over the link. LTSSM operates at the physical layer level and transits through various states and substates during link initialization, training, and management. During each of these states, appropriate physical layer packets (training sequences) are exchanged between the link partners to initialize, train, and manage the link.

To begin communication with a device, the link training process must complete successfully. This makes link training one of the most crucial process in testing and validating a DUT.

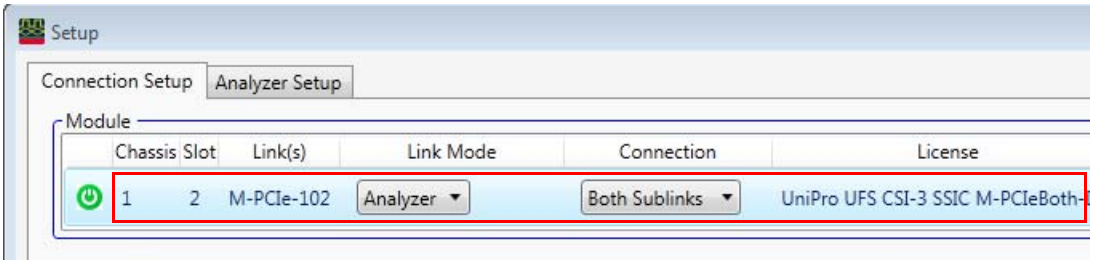
The **LTSSM Overview** pane in the Protocol Viewer window of the Logic Analyzer GUI helps you in verifying the link training process and finding out reasons for any failure in this process. This pane provides an overview of the link training process by displaying a sequential list of the LTSSM states and their transitions.

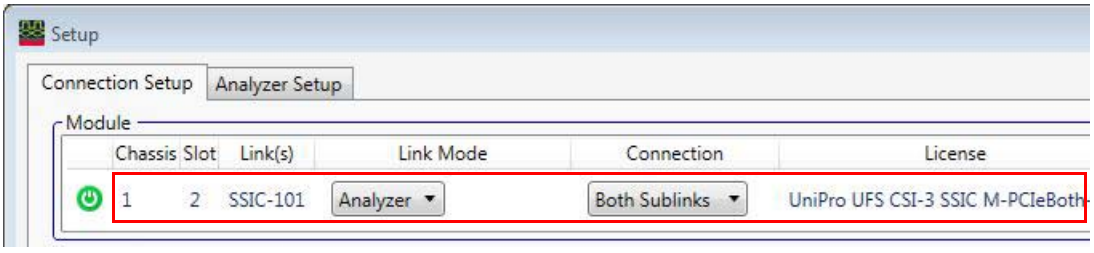
To display these states and transitions, the software analyzes a user-specified data range or the entire data captured in a trace and presents the list of transitions/states data, from that trace.

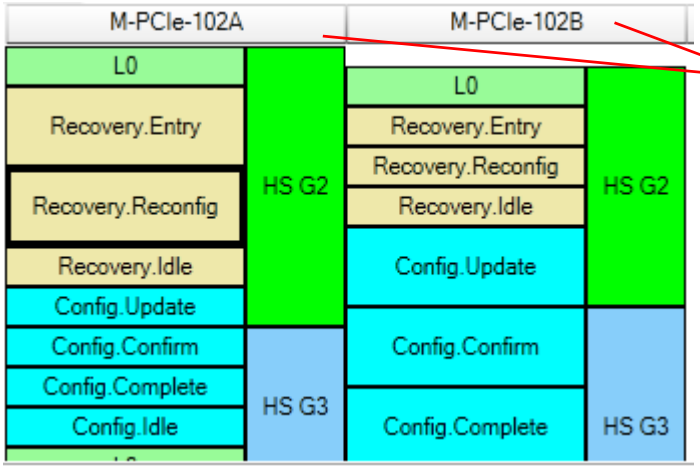


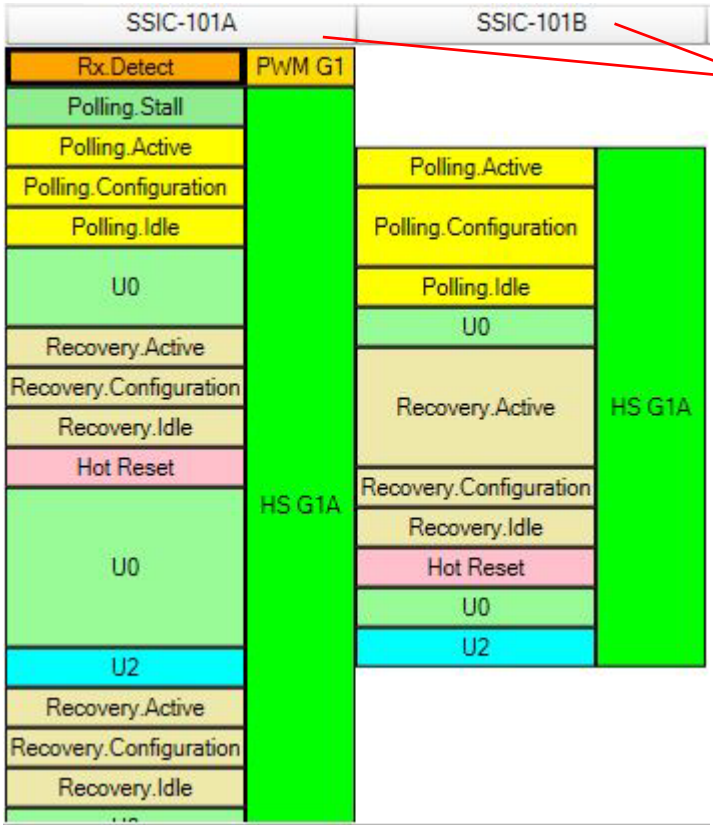


The LTSSM Overview pane can display LTSSM transition and state data, for one as well as both sub-links. However, this display depends on how you configured the connection of data capture (One Sublink, or Both Sublinks) in the Connection Setup tab of the Setup dialog box of the U4431A module. For instance, if you configured the data capture of both sublinks using U4431A, then LTSSM states are displayed for both links.









Using the LTSSM Overview pane, you can view the LTSSM transitions and states, during events such as:

- Link initialization and configuration to bring the link to an operational state.
- Link recovery or retraining in situations such as speed changes, power management, or recovering from a link error.
- Downgrading or upgrading the link speed in response to a link speed change request.

Prerequisites

Before you can view LTSSM transitions and states, you need to ensure that:

- You have the appropriate node/server software license available and installed for the LTSSM Overview feature.
- You have set up the U4431A analyzer module and captured the data for the required link(s).
- You have configured the LTSSM setup to get the data display in the LTSSM overview pane according to your requirements. (Described in the next section)

Configuring and Computing LTSSM States

To view LTSSM data in the LTSSM Overview pane, you need to configure the pane settings and initiate computation of the LTSSM data display based on these settings.

1 Access the captured data along with the U4431A setup details in the Logic Analyzer GUI.

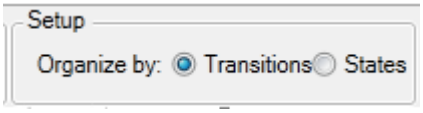
2 Click the **Protocol Viewer** node connected to the U4431A module in the **Overview** window of the Logic Analyzer GUI.

3 Click the **LTSSM Overview** tab in the lower pane of the Protocol Viewer window.

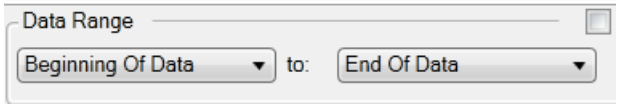
4 Configure the LTSSM setup to get a display of LTSSM States/Transition as per your requirements.

a To configure how the computed LTSSM data should be organized in the LTSSM Overview pane, select the appropriate **Organize By** mode - By Transitions, or By States from the **Setup** groupbox.

- **Transitions** displays the LTSSM states transitions in the results for navigating through the occurrences of the state transition events in the analyzed data.
- **States** displays the LTSSM states in the results for navigating through the occurrences of the state entry events in the analyzed data.



b In the **Data Range** groupbox, specify the start and end range for the captured data in the trace for which you want to display the LTSSM data. Only the specified range of data is analyzed to detect the LTSSM states. The default selections in the Data Range group box ensure that the LTSSM data is displayed for the entire trace. However, you can set markers in the packets listing in the upper pane and then specify the data range using these markers so that LTSSM data is displayed only for that specific range of packets.



5 Click the **Compute** button displayed with the **Data Range** fields. There are following two options available as a drop-down menu with this Compute button.

- **Compute All** allows you to compute traffic overview statistics, decoded transactions, offline performance summary, and LTSSM data (in their respective tabs) for the captured packets by a single click of this button.
- **Compute This** allows you to compute only LTSSM data from the captured packets. When you click **Compute**, then also only LTSSM data is computed.

NOTE

If you want to compute LTSSM data in background while the U4431 module is capturing data, then select the **Compute All on Run** check box in the Protocol Viewer and click the **Run** toolbar button. This allows you to perform a compute for all the relevant tabs in Protocol Viewer automatically after the data capture is complete. It thereby helps you perform a faster compute than performing a compute for each tab individually after the data is captured.

The LTSSM data is computed and displayed for the specified data range and configured link direction. Refer to the next sections to view how the LTSSM states and transitions data is displayed.

NOTE

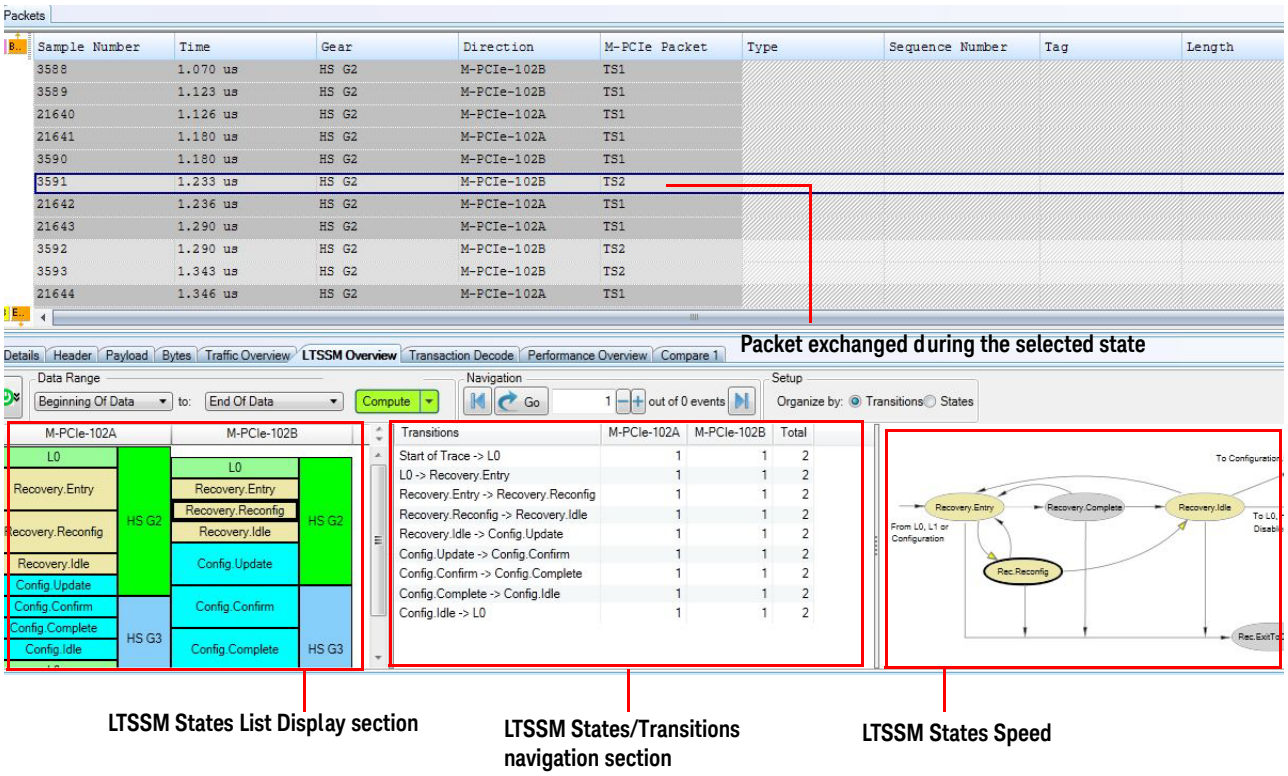
You need to recompute the LTSSM states display results if you want to change:

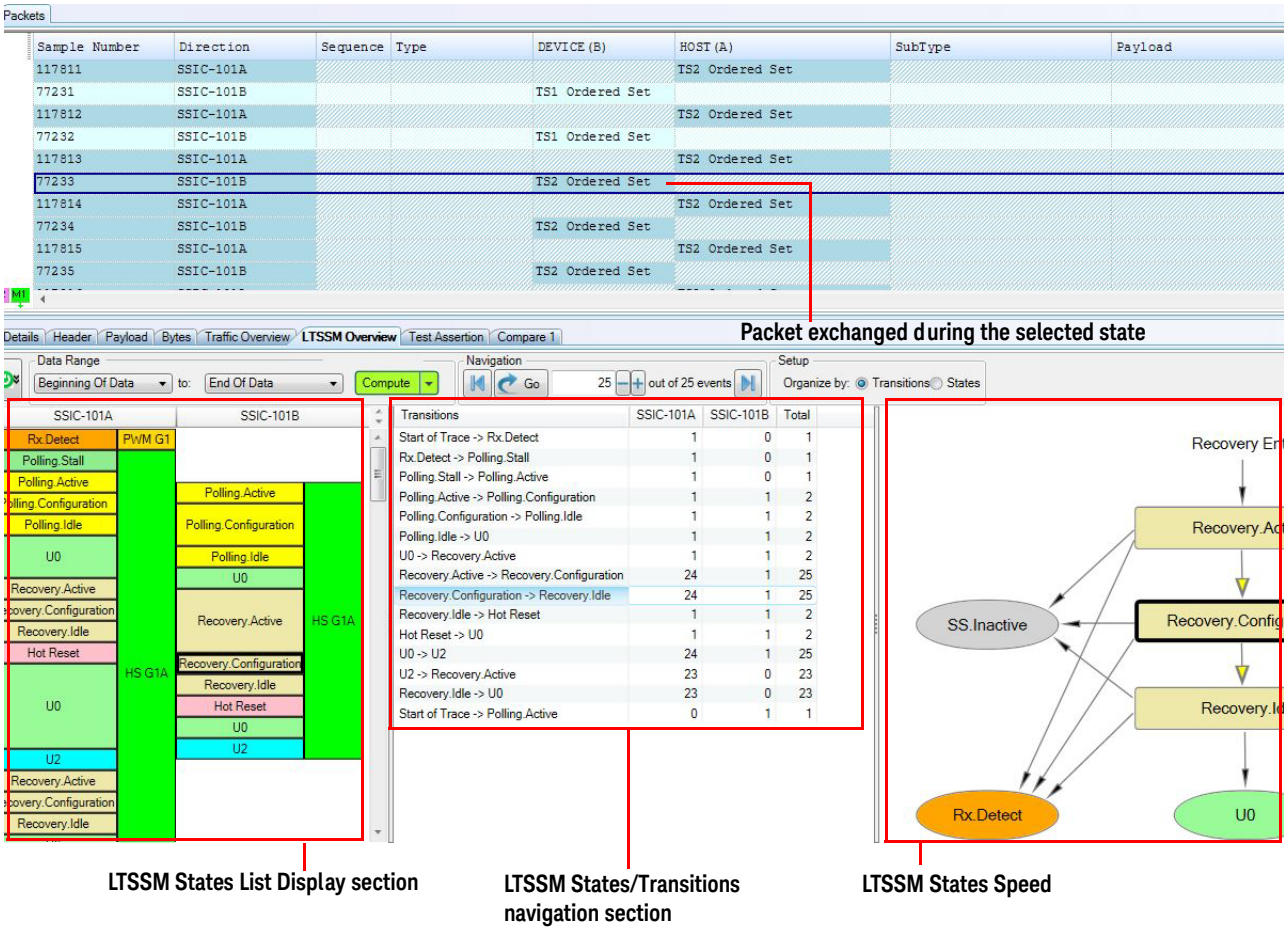
- the data range for which results are to be displayed.
- the LTSSM navigation mode for the navigation pane.
- the link direction for which results are to be displayed.

You can hide or display the LTSSM Overview pane using the  toolbar button in the upper pane of the Protocol Viewer window.

Viewing LTSSM States/Transitions Data

The screen below displays the results of a compute operation for LTSSM states followed by a description of these results.





The LTSSM states results are described in the following sections, as highlighted in the pane in the above screen.

LTSSM State Results	Description
LTSSM States List Display section	This section displays a list of the LTSSM states in the sequence in which these occurred in the trace or the specified part of the trace. The states are grouped and listed for a link direction. In the above screen, LTSSM states are displayed for both sublinks When you click a state in the list, the packet that is exchanged as the first packet for that state occurrence is highlighted in the upper pane of the Protocol Viewer. This provides you a quick start point for viewing and navigating through the packets exchanged during a particular state. Moving the mouse pointer to a state presents a tool tip with useful information about the state such as the time tag for the state and the packet exchanged at the state change.
LTSSM States/Transitions Navigation section	This section displays a list of the applicable LTSSM states or state transition names. The display of states or transitions in this section depends on whether you selected **Transitions** or **States** organization mode in the **Setup** groupbox. In the above screen, the display and organization in the navigation section is as per the *Transitions* mode. For each of these states/transitions, the section displays the number of events representing the number of occurrences of these states or transitions in the analyzed data. Using this section, you can easily navigate through these occurrences of LTSSM states or transitions and the packets exchanged during these occurrences.
LTSSM States Speed section	For each state in LTSSM States List Display section, the applicable link speed during the state is displayed. Clicking a speed in this list highlights the packet representing the transition to that speed in the upper pane of the Protocol Viewer. Clicking the gear on LTSSM States List Display section highlights states speed in blue text.

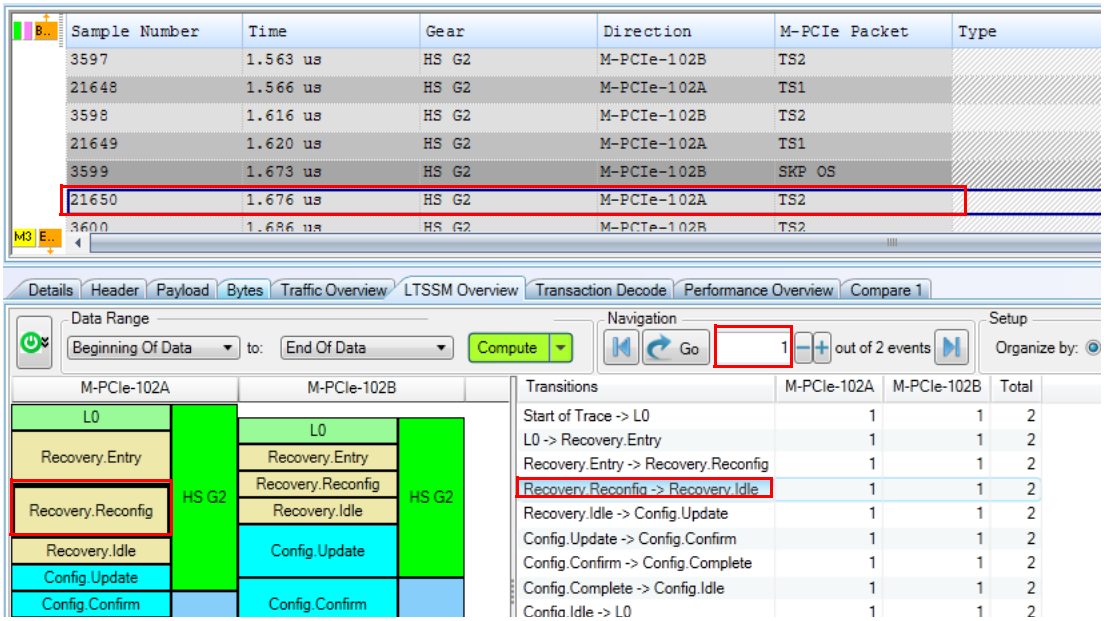
Navigating Through the LTSSM Transitions/States occurrences

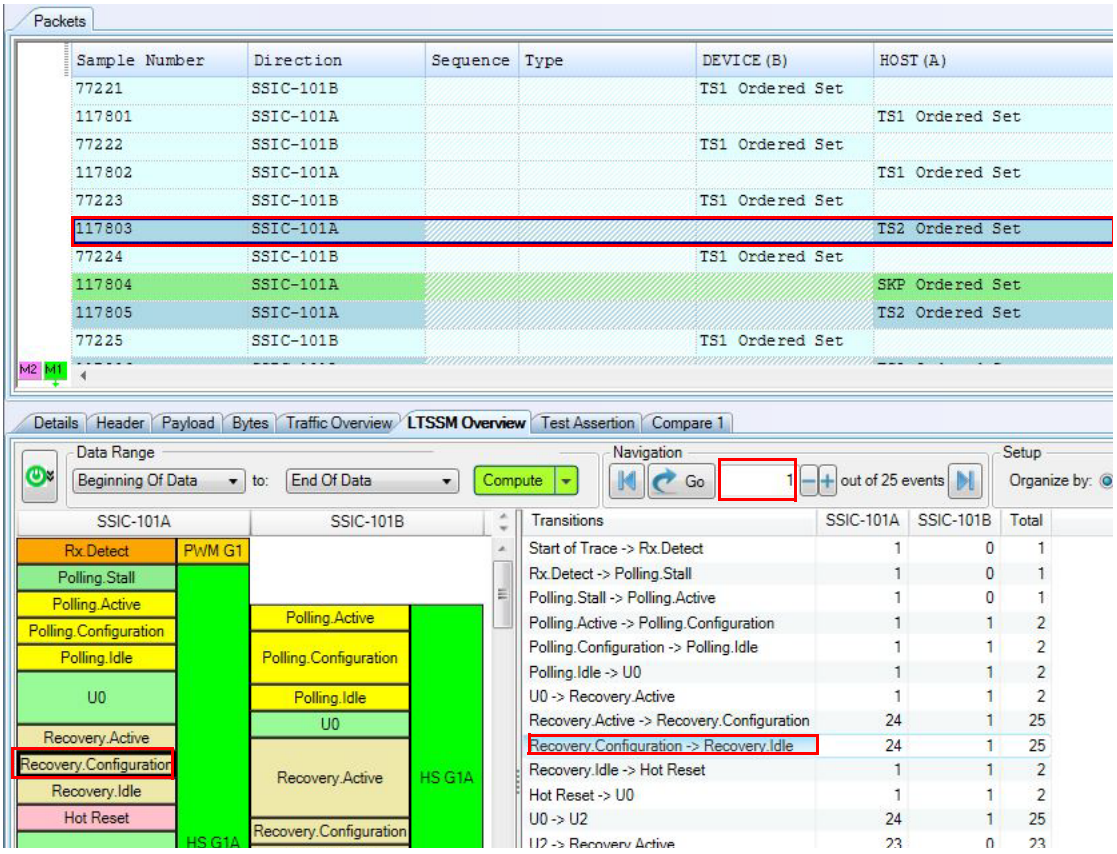
You can use the Navigation groupbox in the LTSSM Overview tab to navigate through LTSSM transitions/states occurrences.

1 To navigate through the occurrences of a state for both the directions, select the particular transition/state occurrence in the navigation section. To navigate through the occurrences of a state for a specific direction, select the number of events displayed for that direction in front of the transition/state name in the navigation section.

2 Either type the number of occurrence to which you want to navigate in the Navigation groupbox and click **Go** or click the **+** or **-** buttons in the groupbox to sequentially move to next or previous occurrence of the transition/state.

The specified occurrence of the selected state/transition is highlighted in the state list display and the associated packet is highlighted in the upper pane of the Protocol Viewer window. For instance, in the following screen, the first occurrence of the transition from Recovery.Reconfig to Recovery.Idle is highlighted in the states list along with the TS2 packet representing the beginning of the first occurrence of Recovery.Idle state.





3 Click  toolbar button to navigate to the first packet of the first occurrence of the selected state/transition.

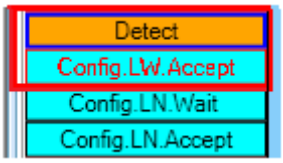
4 Click  toolbar button to navigate to the packet representing the transition to the last occurrence of the selected state/transition.

NOTE You can also double-click a state, or transition entry to quickly navigate to its associated packet in the upper pane.

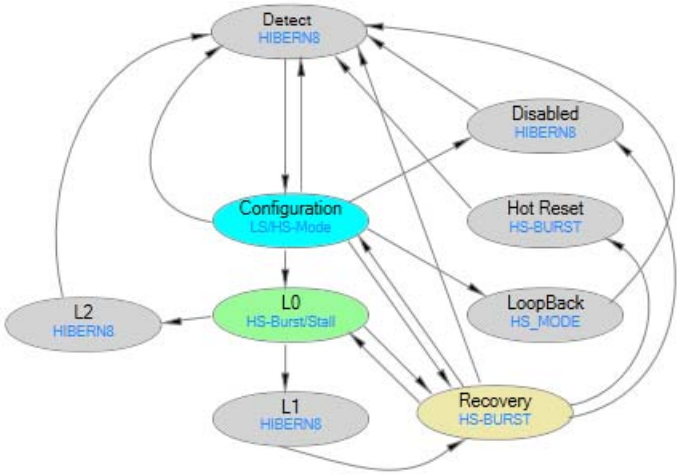
Interpreting LTSSM States and Transition Results

Following are some important points about interpreting the states and transitions displayed in the LTSSM Overview tab:

- **Errored LTSSM states** – If an LTSSM state is listed in red colored text, it indicates an erroneous state transition. Following is an example of an erroneous state transition from Detect to Config.LW.Accept. There should have been a transition to the Polling state after Detect.



- The following are the speeds in the speed column of the states listing.
  - HS
  - LS
  - PWM
- The pane displays LTSSM states as detected from the trace. If some events are not represented in the captured data in the trace, then these events will not be part of the LTSSM state transitions list in the LTSSM Overview pane.
- The LTSSM state diagram highlights which states have been entered in a given trace. This diagrammatic representation can help you quickly assess whether or not the states occurred as expected and required.



7 Computing and Viewing Decoded Transactions

The U4431A Analyzer module can decode and display transactions from the captured MPCIe and SSIC data. This chapter describes how you can compute and view these decoded transactions.

Transaction Decoding - Overview

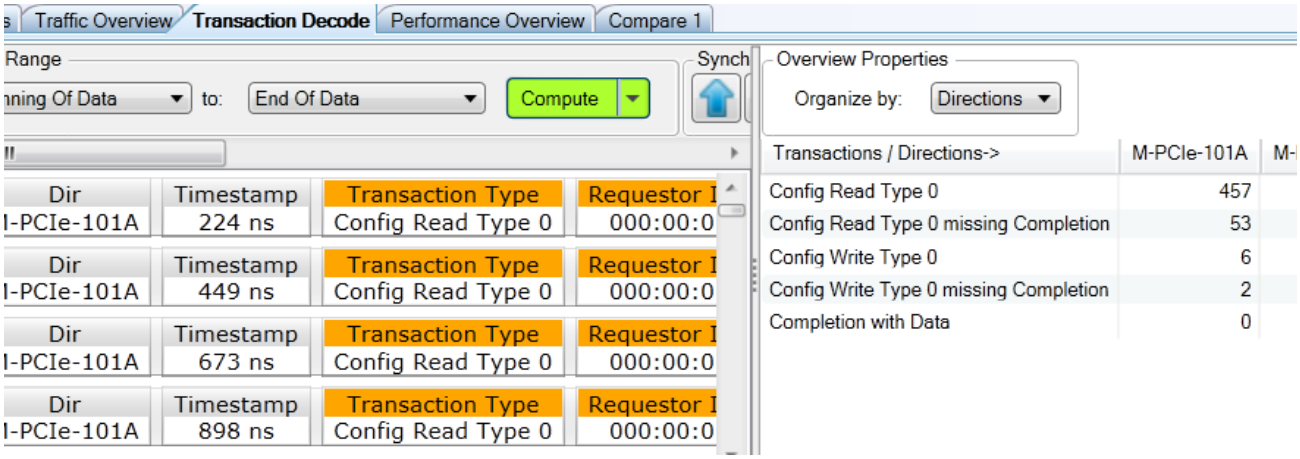
The **Transaction Decode** tab in the Protocol Viewer window allows you to compute and view transactions decoded from the captured MPCIe or SSIC traffic. The decoding and display of transactions is done as per the relevant storage protocol specifications to help you easily correlate the decoded data to the protocol specifications and evaluate DUT's compliance to these specifications.

Types of Protocols Supported

Decoding of MPCIe and SSIC transactions is supported.

Transaction Decode Tab

You use the **Transaction Decode** tab displayed in the lower pane of the Protocol Viewer to compute and view decoded transactions.



This tab consist of two panes. The left pane displays a list of transactions. The right pane displays the number of occurrences of these transactions.

Configuring and Computing Decoded Transactions

Before you Start

- You should have purchased and installed the U4431 *Transaction Decoder* license for computing decoded transactions.

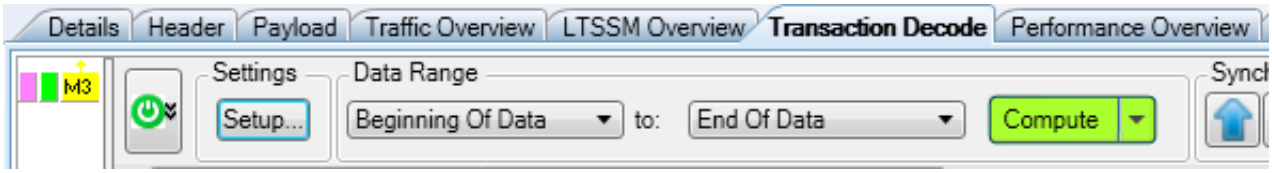
  On purchasing the license, you receive an entitlement certificate. Follow the instructions in this certificate to install the license.
- Ensure that the data in the required direction(s) is already captured and available in the Logic and Protocol Analyzer GUI for transaction decoding. You may save the captured data in a Logic Analyzer configuration (.ala) file and access this data offline for transaction decoding.
- It is recommended that you capture the device's complete boot process data so that the required device setup details are available for decoding transactions accurately.

Computing Transactions from the Captured Data

1 Click the **Transaction Decode** tab in the Protocol Viewer window.

2 In the **Data Range** groupbox, specify the start and end points of the captured data for which you want to compute decoded transactions. Only the specified range of data is analyzed to compute transactions. Following options are available for setting the data range.

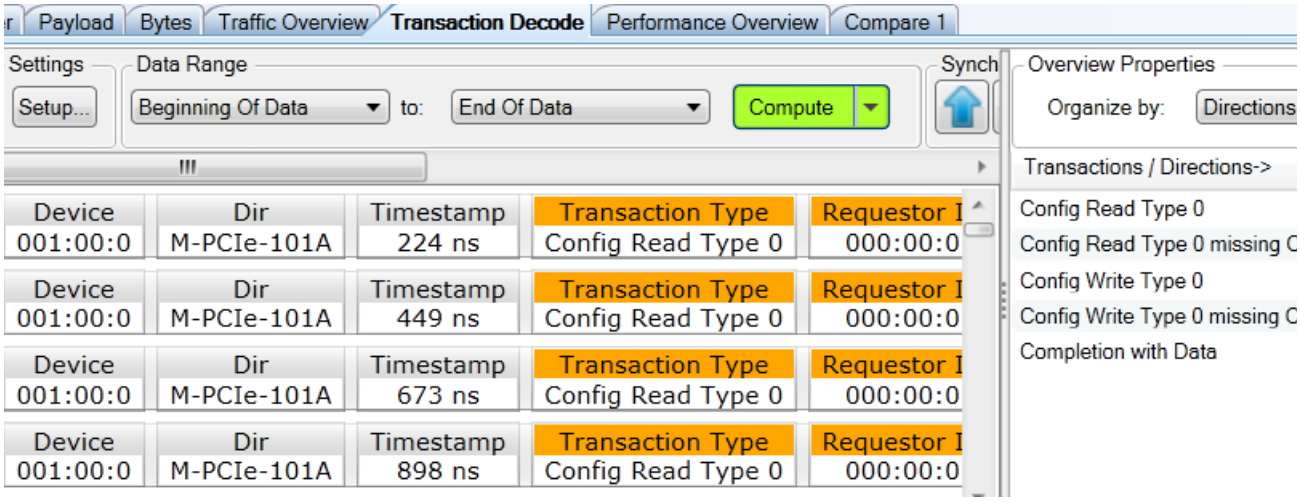
- **Beginning and End of data** - This data range selection ensures that transactions are computed for the entire trace.
- **Trigger** - Selecting Trigger in the data range ensures that transactions are computed from the point where the U4431 module's trigger condition was met.
- **Markers** - Selecting markers in the data range ensures that transactions are computed for the specific portion of MPCIe traffic defined by markers. Refer to "Defining Markers for Setting the Computation Range" on page 96 to know more.



3 Click the **Compute** button displayed with the Data Range fields. The **Compute This** and **Compute All** are the two options available with this button.

- **Compute All** allows you to compute traffic overview statistics (see page 64), decoded transactions, and offline performance summary (see page 156) for the captured packets by a single click of this button.
- **Compute This** allows you to compute only the decoded transactions from the captured packets. When you click **Compute**, then also only the decoded transactions are computed.

Transactions are computed and displayed for the specified data range.



NOTE

You need to recompute the decoded transaction results if you want to change:

- the data range for which transactions are to be displayed.
- the storage protocol for which transactions are to be displayed.
- the device setup such as base address, size, or the number of queues.

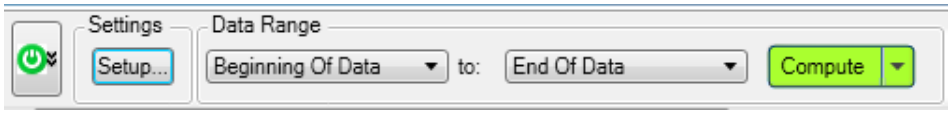
Defining Markers for Setting the Computation Range

If the captured traffic is too large and you want to view decoded transactions from a specific portion of this traffic, then you can limit the computation range by defining start and end markers in the traffic.

To define markers

1. From the upper pane of the Protocol Viewer, right-click the row in the captured traffic that should act as the starting point for transaction decode.
2. Select **Place Marker** from the displayed context menu and then select an existing marker or click **New Marker** to define a new marker at this point.

Once markers are defined, these are available for selection in the **Data Range** group box of the **Transaction Decode** tab.



Defining / Verifying the Device Setup

While computing decoded transactions, the Logic and Protocol Analyzer software automatically discovers the required device details such as device ID, type, and base address from the captured data. You can view this device related data in the **Transaction Decode Setup** dialog box.

NOTE

At times, the required device details are not available in the captured data and therefore cannot be autodiscovered from the captured data. In such situations, it becomes mandatory for you to specify these details. To avoid such a situation, you can ensure that device details are available by capturing the device's complete boot process.

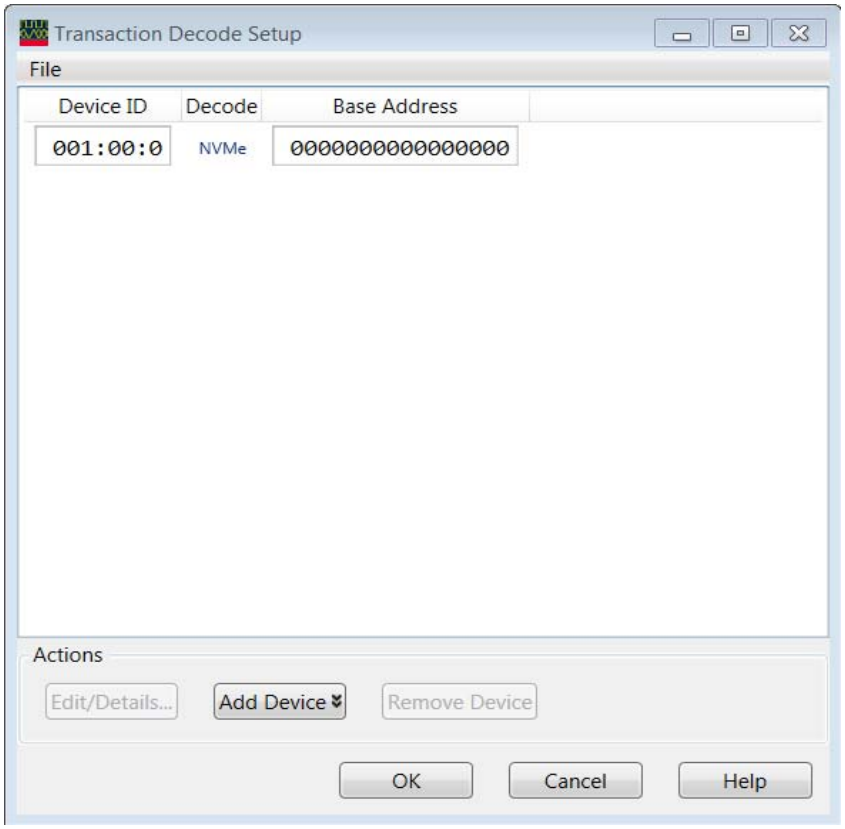
It is recommended that you verify and modify device details (if needed) for accurate transaction decode computation.

You use the **Transaction Decode Setup** dialog box to define/modify/verify the information about the device being tested. The types of information displayed in this dialog box differ based on the storage protocol decode applicable for the device.

To verify/modify device details

1 Click **Setup** in the **Transaction Decode** tab of the Protocol Viewer.

The **Transaction Decode Setup** dialog box is displayed with the device details autodiscovered from the captured data after you have computed the decoded transactions. In the following screen, the setup is displayed for MPCIe data.



2 Modify **Device ID** and **Base Address**, if needed.

3 You can also add a new device details by clicking **Add Device** and then selecting the appropriate device type applicable for the DUT.

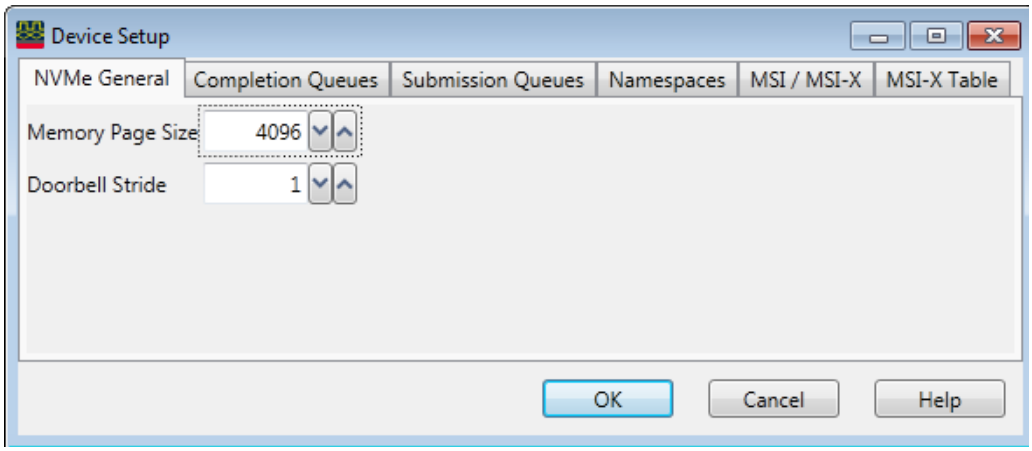
4 To delete the setup details of an existing device, select the row of the device and click **Remove Device**.

5 Double-click the device row to view/edit its details. Alternatively, select the device row and click **Edit Details...**

The **Device Setup** dialog box is displayed. The tabs and fields in this dialog box differ based on the storage protocol decode applicable for the device.

To view/edit details of an NVMe device

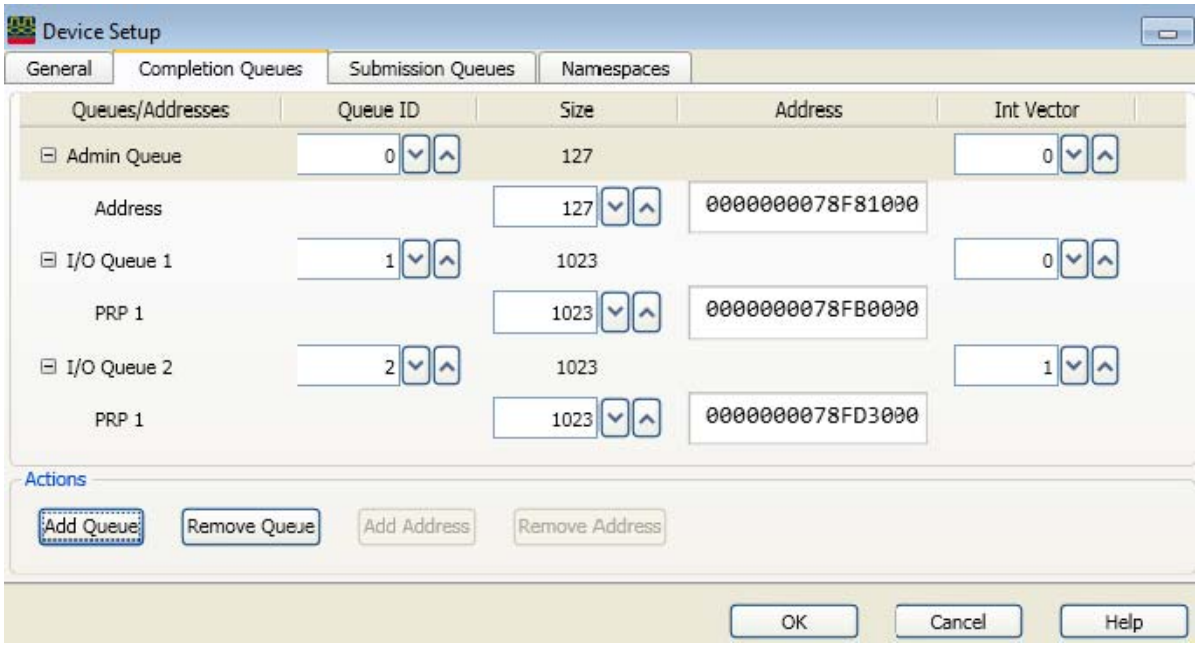
All device details such as its submission and completion queues and namespaces are autodiscovered while computing decoded transactions. If needed, you can add, remove, or edit these details of an NVMe device in the Device Setup dialog box.



1 The **Memory Page Size** field indicates the size of the physical memory page configured by the host software. The Memory Page Size value is a part of the controller configurations that the host can set and modify. This value is used to set the size of PRP entries.

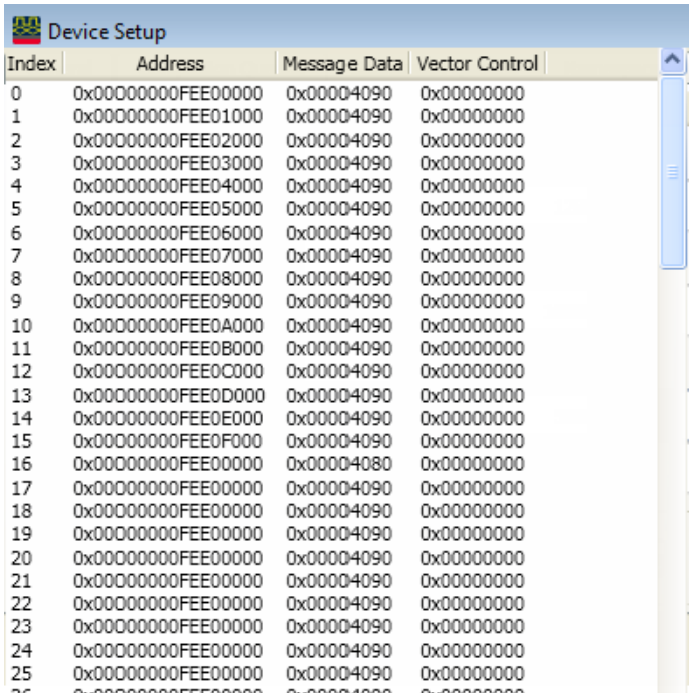
2 The **Doorbell Stride** field indicates the number of bytes to be used in memory space to separate doorbell registers. This value is a part of the controller capabilities.

3 Click the **Completion Queue** tab. The details of Admin and I/O completion queue(s) autodiscovered by the software from the captured data are displayed. If these are not autodiscovered, you can add a new queue by clicking **Add Queue** and specifying its ID, size, base address, and the unique MSI-X vector that the controller allocated to this queue to respond back.



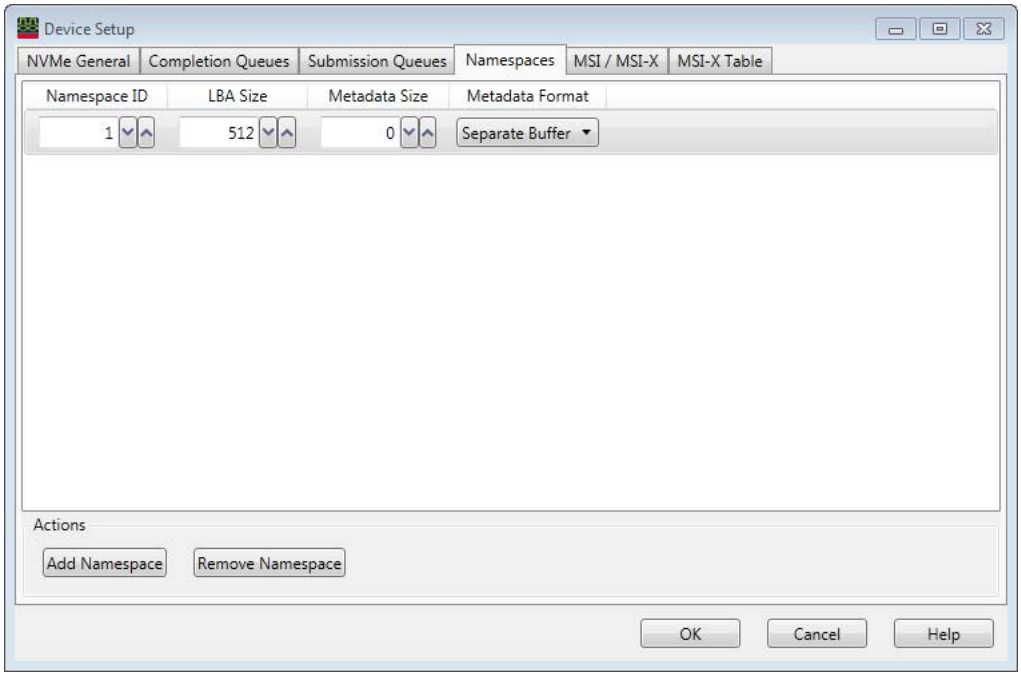
4 Click the **Submission Queue** tab. The details of Admin and I/O submission queue(s) autodiscovered by the software from the captured data are displayed. The completion queue associated with a submission queue is also displayed. A queue ID is used to display this association. If a queue's details are not autodiscovered, you can add a new queue by clicking **Add Queue** and specifying its ID, size, and base PRP address(es).

5 If a queue is physically non-contiguous, multiple PRP addresses are displayed for the queue representing multiple memory chunks. For a non-contiguous queue, you can add multiple PRP addresses by selecting a PRP address entry of the queue and clicking **Add Address**.



6 Click the **Namespaces** tab. The details of the controller's namespace(s) autodiscovered by the software from the captured data are displayed. If these are not autodiscovered, you can add a new namespace by clicking **Add Namespace** and specifying the following details.

- **LBA Size** - The Logical Block Address (LBA) data size (in bytes) that the namespace supports.
- **Metadata Size** - The number of metadata bytes provided per LBA.
- **Metadata Transfer Mode** - The metadata may be transferred either as part of the LBA by creating an extended LBA or as a separate contiguous buffer of data. When this field is set to **End of data LBA**, it indicates that the metadata is transferred at the end of the data LBA. When this field is set to **Separate Buffer**, it indicates that all the metadata for a command is transferred as a separate contiguous buffer of data.



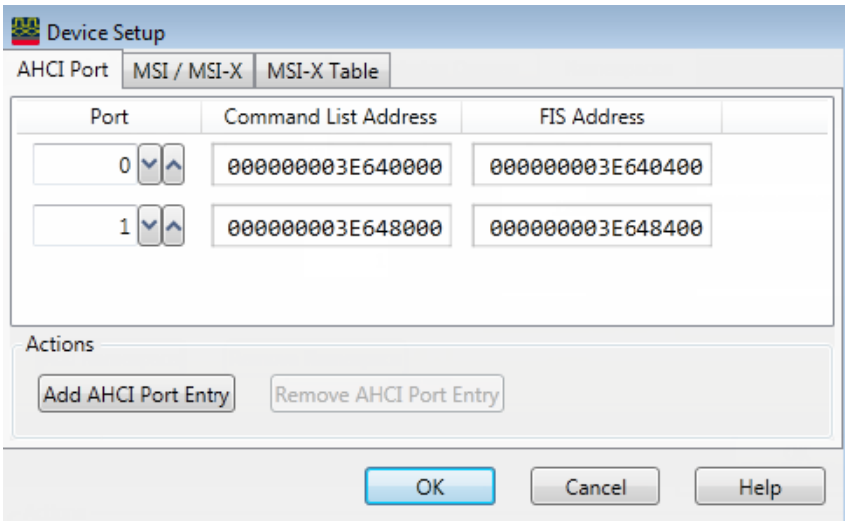
7 Click the **MSI / MSI-X** tab to view/modify the MSI / MSI-X capabilities that were discovered for the Name device.

8 Click the **MSI-X** tab to view/add/modify the MSI-X table entries that were discovered for the Name device.

9 Click **OK** to confirm and apply these settings.

To view/edit details of an AHCI device

An AHCI device's details such as the ports that it supports are autodiscovered while computing decoded transactions. If needed, you can add, remove, or edit these details of an AHCI device in the Device Setup dialog box.



1 In the **AHCI Port** tab, click the **Add AHCI Port Entry** button if you want to add an AHCI port that the device supports. This tab displays the autodiscovered ports of the AHCI device.

2 In the **Port** field, view/modify the port number.

3 In the **Command List Address** field, view/modify the base physical address of the Command List of the selected AHCI port. A command list refers to the list of commands that the HBA fetches from the command list base address to execute. The base address of the command list is as per the *PxFB* and *PxFBU* port registers.

4 In the **FIS Address** field, view/modify the base address of the FISes (Frame Information Structure) received for the selected AHCI port. An FIS refers to the frame of information exchanged between the host and device. The base address of the FIS is as per the *PxFB* and *PxFBU* port registers.

5 Click the **MSI / MSI-X** tab to view/modify the MSI / MSI-X capabilities that were discovered for the AHCI device.

6 Click the **MSI-X** tab to view/add/modify the MSI-X table entries that were discovered for the AHCI device.

7 Click **OK** to confirm and apply these settings.

To view/add details of an SSIC device

When computing decoded SSIC transactions, you:

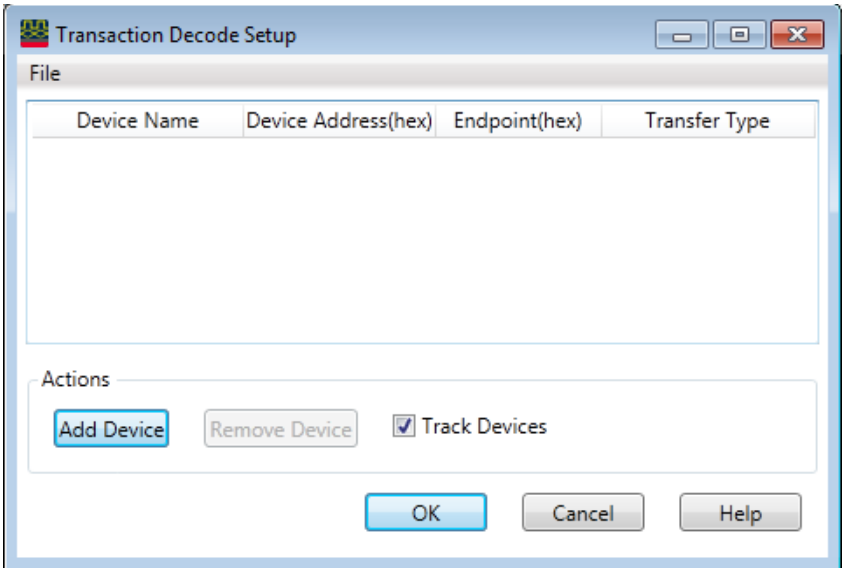
- either manually add details for the SSIC device(s) associated with the captured data.
- or configure the software to automatically discover the SSIC device details from the captured data.

In both these cases, you use the **Transaction Decode Setup** dialog box to add/autodiscover SSIC device details. The SSIC transactions are decoded as per the added / autodiscovered SSIC device details.

If the required device details are not available in the captured data, then the software will not be able to autodiscover the device details from the captured data. In such a situation, it becomes mandatory for you to specify these details for the accurate transaction decode.

1 Click **Setup** in the **Transaction Decode** tab of the Protocol Viewer.

The **Transaction Decode Setup** dialog box is displayed. In the following screen, the setup for SSIC devices is displayed before the transactions are computed.



2 To manually add a device:

a Click **Add Device**.

b Select a **Device Name**. For SSIC devices, the software supports **CDC** device class and **ACM**, **NCM**, and **MBIM** subclasses.

c Specify the address and endpoint used by the device.

d From the **Transfer Type** listbox, select the transfer/endpoint type based on the endpoint usage. The software supports the following transfer/endpoint types:

- **Control** - For initial configuration of the device and command and status operations
- **Interrupt In or Interrupt Out** - For event notifications
- **Isochronous** - For raw or formatted data communication with guarantee of bandwidth or minimum latency
- **Bulk In and Bulk Out** - For large bursty data with error-free delivery, but with no guarantee of bandwidth.

e Click **OK** and then compute the transactions. The added device details will be used for such a compute operation.

Or

3 To configure the software to autodiscover SSIC device details from the captured data:

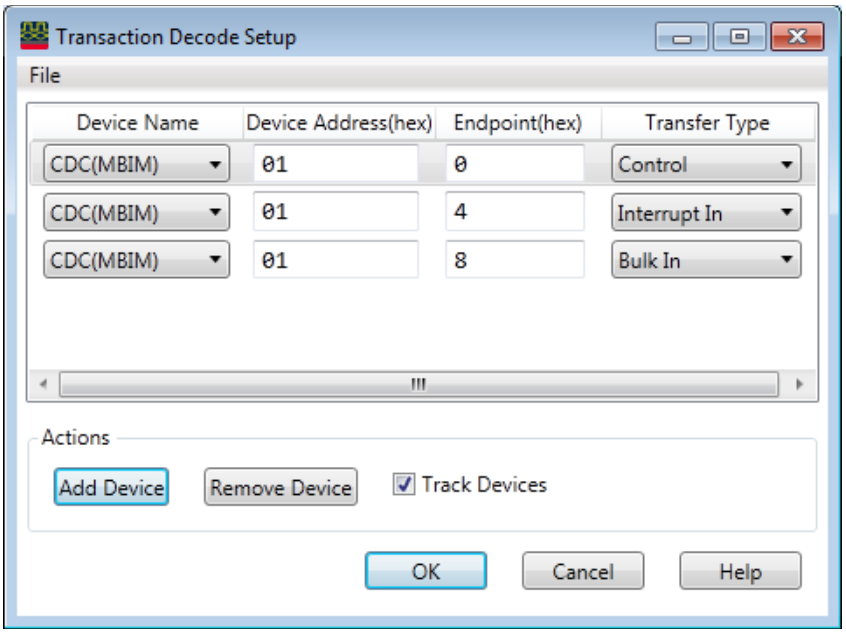
a Select the **Track Devices** checkbox in the Transaction Decode Setup dialog box.

b Click **OK** to close this dialog box.

c Click **Compute** in the **Transaction Decode** tab. When you click Compute, the software tracks the SSIC devices from the trace, populates these device details in the Transaction Decode Setup and computes the decoded transactions using these details.

d Open the **Transaction Decode Setup** dialog box again to view the autodiscovered device details after the computer operation.

The following screen displays three SSIC endpoints autodiscovered from the captured data.



To delete the setup details of an existing device, select the row of the device and click **Remove Device**.

See Also

"Viewing SSIC Transactions" on page 134

Saving the Device Setup Details

You can save the device, its queues, and namespace details (specified using the Transaction Decode Setup dialog box) in a Transaction decode (.tdprop) file. Once saved, you can open this file later in the Transaction Decode Setup dialog box to quickly access and set up the device details for a transaction decode computation.

To save device details

1. Click **Setup** in the **Transaction Decode** tab of the Protocol Viewer.
   The **Transaction Decode Setup** dialog box is displayed. Modify the device details as needed.
2. Click **File** > **Save As...**
3. Specify a name for the Transaction Decode setup file and click **Save**.

To open a previously saved setup file for transaction computation

1. Click **Setup** in the **Transaction Decode** tab.
   The **Transaction Decode Setup** dialog box is displayed.
2. Click **File** > **Open**.
3. In the **Open** dialog box, select the Transaction decode (.tdprop) file that contains the device setup details.
4. Click **Open**.

You can edit a saved .tdprop file by opening it in an XML Editor or the Transaction Decode Setup dialog box.

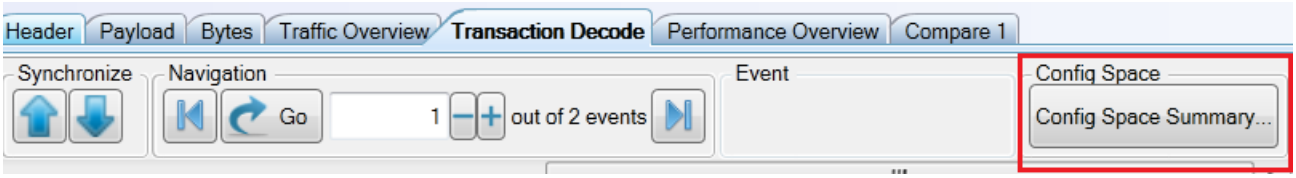
Viewing the Configuration Space of a DUT

While computing MPCIe decoded transactions, the Logic and Protocol Analyzer software automatically discovers and reads details from various MPCIe configuration registers in the DUT's configuration space. It displays register values for DUT's capabilities such as PCI, PCIe, MPCIe, Power Management, MSI, Virtual Channel, and Secondary PCIe Extended capabilities.

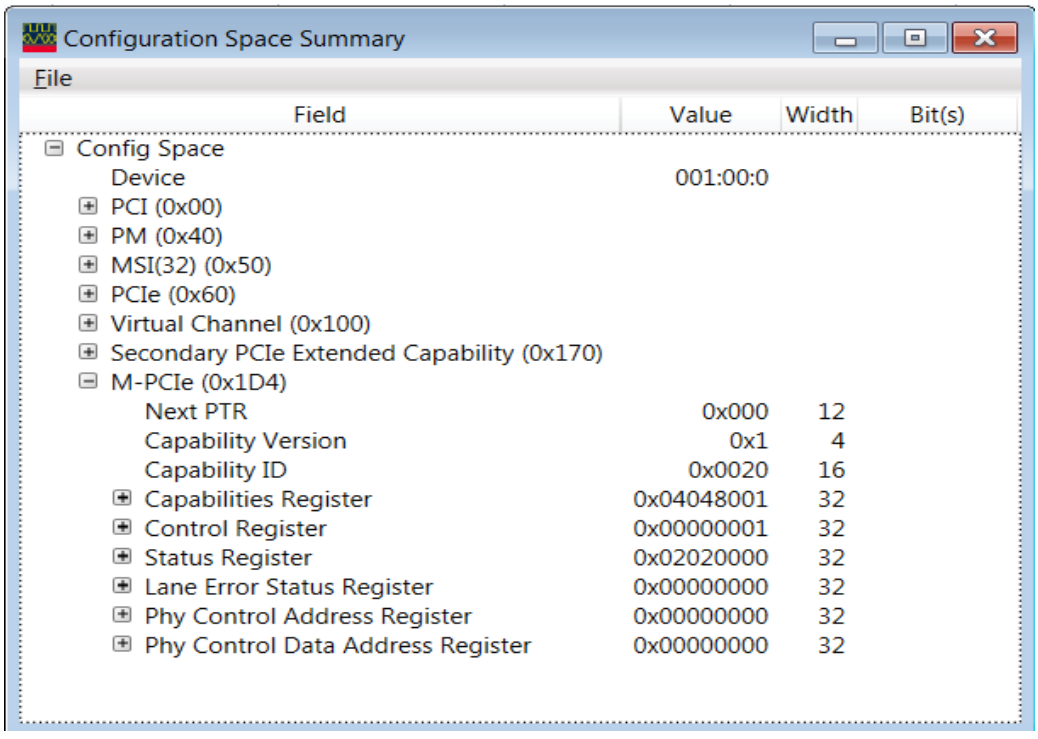
To view the configuration space of a DUT

1 Compute decoded transactions in the **Transaction Decode** tab.

2 Click the **Config Space Summary** button displayed in the top panel of the Transaction Decode tab.



The configuration space summary of the DUT is displayed.



Viewing the Attribute Summary of a DUT

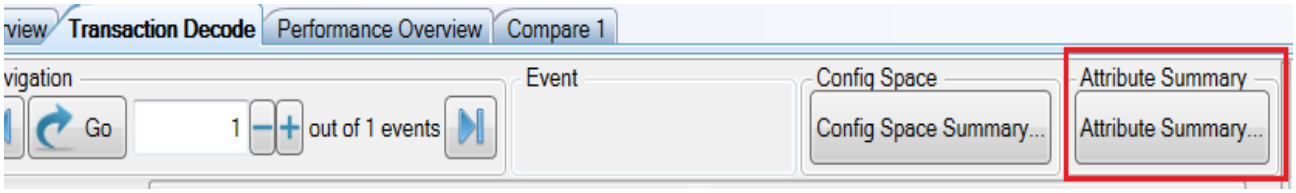
While computing decoded transactions, if the Logic and Protocol Analyzer software detects that the captured data includes PWM G1 data when RRAP packets are transmitted, it displays a button named Attribute Summary button in the Transaction Decode tab. You can use this button to display details related the attributes discovered from the decode of RRAP packets found in the captured data.

NOTE This button is not displayed if the captured data does not include RRAP packets.

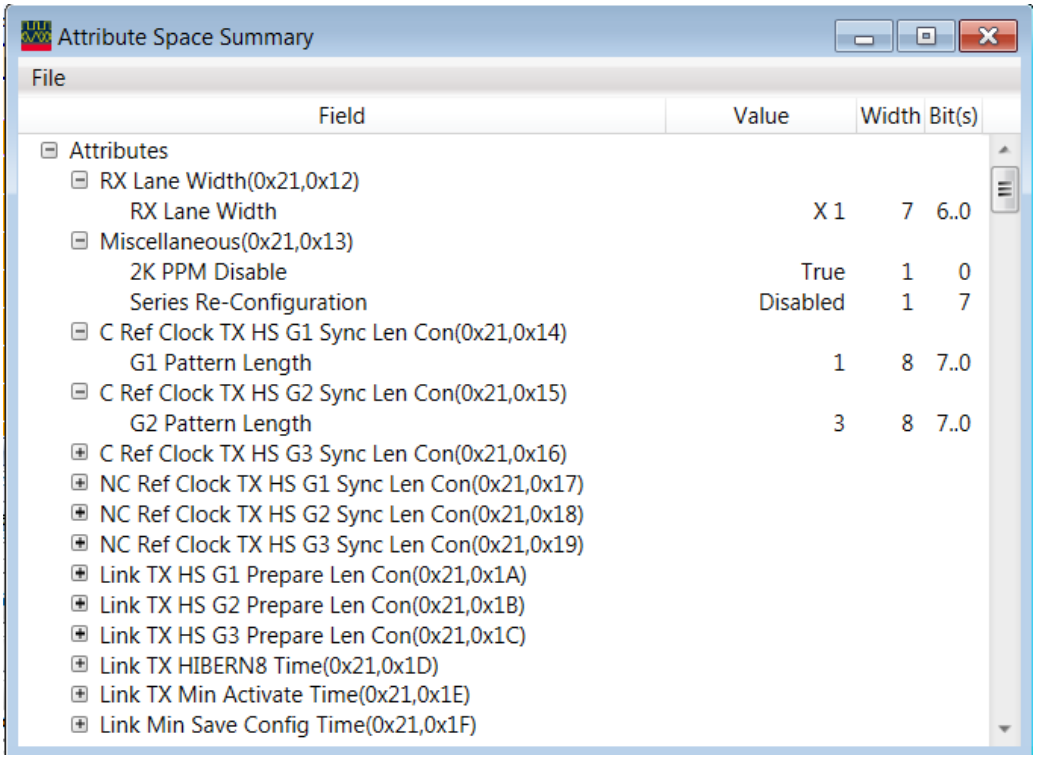
To view the attribute space summary of a DUT

1 Compute decoded transactions in the **Transaction Decode** tab.

2 Click the **Attribute Summary** button displayed in the top panel of the Transaction Decode tab.



The attribute space summary of the DUT is displayed.



Configuring Timestamps Displayed in the Computed Transactions

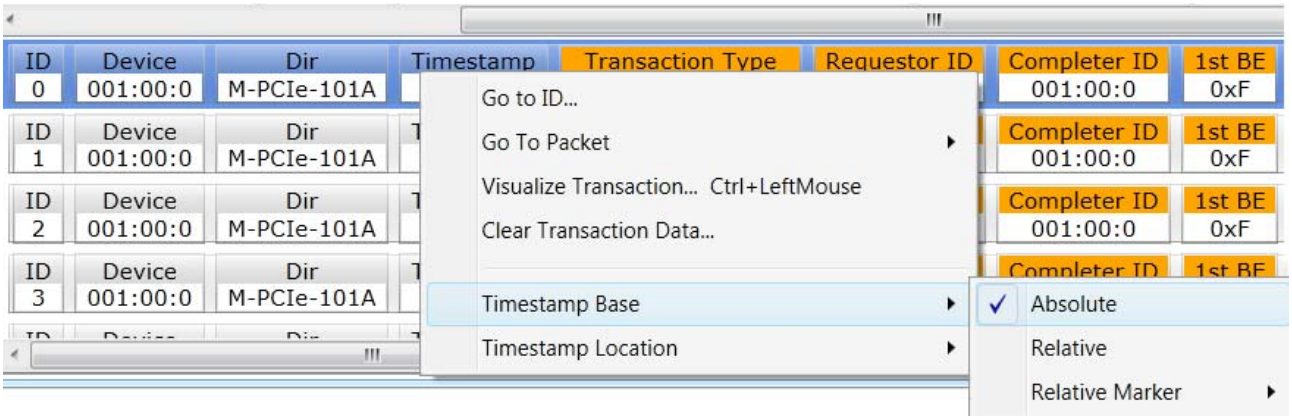
Each computed transaction has a timestamp displayed for it. You can configure:

- where you want the timestamp field to be located for the computed transactions.
- the base to be used for the calculation of timestamps of transactions.

To configure timestamp settings

1 Right-click the **Timestamp** field displayed for a computed transaction.

2 Select **Timestamp Base** from the context-menu.



3 From the displayed submenu, you can select:

- **Absolute -** Timestamp for a transaction is calculated on the basis of the absolute time of the first MPCIe packet applicable for that transaction. This is the default selection.
- **Relative** - Timestamp for a transaction is calculated relative to the time of the first packet applicable for the previous transaction in the list.
- **Relative Marker** - Timestamps are calculated relative to the marker that you select from the displayed submenu.

Changing the timestamp base recalculates the displayed timestamps based on the selected base.

4 Right-click the **Timestamp** field displayed for a computed transaction and then select **Timestamp Location** from the context-menu.

5 By default, timestamps are displayed in the front of transactions data. If required, you can move the display of timestamps to the back of transactions data.

Saving the Computed Transaction Data

Once you computed the transaction data, you can save it in a logic analyzer .ala configuration file. The transaction data is saved along with the captured MPCIe traffic, device setup details, and any other configurations that you made in the Protocol Viewer window.

NOTE You do not need the Transaction Decoder software license to view the saved transaction data. However, if you want to recompute transactions from the saved trace, then you need the Transaction Decoder software license.

To save the computed transaction data

1 Click **File** > **Save as**.

2 In the **Save As** dialog box, specify the name of the file.

3 Ensure that the **Standard Configuration (*.ala)** option is selected as the file type and **All Data and Setup** is selected in the **File Options** group box.

4 Click **Save**.

To access and view previously saved transaction data

1 Click **File** > **Open**.

2 In the **Open** dialog box, navigate to the **Standard Configuration (*.ala)** file in which you saved the transaction data.

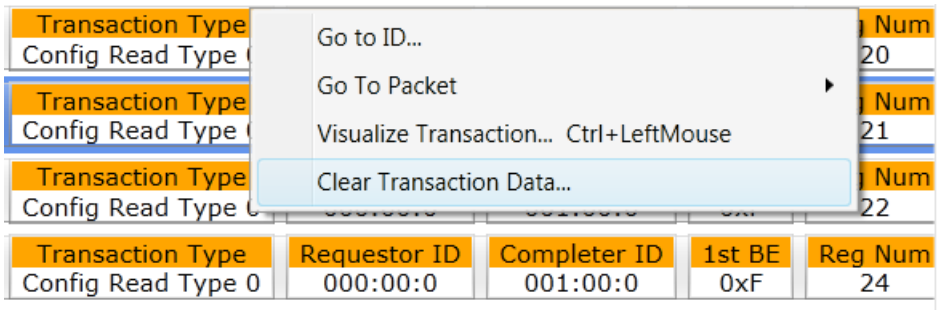
3 Click **Open**.

Clearing the Computed Transaction Data

You can clear the computed transaction data displayed in the Transaction Decode tab.

1 Right-click a transaction displayed in the Transaction Decode tab.

2 Click **Clear Transaction Data...** from the right-click context menu.



3 Click **Yes** to confirm.

The computed transactions display is cleared.

If the transactions data was previously saved in an .ala file, then reloading the .ala file redisplays the transactions data after it has been cleared. If the transactions data was not previously saved, you can specify the data range and then click **Compute** again to redisplay the transactions data.

Interpreting and Navigating Through the Transaction Decode Results

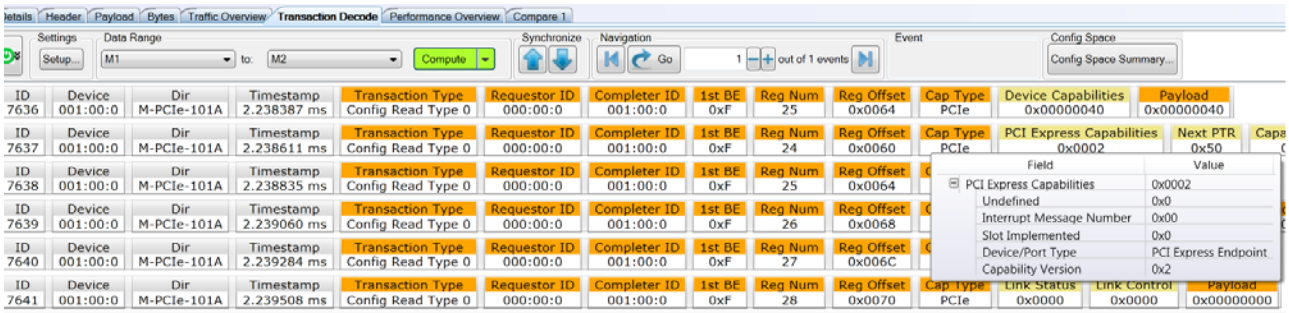
Transactions are decoded and computed from a set of captured packets that contain the storage protocol information associated with that transaction.

Decoded transactions are displayed in the *Transaction Decode* tab in an order based on the timestamp of the first packet associated with a transaction. One or many packets may be associated with a transaction.

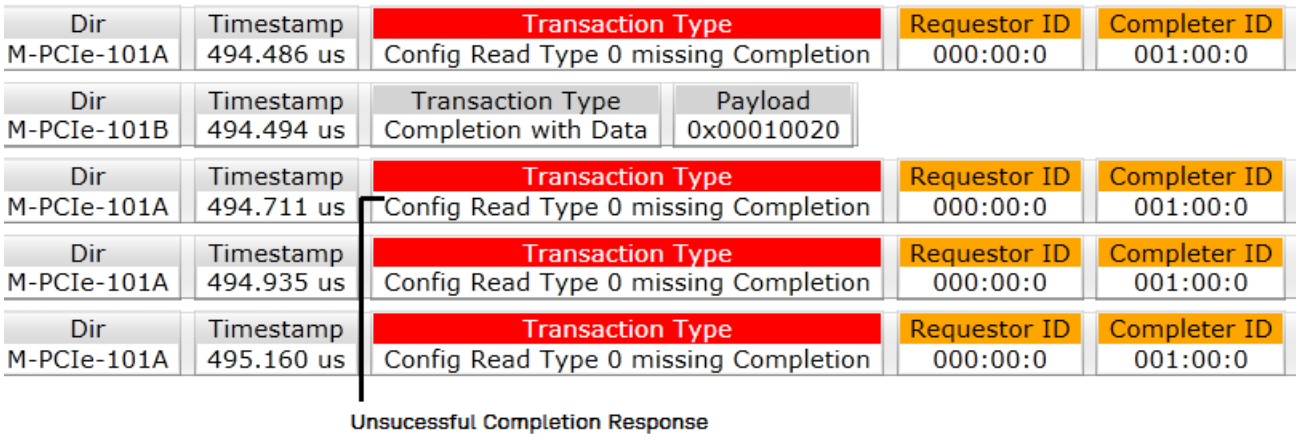
Transaction Details Pane

In the left pane of the Transaction Decode tab, transactions with their details are displayed. In this pane, a transaction is uniquely identified by a Transaction ID.

The details displayed for transactions may vary based on their type and the applicable storage protocol. For instance, moving the mouse pointer to a field of a decoded MPCIe transaction presents a tool tip with information about that field.



Different colors are used to clearly indicate transactions of different types. Errored transactions are displayed in red.



NOTE

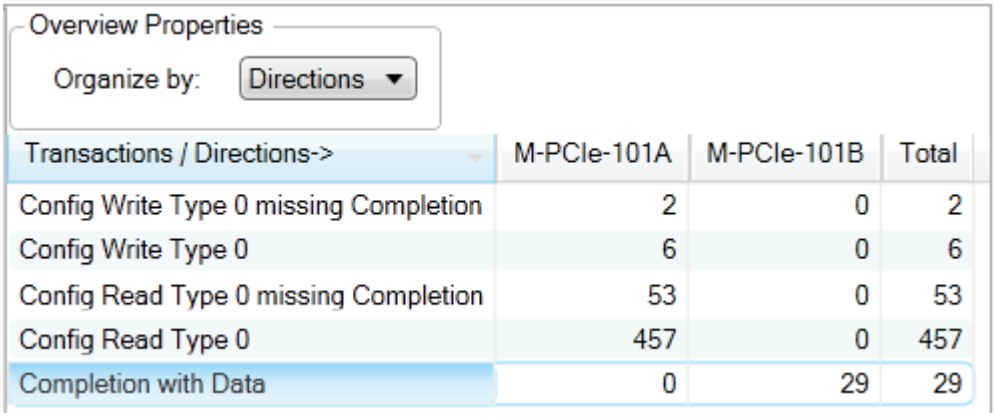
If you are displaying data captured from multiple U4431A modules in a single Protocol Viewer instance, then transactions from these multiple modules will be interleaved based on the timestamp of the first packet in the transaction.

To display transactions for each module separately, you can add a Protocol Viewer instance to each module in the Overview pane of the Logic and Protocol Analyzer GUI.

Transaction Overview Pane

The right pane of the Transaction Decode tab displays statistics for the computed transactions. This pane lists the transaction types applicable for the computed transaction data and the number of events/occurrences for each transaction type in the computed data.

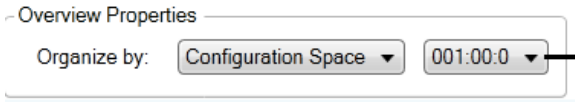
The Overview pane for the SSIC transactions is described in the "Viewing SSIC Transactions" on page 134.



Transactions / Directions->	M-PCIe-101A	M-PCIe-101B	Total
Config Write Type 0 missing Completion	2	0	2
Config Write Type 0	6	0	6
Config Read Type 0 missing Completion	53	0	53
Config Read Type 0	457	0	457
Completion with Data	0	29	29

In this pane, you can organize the number of transaction occurrences/events on the basis of the organization types described in the following table.

Organized by	Description
Direction	Transaction occurrences are organized on the basis of the Uplink and Downlink directions in the captured data. If the captured data is unidirectional, then this view displays occurrences for one direction only.
Configuration Space	Selecting this option displays the number of occurrences of only MPCIe config space transactions organized on the basis of reads and writes to MPCIe configuration registers in the DUT's configuration space.



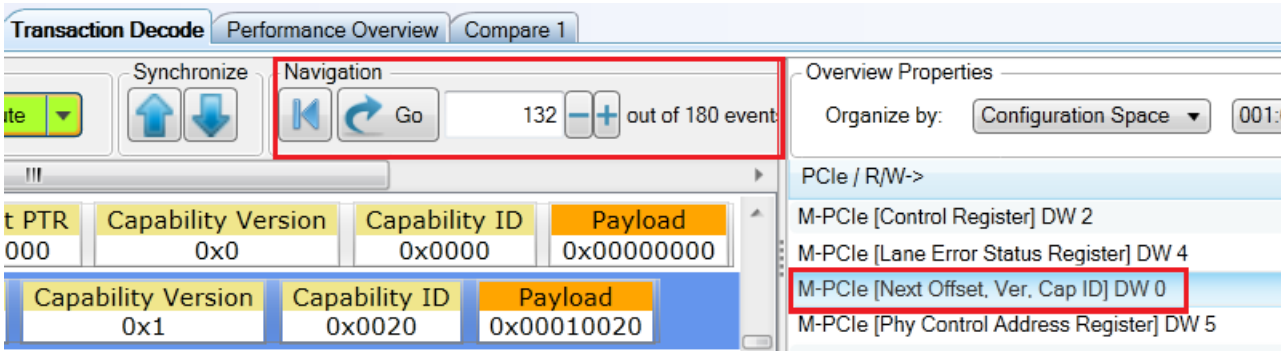
PCIe / R/W->	Read	Write	Total
M-PCIe [Capabilities Register] DW 1	180	0	180
M-PCIe [Control Register] DW 2	180	0	180
M-PCIe [Lane Error Status Register] DW 4	180	0	180
M-PCIe [Next Offset, Ver, Cap ID] DW 0	180	0	180
M-PCIe [Phy Control Address Register] DW 5	180	0	180
M-PCIe [Phy Control Data Register] DW 6	180	0	180
M-PCIe [Status Register] DW 3	180	0	180
MSI(32) [Message Addr (31:0)] DW 1	360	0	360
MSI(32) [Message Control, Next Cap, Cap ID] DW 0	539	0	539
MSI(32) [Message Data] DW 2	360	0	360

Navigating Through Transactions

You can easily navigate through the occurrences of a particular type of transaction in the computed transaction data.

1 From the Transaction Overview pane on the lower-right, select a transaction type whose occurrences you want to navigate and view.

The navigation bar in the Transaction Decode tab now displays the total number of occurrences found for the selected transaction type.

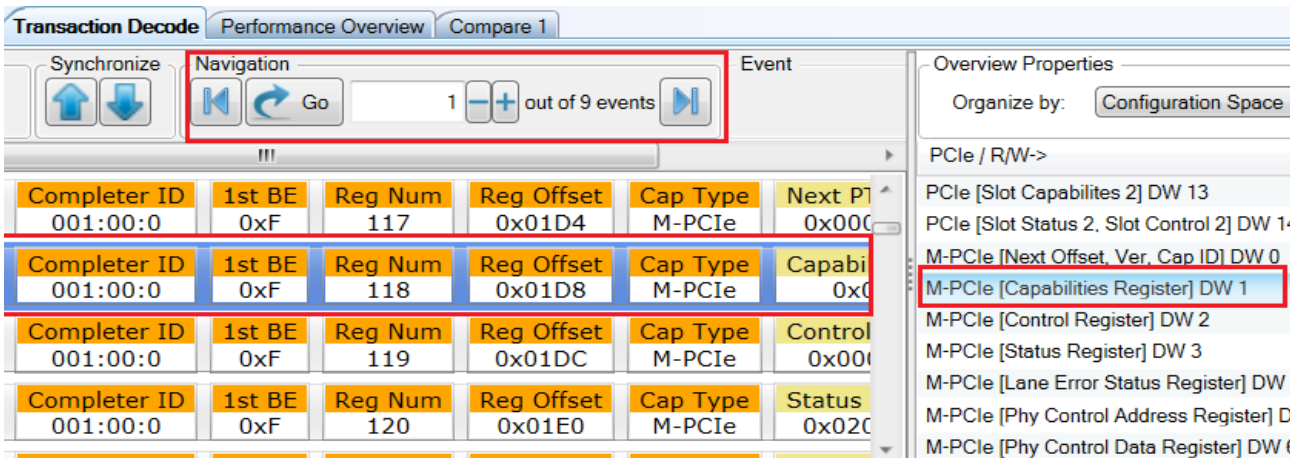


2 To view the first occurrence of that transaction type in the computed data, click button in the navigation bar. Click to go to the last occurrence of the selected transaction type.

3 To sequentially move through the occurrences of a transaction type, use the buttons in the navigation bar.

4 To go to a specific occurrence of a transaction type, specify the event/occurrence number in the text box displayed in the navigation bar and click **Go**.

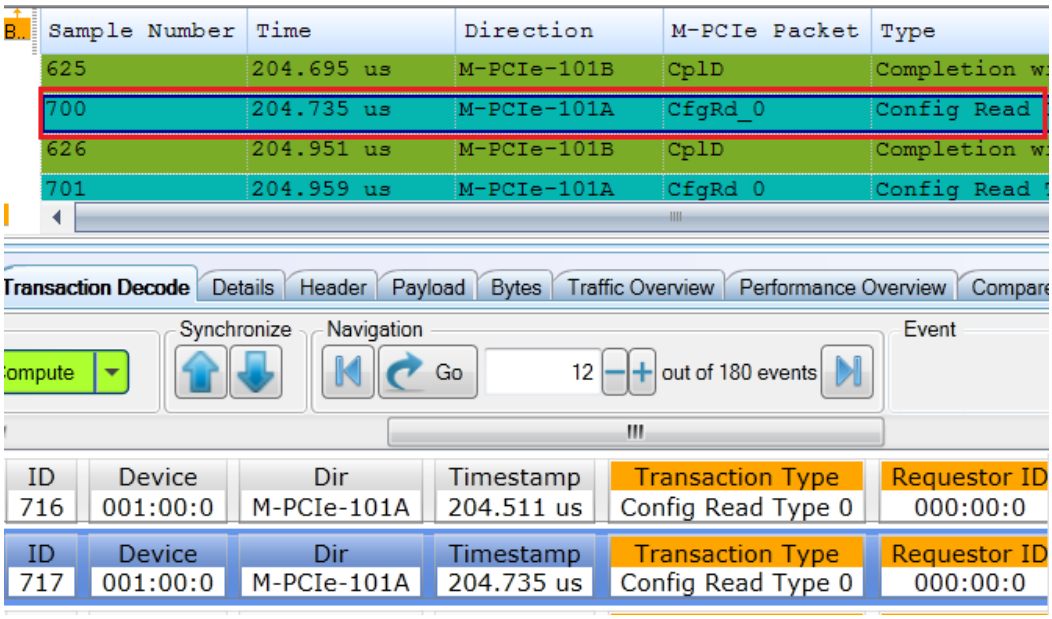
You can also double-click a particular occurrence number in the right pane to navigate to the first transaction mapped to that occurrence number. For instance, in the following screen, the first occurrence of a Config Read transaction to the *MPCIe Capabilities Register* is highlighted on double-clicking its occurrence number in the right pane.



Navigating Between Transactions and their Associated Packets

From a transaction listed in the Transaction Decode tab, you can quickly navigate to the packet(s) exchanged for that particular transaction. To accomplish this, you need to double-click a transaction in the Transaction Decode tab. Doing so, highlights the packet exchanged as the first packet for that transaction in the upper pane of the Protocol Viewer. You can also use the **Synchronize** button in the Transaction Decode tab to synchronize the display of packets data with the selected transaction.

In the following screen, double-clicking the transaction for Config Read to the Capabilities Register highlights the memory read packet for this transaction.



NOTE

You can also quickly navigate from a packet displayed in the upper pane to its applicable transaction in the Transaction Decode tab. To accomplish this, you select the packet and then click the **Synchronize** button displayed in the Transaction Decode tab. This synchronizes the display of transactions with the selected packet.

Navigating to a specific packet of a transaction

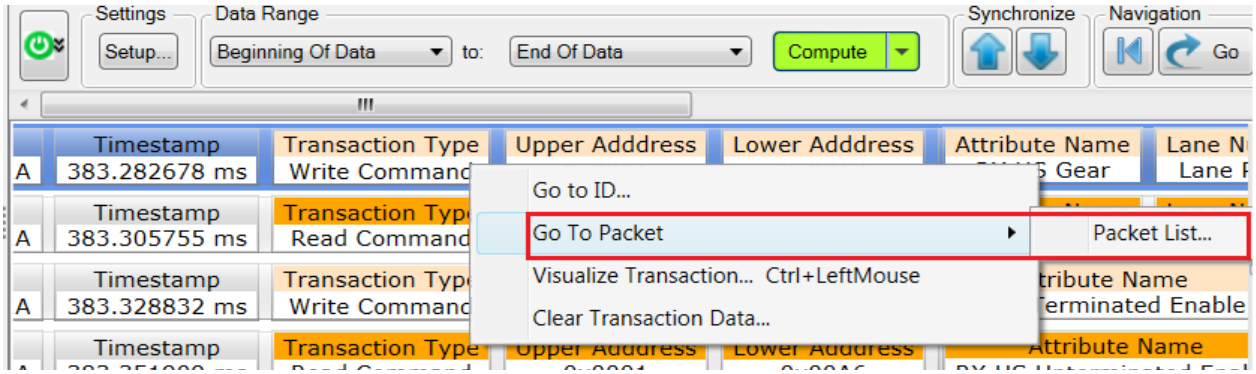
A transaction may have multiple packets associated with it.

For SSIC transactions, the associated packets are listed nested within the applicable transaction. For MPCIe. you can directly navigate to any packet of a transaction. To do this,

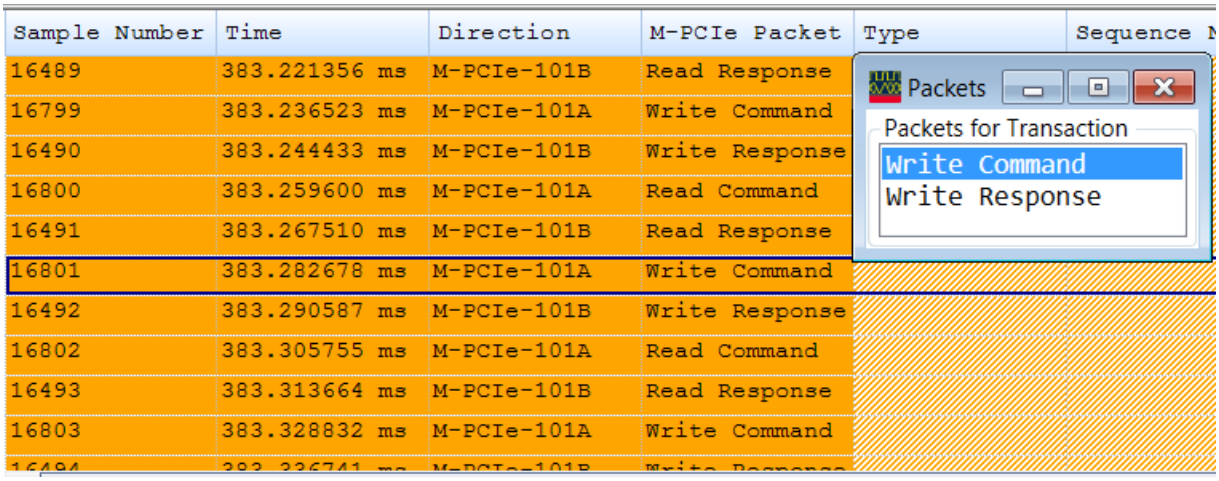
1 Right-click a transaction.

2 Select **Go To Packet > Packet List**.

A list of packets applicable for that transaction are displayed in a sequential order.



3 Select a packet from this list. The packet that you selected gets highlighted in the upper pane of the Protocol Viewer.



Visualizing a Transaction Set (Super Transaction)

This topic is NOT applicable to the SSIC transactions decode. To get information relevant to SSIC transactions, refer to "Viewing SSIC Transactions" on page 134.

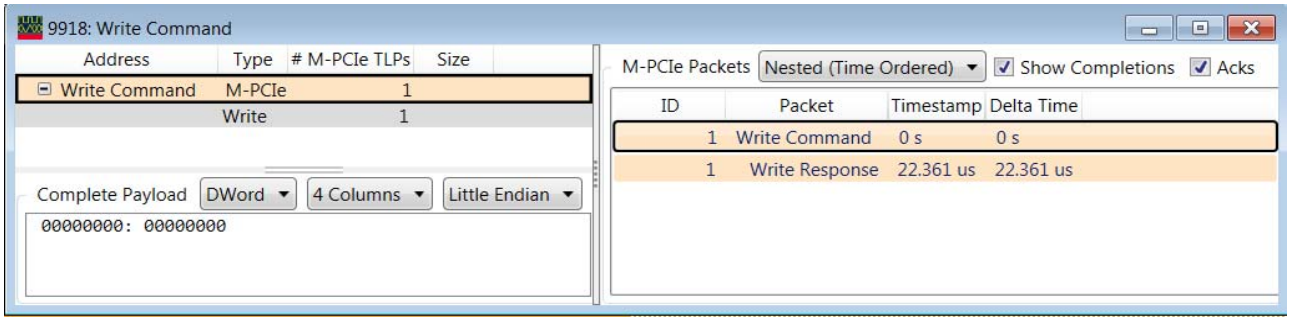
At times, you may want to visualize a decoded transaction as a super transaction with all its related transactions forming a complete set. This provides a visual sequential flow of transactions that were related to a particular transaction.

To visualize a transaction set

1 Right-click a decoded transaction.

2 Select **Visualize Transaction...** from the context menu.

For instance, in the following screen, the *MPCIe* transaction has been visualized as a complete set of transactions.



In the above screen, the transaction set for the MPCIe Write transaction is displayed: It begins with the Write command followed by a response to this Write command.

For each of these related transactions in a set, you can view:

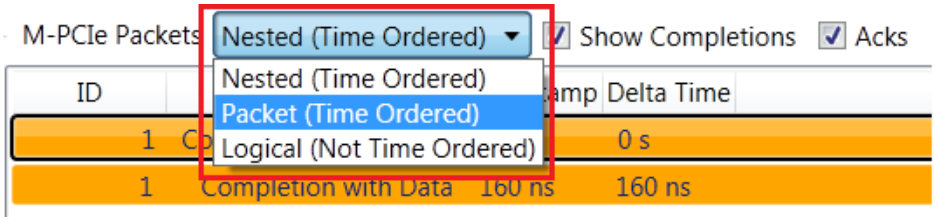
- the associated raw payload in the lower pane.
- the applicable MPCIe packet(s) in the right pane.

NOTE Clicking a super transaction displays the combined payload for the complete set of transactions.

MPCIe Packets applicable for a Transaction Set

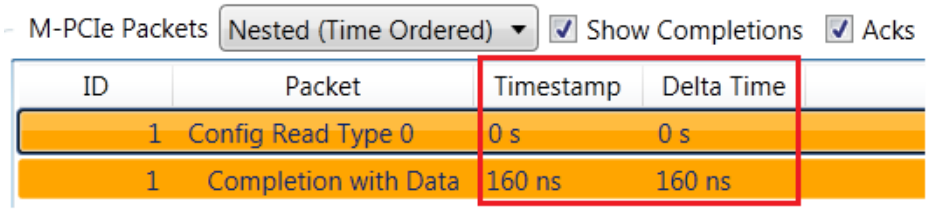
In the right pane of the Visualize Transaction dialog box, by default, MPCIe packets are listed as per the *Nested (Time Ordered)* option. This means packets are listed in a time based sequence with the completion packets nested within the applicable Read packets. If required, you can change this display to:

- list the packets in a logical sequence instead of time based sequence - *Logical (Not Time Ordered)* option.
- list the packets without nesting the completions within the read packets - *Packets (Time Ordered)* option.



For each of the listed packets, the following time related fields are displayed:

- **Timestamp** - This field is relative to the timestamp of the first packet displayed for the transaction set. (The first packet timestamp is taken as 0s.).
- **Delta Time** - This field displays the difference between the timestamp of the currently selected packet and the previous packet in the list of packets for the transaction set.



You can enable or disable the display of Completions and Acks packets from the packets list in the right pane using the **Show Completions** and **Acks** checkboxes respectively.

Viewing NVMe Transactions

Previous help topics described how you can configure and compute transactions in the Transaction Decode tab. This topic is specific to NVMe transactions and provides examples of how you can interpret NVMe transactions that are computed and displayed in the Transaction Decode tab.

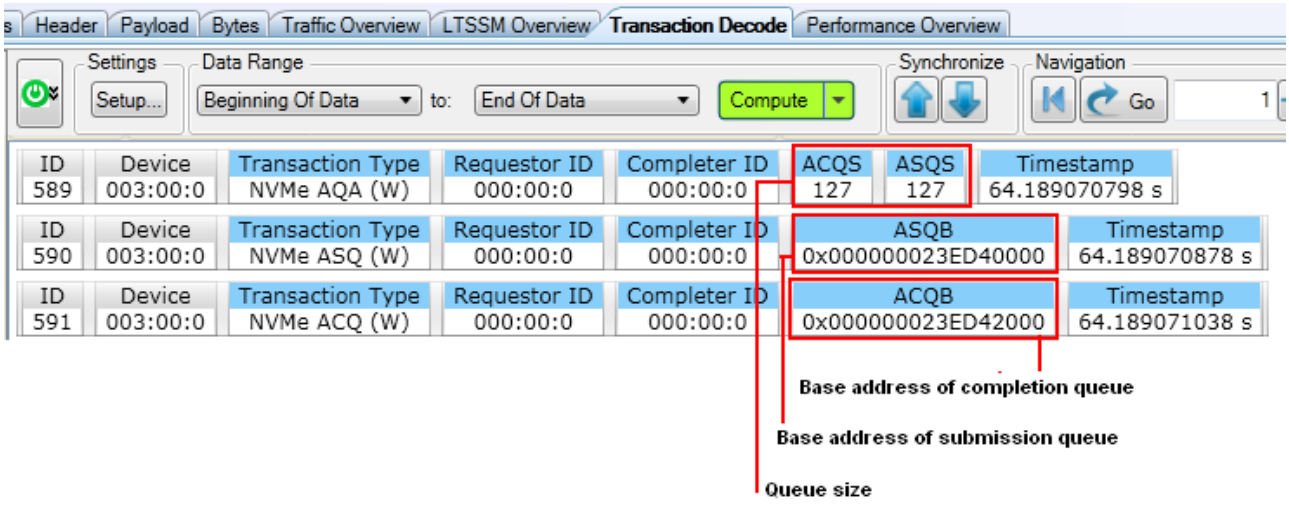
The decoded NVMe transactions provide you a sequential view of the communication cycle between the host software and NVMe controller for various requests placed by the host software in queues. You can check how the NVMe controller responds to and completes these requests. At the administrative management level, you can verify how the NVMe controller handles admin requests such as queue management requests.

For NVMe, the Transaction Decode tab displays transactions for:

- NVMe controller initialization such as Admin queue configuration
- NVMe Admin and I/O commands submission and their responses (completions).
- MSI-X interrupts initiations by controller
- NVMe I/O submission and completion queues management

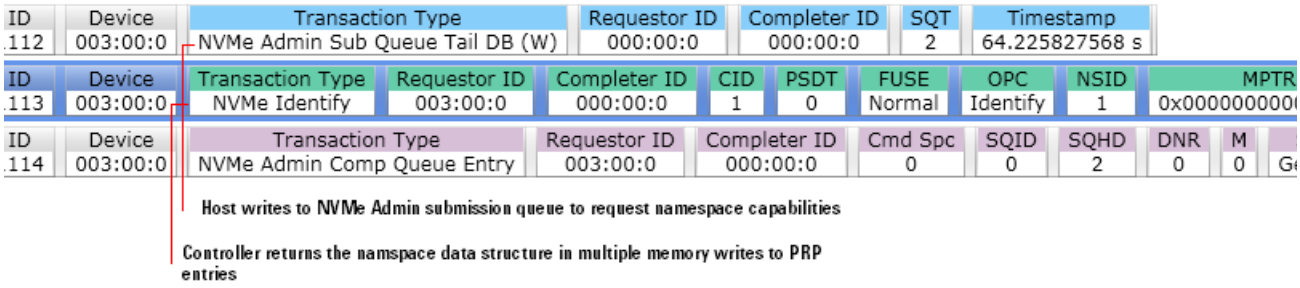
Viewing Transactions for NVMe Controller Initialization

The following screen displays a set of transactions for NVMe controller initialization. The *AQA*, *ASQ*, and *ACQ* registers are modified to define the Admin Submission Queue and its corresponding Admin Completion Queue. The transactions display the Admin queue attributes in terms of size (128 entries) and the base address (64-bit physical address) to be used for Admin submission queue and completion queue.

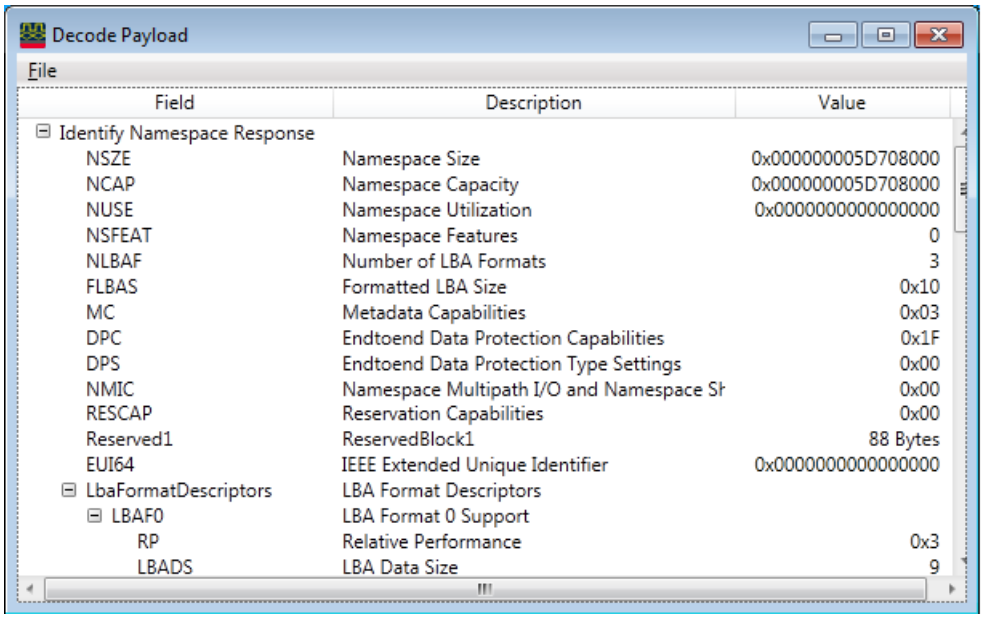


Viewing Admin Command Transactions

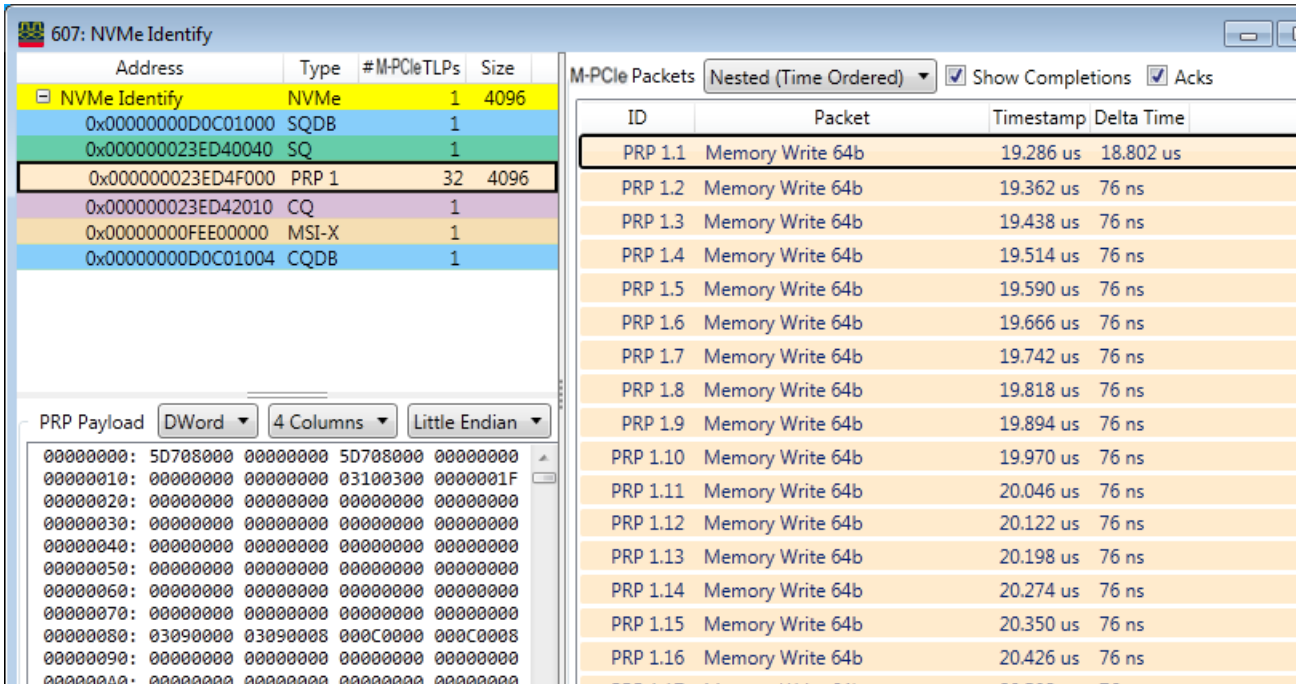
The following screen displays transactions related to the *Identify* command submission and completion to return capabilities and status of a specific namespace.



You can further view the namespace capabilities returned by the Identify command by right-clicking the NVMe Identify transaction and selecting **Decode Payload**.

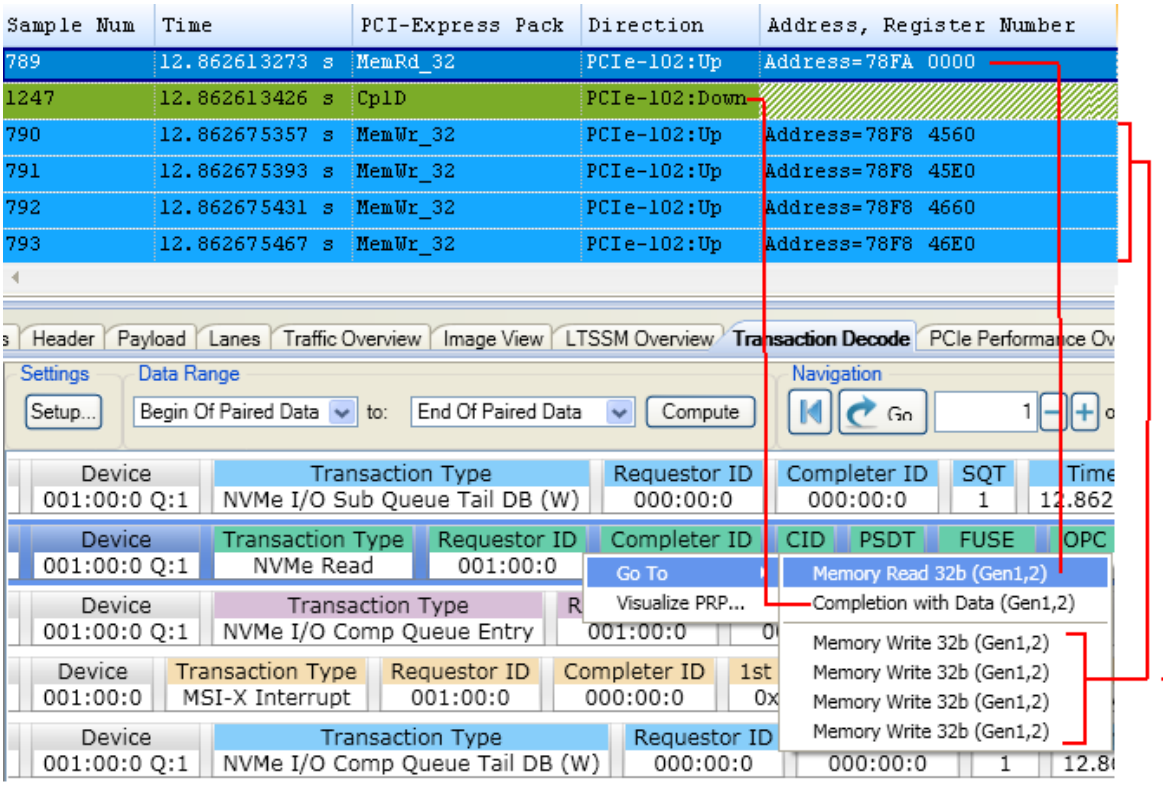


You can also visualize how the data returned by the Identify command is stored in specific PRP entries by right-clicking the transaction and selecting **Visualize Transaction...** (see page 121 for details).



Viewing NVMe I/O Command Transactions

The following screen displays an *NVMe Read* transaction to read data from the starting LBA specified in the Read command.



For this transaction, the controller:

- first performs a *Memory Read* to read the request entry from the base address of submission queue 1.
- then fetches the requested data from system memory
- then performs multiple Memory Writes to write this data to the applicable PRP entries for data transfer to host.

In the transaction following the NVMe Read transaction, the controller updates the completion queue 1 with the status of the Read command completion.

Viewing a Complete Set of Transactions for a Command Submission and Completion

The following screen displays a complete set of NVMe transactions between the host and controller. This set of five transactions represents the steps involved in the *NVMe Write* command submission and completion process. 2

ID	Device	Transaction Type	Requestor ID	Completer ID	SQT	Timestamp
999	001:00:0 Q:3	NVMe I/O Sub Queue Tail DB (W)	000:00:0	000:00:0	12	14.912247991 s

ID	Device	Transaction Type	Requestor ID	Completer ID	CID	PSDT	FUSE	OPC	NSID
1000	001:00:0 Q:3	NVMe Write	001:00:0	000:00:0	12	0	Normal	Write	1

ID	Device	Transaction Type	Requestor ID	Completer ID	Cmd Spc	SQID	SQHD
1001	001:00:0 Q:3	NVMe I/O Comp Queue Entry	001:00:0	000:00:0	0	3	12

ID	Device	Transaction Type	Requestor ID	Completer ID	1st BE	Address	Payload
1002	001:00:0 Q:3	MSI-X Interrupt	001:00:0	000:00:0	0xF	0xFEE04000	0x000040A0

ID	Device	Transaction Type	Requestor ID	Completer ID	CQH	Timestamp
1003	001:00:0 Q:3	NVMe I/O Comp Queue Tail DB (W)	000:00:0	000:00:0	12	14.912765426 s

The following list describes the set of five transactions displayed above.

1 The first transaction indicates to the controller that a command is submitted for processing.

2 The second transaction represents the NVMe Write command that the host created in the submission queue. The controller reads this command from the submission queue for execution and executes the command.

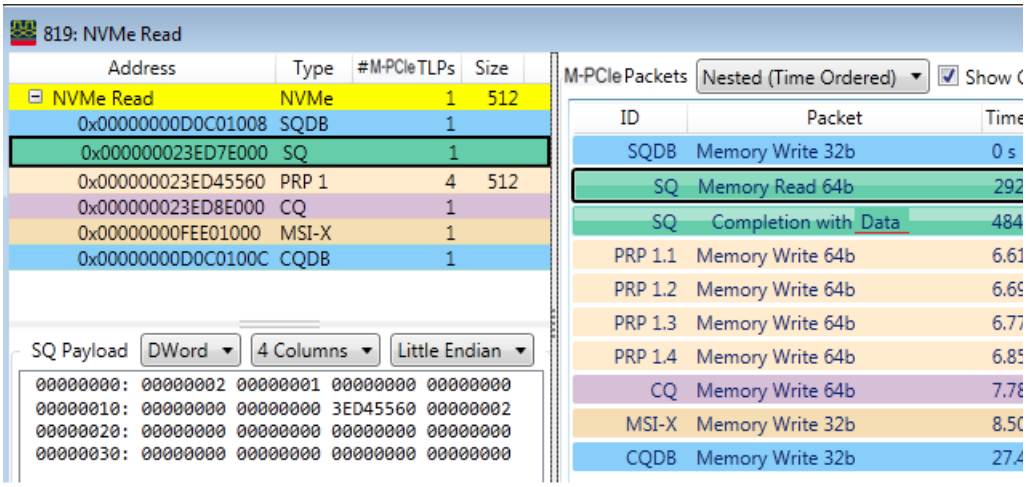
3 The third transaction represents the command completion entry that the controller writes to the completion queue.

4 The fourth transaction represents the MSI-X interrupt generated by the controller to indicate that a completion entry has been added to the completion queue.

5 The fifth transaction indicates to the controller that the host has processed the completion entry that the controller added to the completion queue.

NOTE

You can also visualize a complete set of related transactions (as a super transaction) by right-clicking the transaction and selecting **Visualize Transaction...**. See page 113 for details on this feature. The following is an example of how the set of transactions for an *NVMe Read* command has been sequentially visualized.



Viewing Decoded Payload for NVMe Commands

For NVMe commands that retrieve information from NVMe registers in a specific data structure format, you can view the returned data structure in the same format as defined in the NVMe protocol specifications. For instance, the *Identify* command or the *Get Features* command returns information in a specific data structure format defined in NVMe specifications. The Transaction Decode tab displays the decoded payload for such commands to present the data structure for such commands as per the defined format.

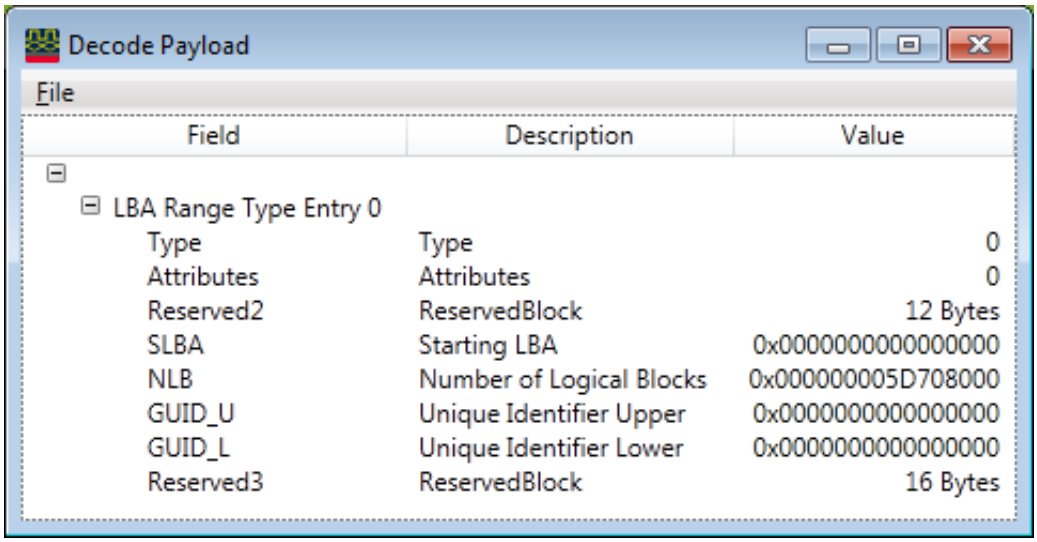
To view the decoded payload of an NVMe command

1 Right-click the transaction displayed for the command in the Transaction Decode tab.

2 Select **Decode Payload...**

The **Decode Payload** dialog box is displayed with the payload fields matching the fields specified in NVMe specifications.

The following screen displays the payload details of the *NVMe Get Features* command. The feature for which the information is retrieved is *LBA Range Type*. The displayed payload is as per the data structure format defined for LBA Range Type in NVMe specifications.



Viewing Decoded PRPs for NVMe Commands

Physical Region Page (PRP) entries are used in read/write transactions to indicate the physical memory locations in system memory. These memory locations are used by controller for data transfers to and from system memory. For a Read request, the PRP entry indicates the memory location where the controller should transfer the data read from system memory. For a Write request, the PRP entry indicates the memory location from where the controller has to gather the data to be written to the system memory.

These addresses can be directly a memory location or a pointer to a location that provides a set of addresses of contiguous memory to perform large read/write operations.

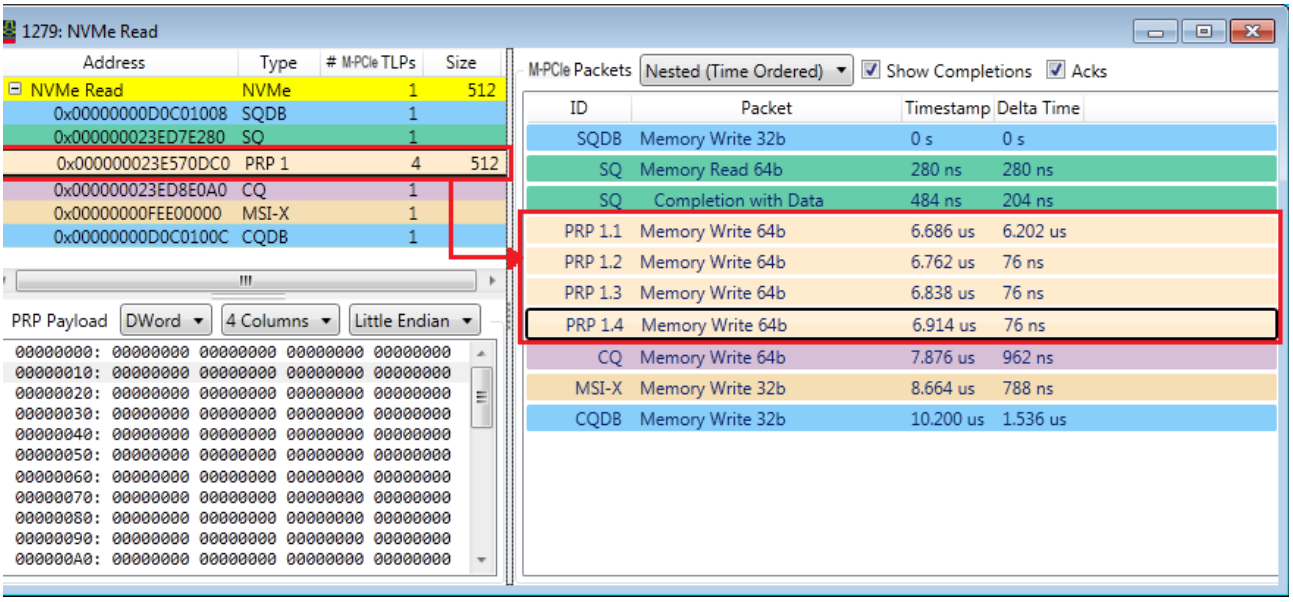
In the Transaction Decode tab, you can view the PRP entries associated with a command that utilizes PRP entries such as an NVMe Read or Write command.

To view the decoded PRPs of an NVMe command

1 Right-click the transaction displayed for the command in the Transaction Decode tab.

2 Select **Visualize Transaction...**

A dialog box is displayed with PRPs and other related transactions associated with the command.



In this dialog box, you can find the following PRP related information.

Component displayed	Description
PRPs applicable for the NVMe command	The displayed PRPs include: ▪ a PRP entry representing an actual physical memory page. ▪ a PRP representing a pointer to a page that defines a PRP List. For such a PRP, a set of PRP entries in a single page of contiguous memory are also displayed underneath. ▪ multiple PRP lists for commands that require multiple PRP Lists for larger read/writes. The last PRP entry in the first list points to the next PRP list used.
Raw payload for each PRP entry	Clicking a PRP entry in the left displays the raw payload for that PRP entry in the lower pane of the dialog box.
Packets associated with each PRP entry	Clicking a PRP entry in the left displays a list of its associated MPCIe packets in the right pane of the dialog box. You can click a packet from this list to navigate directly to that specific packet in the upper pane of Protocol Viewer. This can help you view the complete address range associated with a PRP entry.

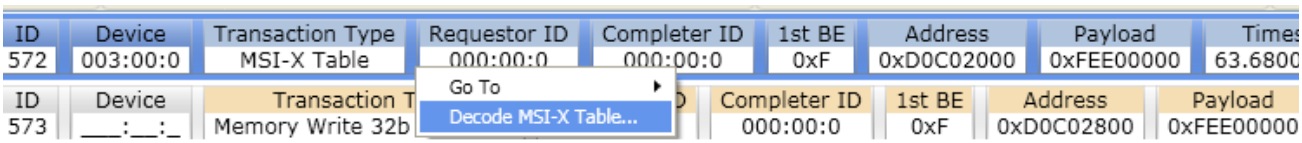
Viewing Decoded MSI-X Table

From the decoded **MSI-X Table** transaction, you can view the index entries of the MSI-X table in a decoded readable format.

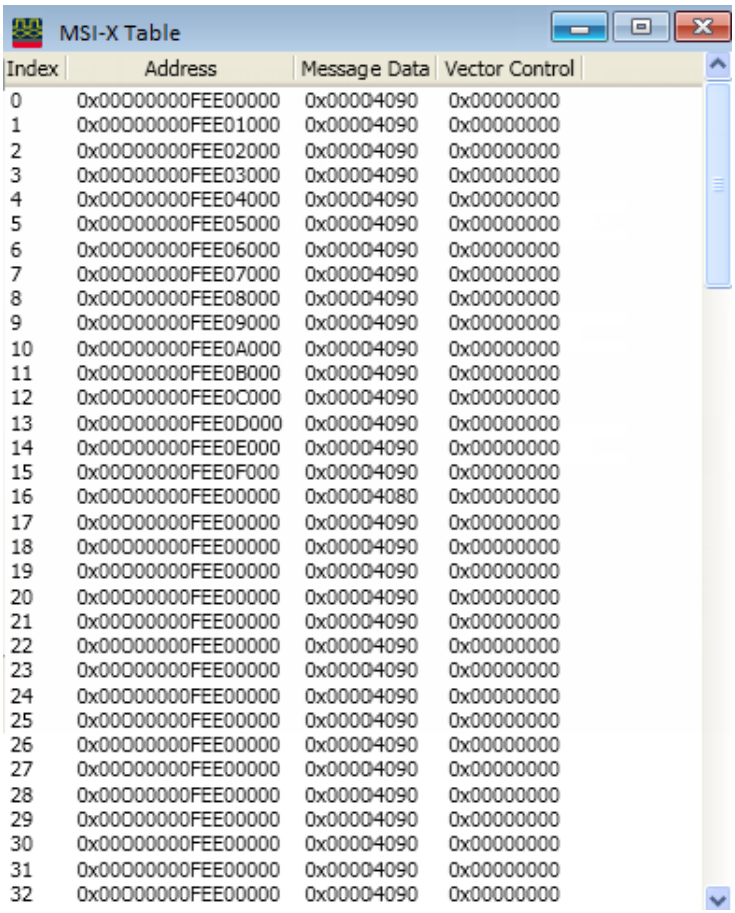
To view index entries of the MSI-X table

1 In the Transaction Decode tab, navigate to the MSI-X Table transaction by double-clicking the **MSI-X Table** transaction type from the right pane.

2 Right-click the highlighted **MSI-X Table** transaction in the left pane of the Transaction Decode tab and then select **Decode MSI-X Table**.

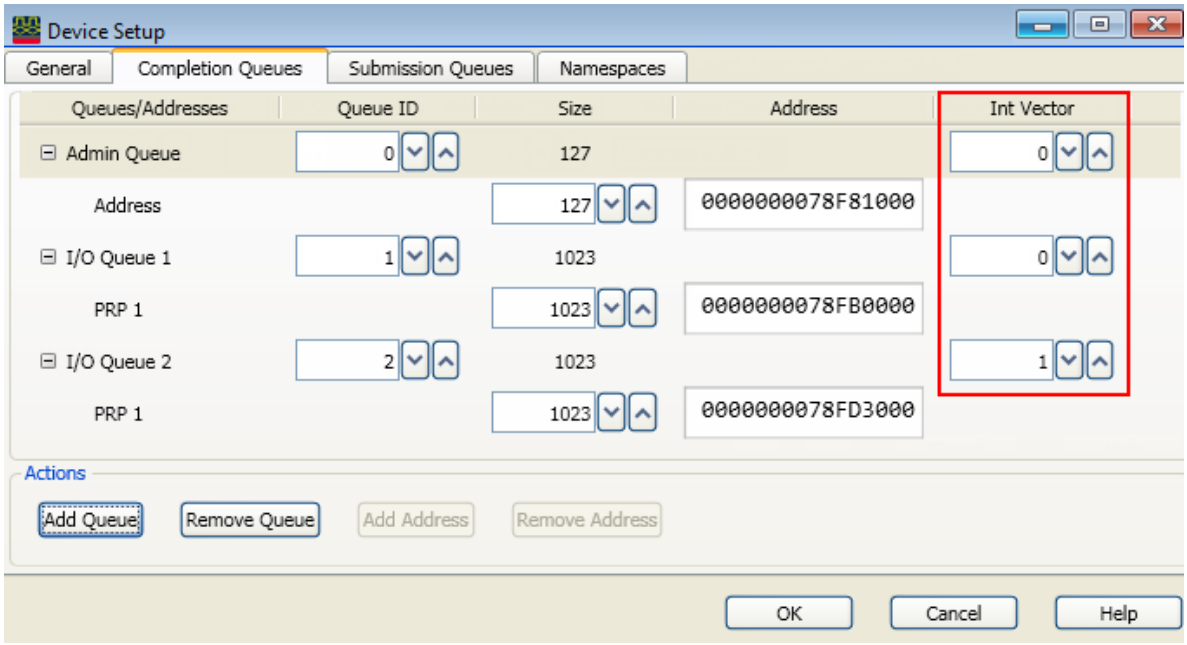


The decoded MSI-X Table is displayed.



Index	Address	Message Data	Vector Control
0	0x00D00000FEE00000	0x00004090	0x00000000
1	0x00D00000FEE01000	0x00004090	0x00000000
2	0x00D00000FEE02000	0x00004090	0x00000000
3	0x00D00000FEE03000	0x00004090	0x00000000
4	0x00D00000FEE04000	0x00004090	0x00000000
5	0x00D00000FEE05000	0x00004090	0x00000000
6	0x00D00000FEE06000	0x00004090	0x00000000
7	0x00D00000FEE07000	0x00004090	0x00000000
8	0x00D00000FEE08000	0x00004090	0x00000000
9	0x00D00000FEE09000	0x00004090	0x00000000
10	0x00D00000FEE0A000	0x00004090	0x00000000
11	0x00D00000FEE0B000	0x00004090	0x00000000
12	0x00D00000FEE0C000	0x00004090	0x00000000
13	0x00D00000FEE0D000	0x00004090	0x00000000
14	0x00D00000FEE0E000	0x00004090	0x00000000
15	0x00D00000FEE0F000	0x00004090	0x00000000
16	0x00D00000FEE00000	0x00004080	0x00000000
17	0x00D00000FEE00000	0x00004090	0x00000000
18	0x00D00000FEE00000	0x00004090	0x00000000
19	0x00D00000FEE00000	0x00004090	0x00000000
20	0x00D00000FEE00000	0x00004090	0x00000000
21	0x00D00000FEE00000	0x00004090	0x00000000
22	0x00D00000FEE00000	0x00004090	0x00000000
23	0x00D00000FEE00000	0x00004090	0x00000000
24	0x00D00000FEE00000	0x00004090	0x00000000
25	0x00D00000FEE00000	0x00004090	0x00000000
26	0x00D00000FEE00000	0x00004090	0x00000000
27	0x00D00000FEE00000	0x00004090	0x00000000
28	0x00D00000FEE00000	0x00004090	0x00000000
29	0x00D00000FEE00000	0x00004090	0x00000000
30	0x00D00000FEE00000	0x00004090	0x00000000
31	0x00D00000FEE00000	0x00004090	0x00000000
32	0x00D00000FEE00000	0x00004090	0x00000000

An MSI-X index entry is associated to a completion queue at the time of creation of the queue. You can view details about the MSI-X index entry associated to a completion queue in the **Completion Queues** tab of the **Setup** dialog box.



Each MSI-X index entry in the MSI-X table has a base address associated to it. This is the address at which the controller writes the MSI-X interrupt for the associated completion queue.

The following screen displays one such MSI-X Interrupt transaction for the completion queue 1.

Viewing AHCI Transactions

This topic is specific to AHCI transactions and provides examples of how you can interpret AHCI transactions that are computed and displayed in the **Transaction Decode** tab. To get information on how you can configure and compute AHCI transactions in the Transaction Decode tab, refer to the following previous topics in this chapter.

- "Transaction Decoding - Overview" on page 94
- "Configuring and Computing Decoded Transactions" on page 95
- "Defining / Verifying the Device Setup" on page 96
- "Interpreting and Navigating Through the Transaction Decode Results" on page 109

For AHCI, the Transaction Decode tab displays transactions for:

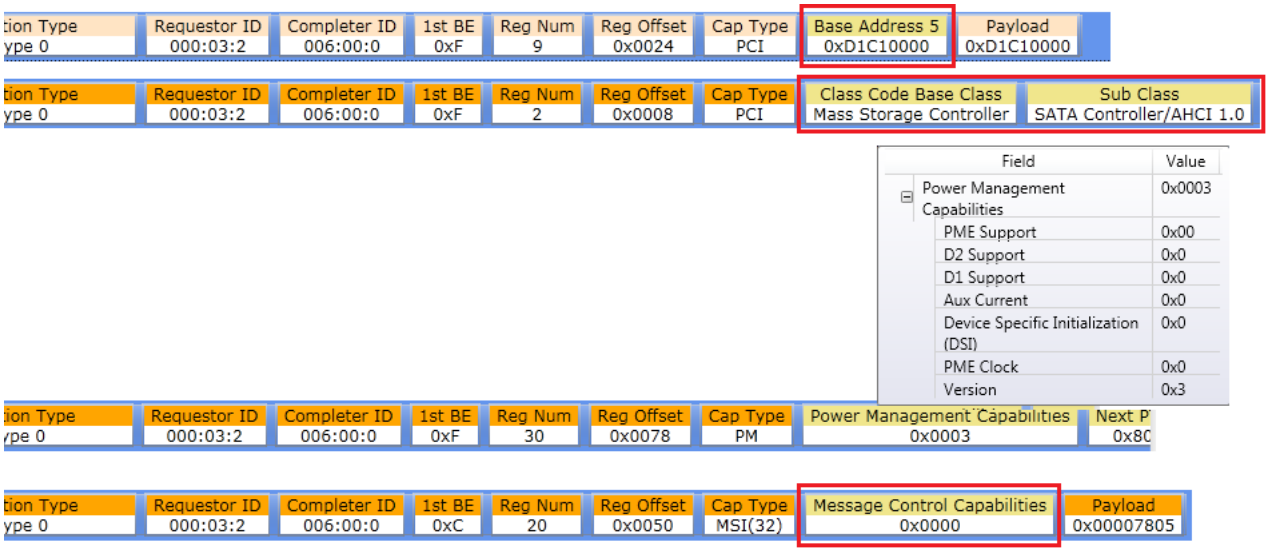
- MPCIe Config space registers
- Generic host control registers
- Port specific registers
- SATA commands

The following sections provide examples of these decoded AHCI transactions.

MPCIe Configuration Space Registers Transactions - Examples

These transactions are MPCIe Config Read and Write requests to various configurations registers in the DUT's config space. You can view the PCI Header and PCI Capabilities for an AHCI device in these transactions. Some examples of the registers that these transactions map to are:

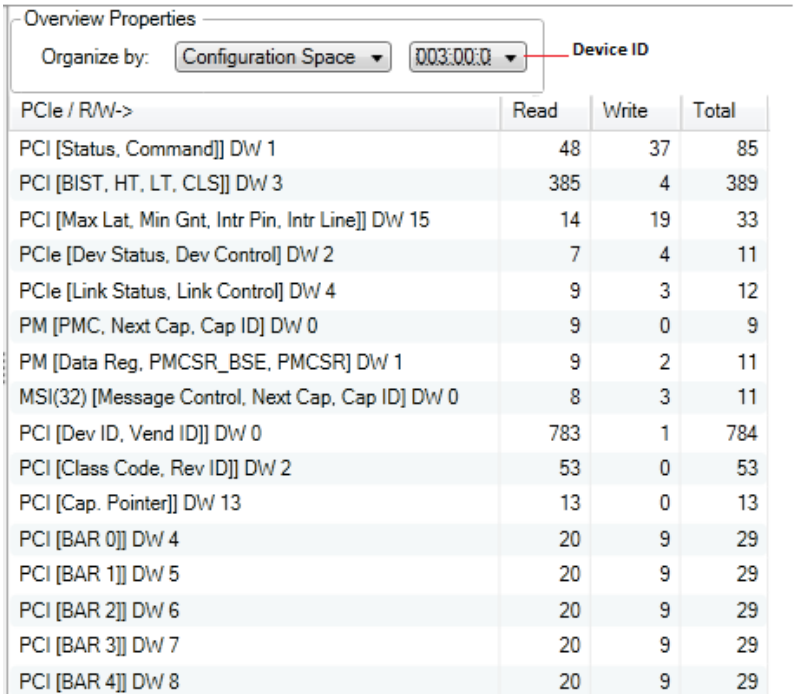
- PCI Header registers such as *Identifier (Device ID), Class Codes, AHCI Base Address <BAR5>, and Capabilities Pointer (CAP).*
- PCI Power Management Capability registers such as *PMCAP*
- Message Signaled Interrupt Capability registers such as *MSICAP*



NOTE

The decoded transactions are a mix of MPCIe Config, generic host control, port-specific and SATA commands transactions. If you are looking for specific MPCIe config space transactions, you may want to use the Overview pane on the right. In this pane, you can choose to organize the registers listing by **Configuration Space**. Doing so, displays the list of MPCIe configuration space registers in this pane. You can then double-click the desired register from this list to navigate to the transaction for this register in the left pane.

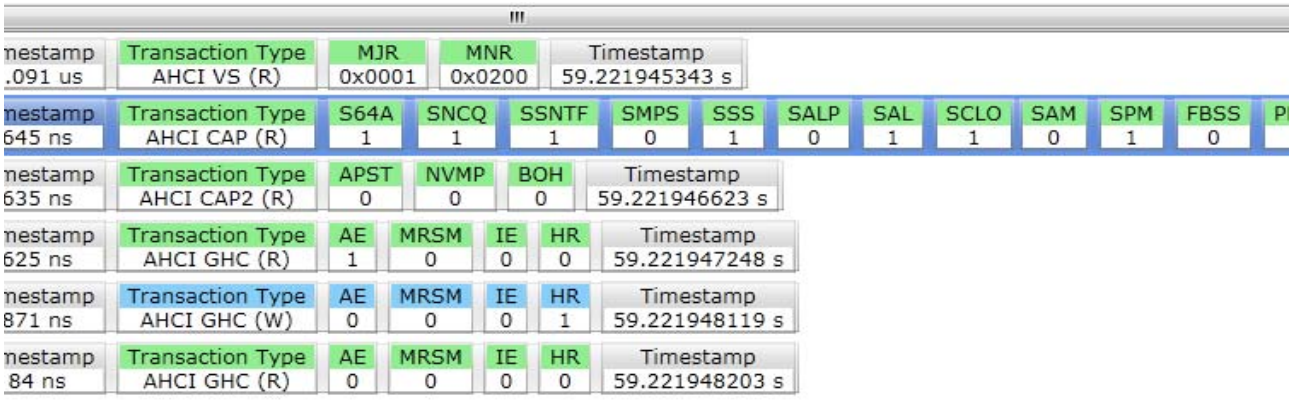
MPCIe config space registers are not displayed in the Overview pane if you have organized this pane by **Directions** or **Ports**.



PCIe / R/W->	Read	Write	Total
PCI [Status, Command]] DW 1	48	37	85
PCI [BIST, HT, LT, CLS]] DW 3	385	4	389
PCI [Max Lat, Min Gnt, Intr Pin, Intr Line]] DW 15	14	19	33
PCIe [Dev Status, Dev Control] DW 2	7	4	11
PCIe [Link Status, Link Control] DW 4	9	3	12
PM [PMC, Next Cap, Cap ID] DW 0	9	0	9
PM [Data Reg, PMCSR_BSE, PMCSR] DW 1	9	2	11
MSI(32) [Message Control, Next Cap, Cap ID] DW 0	8	3	11
PCI [Dev ID, Vend ID]] DW 0	783	1	784
PCI [Class Code, Rev ID]] DW 2	53	0	53
PCI [Cap. Pointer]] DW 13	13	0	13
PCI [BAR 0]] DW 4	20	9	29
PCI [BAR 1]] DW 5	20	9	29
PCI [BAR 2]] DW 6	20	9	29
PCI [BAR 3]] DW 7	20	9	29
PCI [BAR 4]] DW 8	20	9	29

Generic Host Control Registers Transactions - Examples

Generic host control registers such as *AHCI CAP (Host Capabilities)* and *GHC (Global Host Control)* are global HBA registers that are applicable for the entire HBA. The Transaction Decode tab displays decoded transactions for these registers. The following screen shows some such transactions for generic host control registers.



nestamp	Transaction Type	MJR	MNR	Timestamp
.091 us	AHCI VS (R)	0x0001	0x0200	59.221945343 s

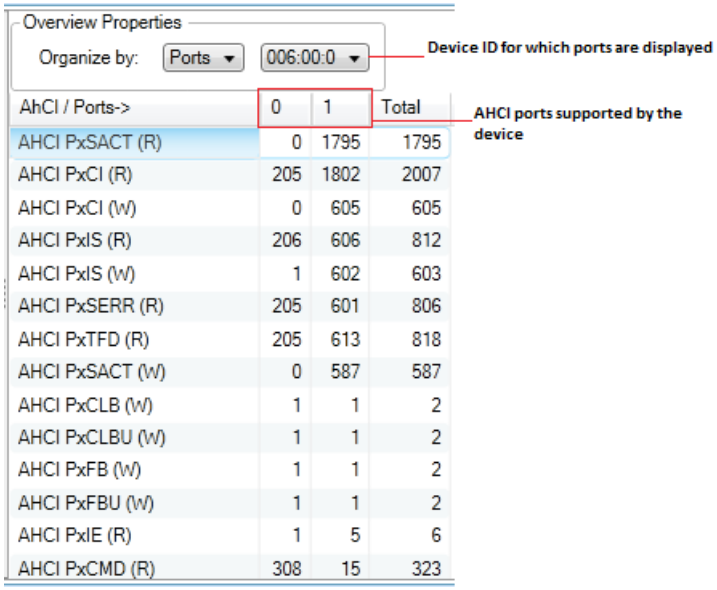
nestamp	Transaction Type	S64A	SNCQ	SSNTF	SMPS	SSS	SALP	SAL	SCLO	SAM	SPM	FBSS	P
645 ns	AHCI CAP (R)	1	1	1	0	1	0	1	1	0	1	0	

nestamp	Transaction Type	APST	NVMP	BOH	Timestamp
635 ns	AHCI CAP2 (R)	0	0	0	59.221946623 s

nestamp	Transaction Type	AE	MRSM	IE	HR	Timestamp
625 ns	AHCI GHC (R)	1	0	0	0	59.221947248 s
nestamp	Transaction Type	AE	MRSM	IE	HR	Timestamp
871 ns	AHCI GHC (W)	0	0	0	1	59.221948119 s
nestamp	Transaction Type	AE	MRSM	IE	HR	Timestamp
84 ns	AHCI GHC (R)	0	0	0	0	59.221948203 s

Port Specific Registers Transactions - Examples

AHCI port specific registers such as *PcCLB*, *PxIS*, *PxSACT* and *PxCDM* are applicable for each port supported by an AHCI device. The read and write transactions to these port registers are also displayed in the Transaction Decode tab. The following screen displays some examples of such ports register accesses.

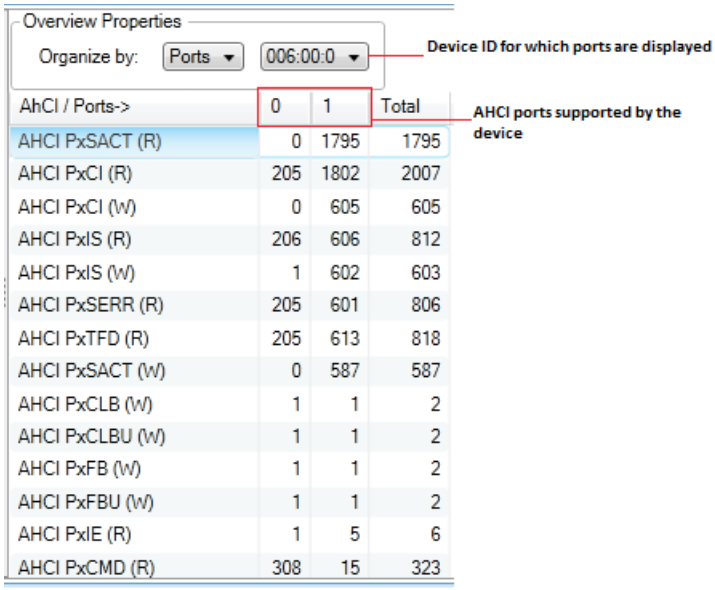


AhCI / Ports->	0	1	Total
AHCI PxSACT (R)	0	1795	1795
AHCI PxCI (R)	205	1802	2007
AHCI PxCI (W)	0	605	605
AHCI PxIS (R)	206	606	812
AHCI PxIS (W)	1	602	603
AHCI PxSERR (R)	205	601	806
AHCI PxTFD (R)	205	613	818
AHCI PxSACT (W)	0	587	587
AHCI PxCLB (W)	1	1	2
AHCI PxCLBU (W)	1	1	2
AHCI PxFB (W)	1	1	2
AHCI PxFBU (W)	1	1	2
AHCI PxIE (R)	1	5	6
AHCI PxCMD (R)	308	15	323

NOTE

The decoded transactions are a mix of MPCIe Config, generic host control, port-specific and SATA commands transactions. If you are looking for a particular type of port specific transactions, you may want to use the Overview pane on the right. In this pane, you can choose to organize the registers listing by **Ports**. Doing so, displays the list of port registers in this pane and organize the transaction occurrences by ports. You can then double-click the desired register from this list to navigate to the transaction for this register in the left pane.

Port registers are not displayed in the Overview pane if you have organized this pane by **Configuration Space**.



AhCI / Ports->	0	1	Total
AHCI PxSACT (R)	0	1795	1795
AHCI PxCI (R)	205	1802	2007
AHCI PxCI (W)	0	605	605
AHCI PxIS (R)	206	606	812
AHCI PxIS (W)	1	602	603
AHCI PxSERR (R)	205	601	806
AHCI PxTFD (R)	205	613	818
AHCI PxSACT (W)	0	587	587
AHCI PxCLB (W)	1	1	2
AHCI PxCLBU (W)	1	1	2
AHCI PxFB (W)	1	1	2
AHCI PxFBU (W)	1	1	2
AHCI PxIE (R)	1	5	6
AHCI PxCMD (R)	308	15	323

SATA Commands Transactions - Examples

The decoded AHCI transactions also include transactions for SATA Commands such as *Identify Device, Set Features, Read FPDMA Queued, Write FPDMA Queued,* and *NOP*.

The following screens display some examples of decoded SATA Commands transactions.

Transaction Type	Port	Command Header	Address	Features	Command	C	R	R1	R2	PM Port	F
AHCI ATA Command SetFeature (R)	1	4	0x13E64ED00	Enable write cache	SET FEATURES	1	0	0	0	0x0	

Transaction Type	Port	Command Header	Address	Features	Command	C	R	R1	R2	PM Port	FIS
AHCI ATA Command RDMA_NCQ (R)	1	28	0x13E64D800	0x00	READ FPDMA QUEUED	1	0	0	0	0x0	0

Transaction Type	Port	Command Header	Address	Features	Command	C	R	R1	R2	PM Port	FIS Type
AHCI ATA Command Identify (R)	1	1	0x13E64D800	0x00	IDENTIFY DEVICE	1	0	0	0	0x0	0x27

Transaction Type	Port	Command Header	Address	Features	Command	C	R	R1	R2	PM Port	F
AHCI ATA Command WRMA_NCQ (R)	1	16	0x13E64D800	0x08	WRITE FPDMA QUEUED	1	0	0	0	0x0	

Visualizing a Complete Set of AHCI Transactions

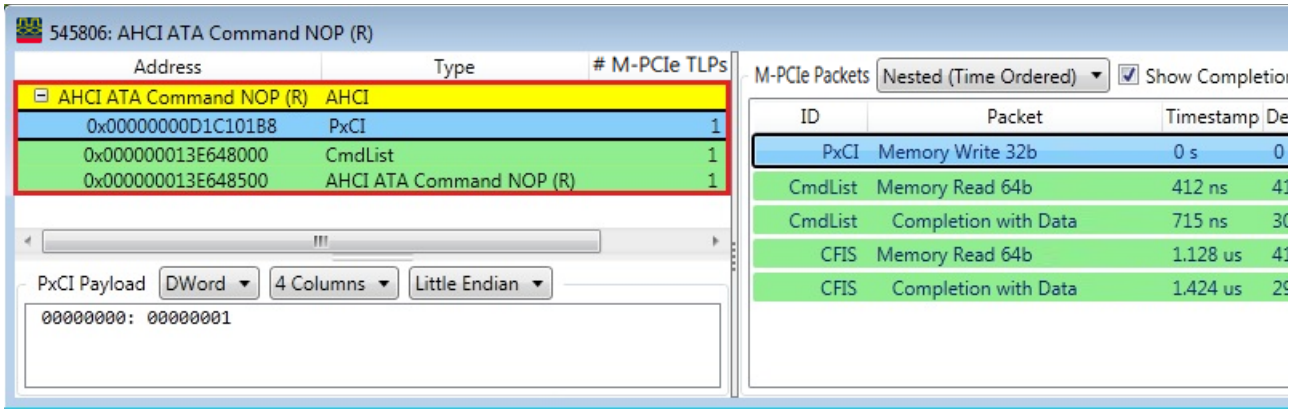
You can view each AHCI transaction displayed in the Transaction Decode tab as a super transaction to visualize its complete set of related transactions. To visualize the transaction set for an AHCI transaction, right-click the transaction and select **Visualize Transaction...**.

NOTE

For a SATA command transaction such as Read FPDMA Queued that has a number of related transactions, the Visualize Transaction feature is particularly useful.

For MPCIe config space or AHCI (generic/port) register transactions, the Visualize Transaction feature displays the read/write access transaction to the register.

Example 1 - Visualize the transaction set for a NOP command transaction



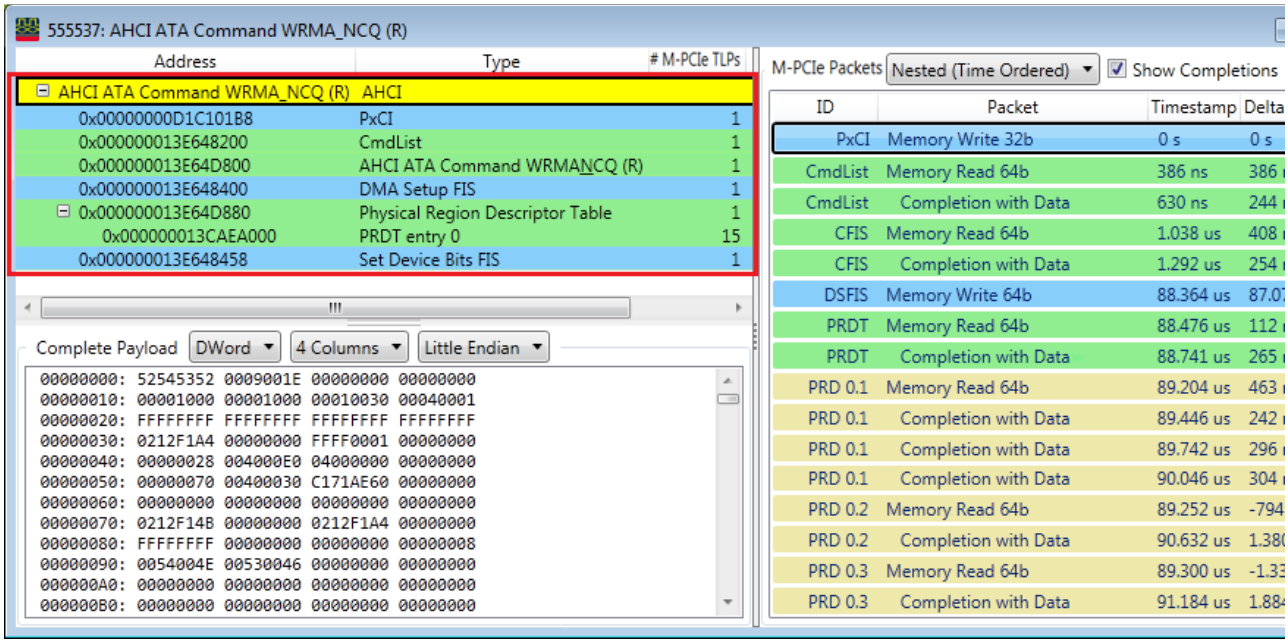
In the above example, the following transactions are a part of the *ATA Command NOP* super transaction.

1 The first transaction is a memory write to the PxCI (Port x Command Issue) register to indicate that a command has been built in memory for a command slot.

2 The second transaction is a memory read for the added command list entry.

3 The third transaction is a memory read of the NOP command from the command table.

Example 2 - Visualize the transaction set for a Write FPDMA Queued command transaction



In the above example, the following transactions are a part of the *ATA Command Write FPDMA Queued* super transaction.

1 The first transaction is a memory write to the PxCI (Port x Command Issue) register to indicate that a command has been built in memory for a command slot.

2 The second transaction is a memory read for the added command list entry.

3 The third transaction is a memory read of the Write FPDMA Queued command from the command table.

4 The fourth transactions is a DMA Setup FIS (DSFIS) memory write to transfer data for the Write FPDMA Queued command in the appropriate slot.

5 The fifth transaction is a PRD Table read transaction. This transaction further expands to display the PRDT entry used for reading the transferred data.

6 The last transaction in the set is a memory write for updating the Set Device Bits FIS. This indicates the completion of the Write FPDMA Queued command.

For more information on visualize transaction, refer to "Visualizing a Transaction Set (Super Transaction)" on page 113.

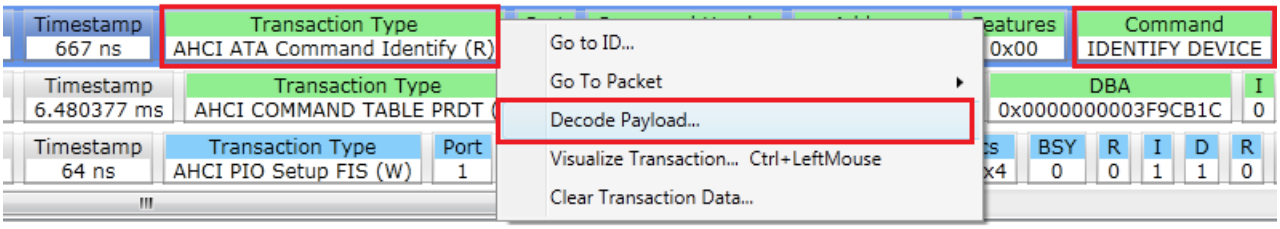
Viewing Decoded Payload for AHCI Commands

For *AHCI ATA IDENTIFY DEVICE* and *IDENTIFY (PACKET) DEVICE* commands that retrieve information in a specific data structure format, you can view the returned data structure in the same format as defined in the Serial ATA specifications. The Transaction Decode tab displays the decoded payload (all 256 words) for these commands to present their data structure as per the defined format.

To view the decoded payload of an AHCI ATA IDENTIFY DEVICE command

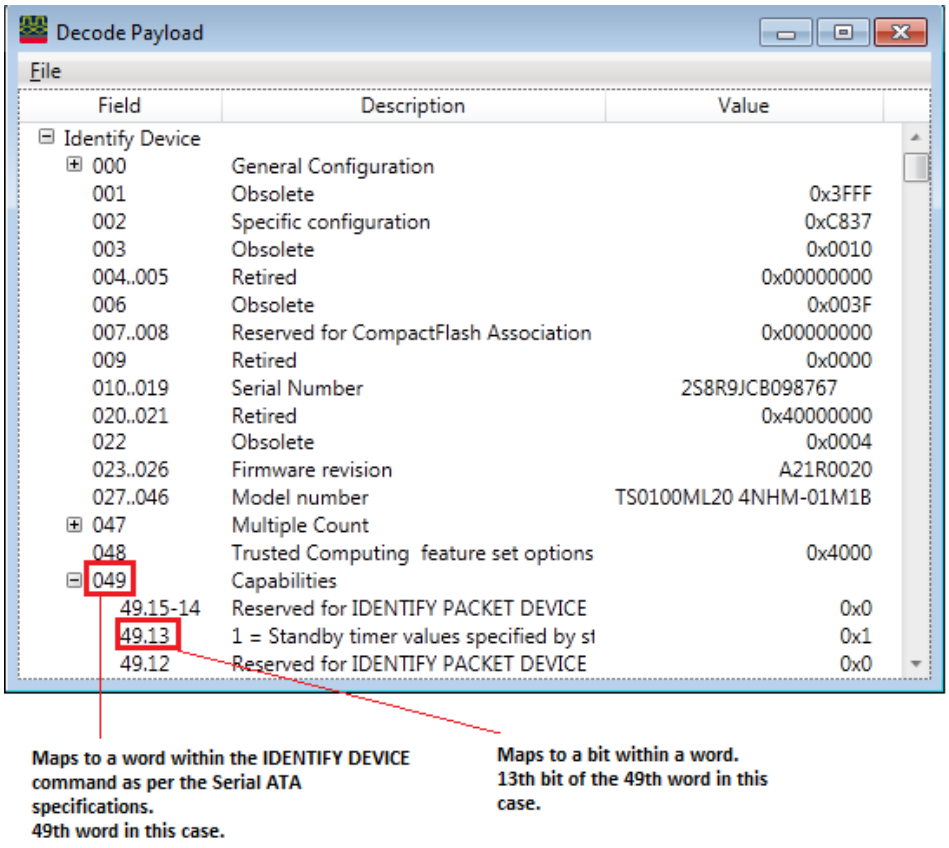
1 Right-click an AHCI ATA IDENTIFY DEVICE transaction in the Transaction Decode tab.

2 Select **Decode Payload...**



The **Decode Payload** dialog box is displayed with the payload fields matching the information structure specified for the IDENTIFY DEVICE command in Serial ATA specifications.

The following screen displays the payload details of the *IDENTIFY DEVICE* command.



Viewing PRDT Entries for an ATA Command

Physical Region Descriptor Table (PRDT) is a table of scatter/gather list. It has zero to many entries. Each entry has a base address and byte counts in system memory where the device reads or writes the actual data transfer if the ATA command is a data transfer command.

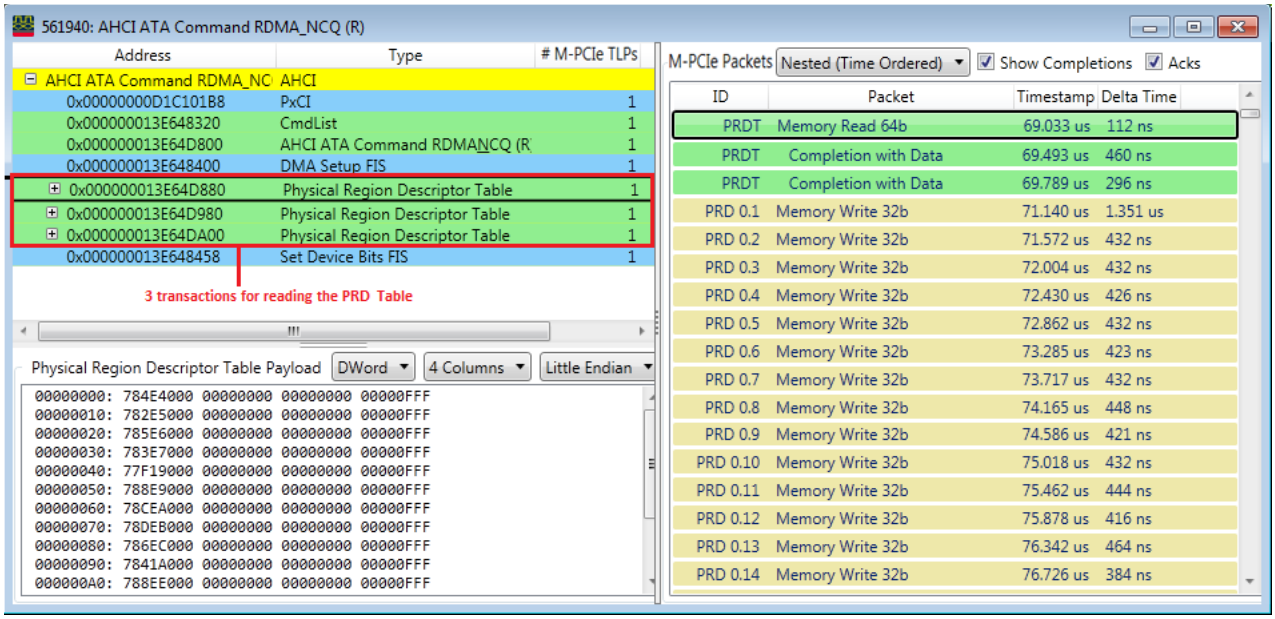
In the Transaction Decode tab, you can view the PRDT entries associated to an ATA command that utilizes PRDT entries such as an *AHCI ATA Identify command* or *AHCI Read FPDMA Queued command*.

To view the decoded PRDT entries of an ATA command

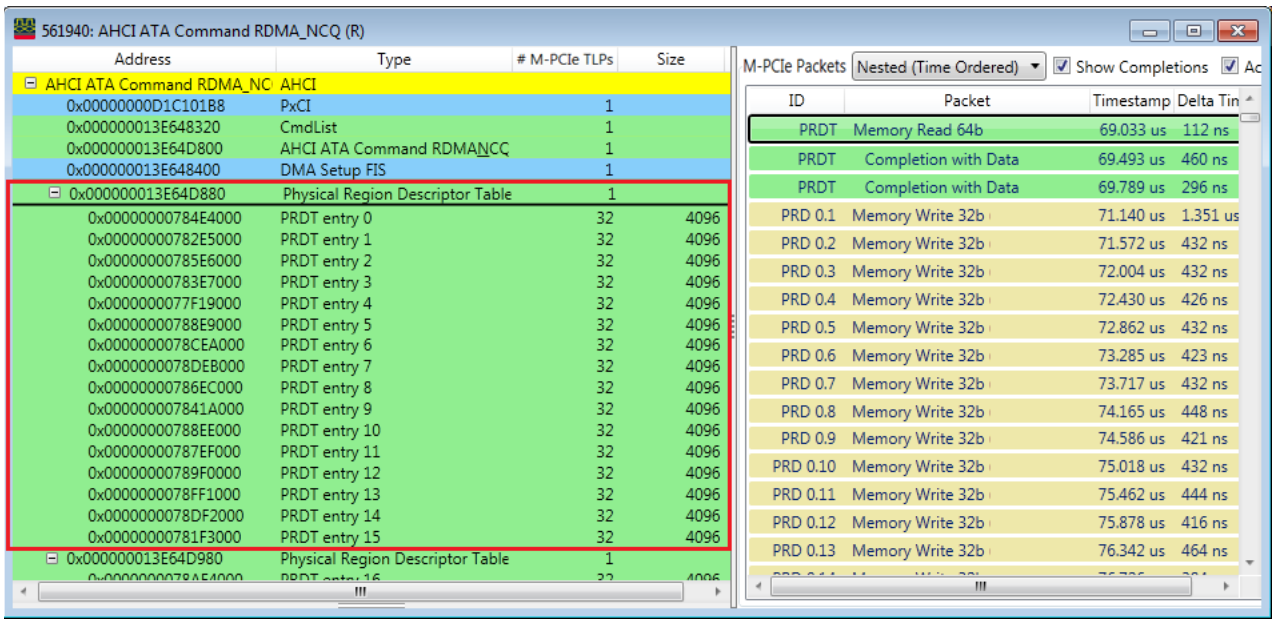
1 Right-click the transaction displayed for the command in the Transaction Decode tab.

2 Select **Visualize Transaction...**

A dialog box is displayed with applicable PRDT transactions and other related transactions for the ATA command. In the following example, the three transactions for reading the PRD table for the Read FPDMA Queued command are highlighted. Clicking one of these PRDT transactions displays the raw payload for that PRDT Read transaction in the lower pane.

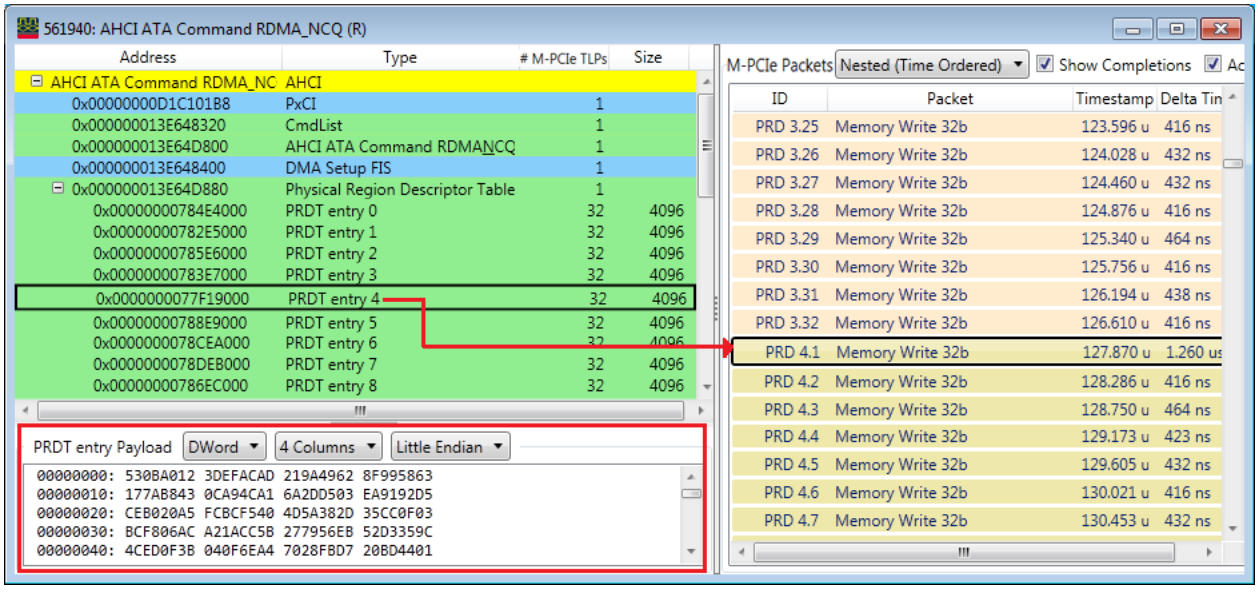


3 Expand each of the displayed PRDT read transactions in the left to view the PRDT entries used for the data transfer. In the following example, the 16 PRDT entries applicable for the first PRDT read transaction are highlighted. With each PRDT entry, its base address, bytes count, and the number of MPCIe TLPs utilized for data transfer are also displayed.



4 Clicking a PRDT entry:

- displays the raw payload for the data transfer for the memory pointed to by that PRDT entry.
- highlights the first MPCIe packet utilized for data transfer for the memory pointed to by that PRDT entry.



Viewing SSIC Transactions

This topic is specific to SSIC transactions and provides examples and information on how you can interpret SSIC transactions that are computed and displayed in the **Transaction Decode** tab. To get information on how you can configure and compute SSIC transactions in the Transaction Decode tab, refer to the following previous topics in this chapter.

- "Transaction Decoding - Overview" on page 94
- "Configuring and Computing Decoded Transactions" on page 95
- "To view/add details of an SSIC device" on page 102
- "Interpreting and Navigating Through the Transaction Decode Results" on page 109

For SSIC, the Transaction Decode tab displays transactions as commands. These commands can be the following requests that the host sends to the device:

- USB standard device requests such as SET_ADDRESS and GET_DESCRIPTOR.
- ACM device subclass requests such as SET_COMM_FEATURE, GET_COMM_FEATURE, and SET_LNE_CODING.
- Network Control Model (NCM) subclass requests such as GET_NET_ADDREESS, GET_CRC_MODE, and SET_NTB_INPUT_SIZE
- Mobile Broadband Interface Model (MBIM) subclass requests such as SEND_ENCAPSULATED_COMMAND and GET_ENCAPSULATED_RESPONSE and also RESPONSE_AVAIABLE and Unsolicited Events notifications from the device to the host.

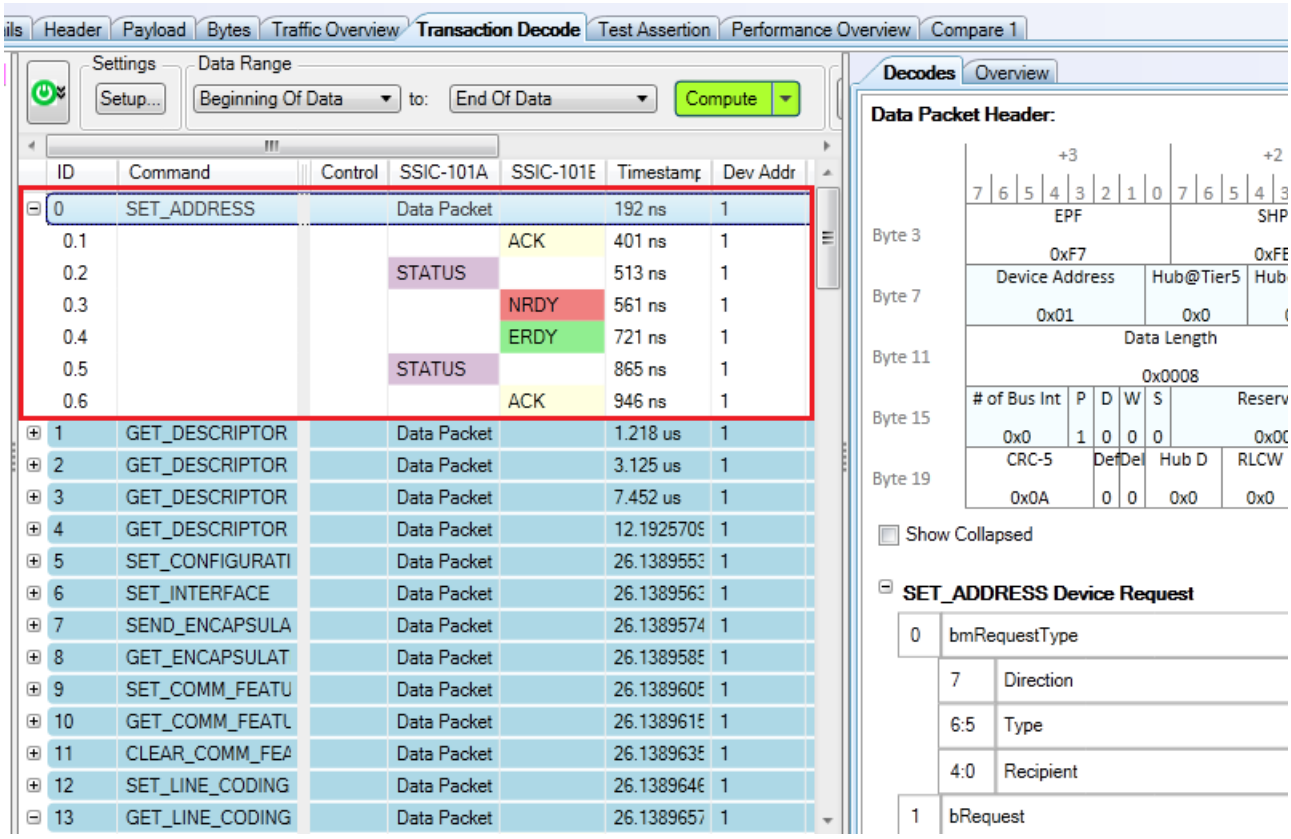
Understanding the SSIC Transactions Display

On expanding an SSIC transaction, you can view all the packets exchanged between the host and device for that transaction (if bidirectional data is captured). Clicking a packet from this list displays the packet's decoded header in the **Decodes** tab on the right. If it is a Data packet, then the Decodes tab also displays the data defining the request sent from the host or the data returned by the device in response to the host's request, as the case may be.

Each transaction row represents the first packet exchanged for that transaction and therefore color coded as per the color coding applicable for this first packet. Clicking the transaction row displays the following information in the Decodes tab on the right.

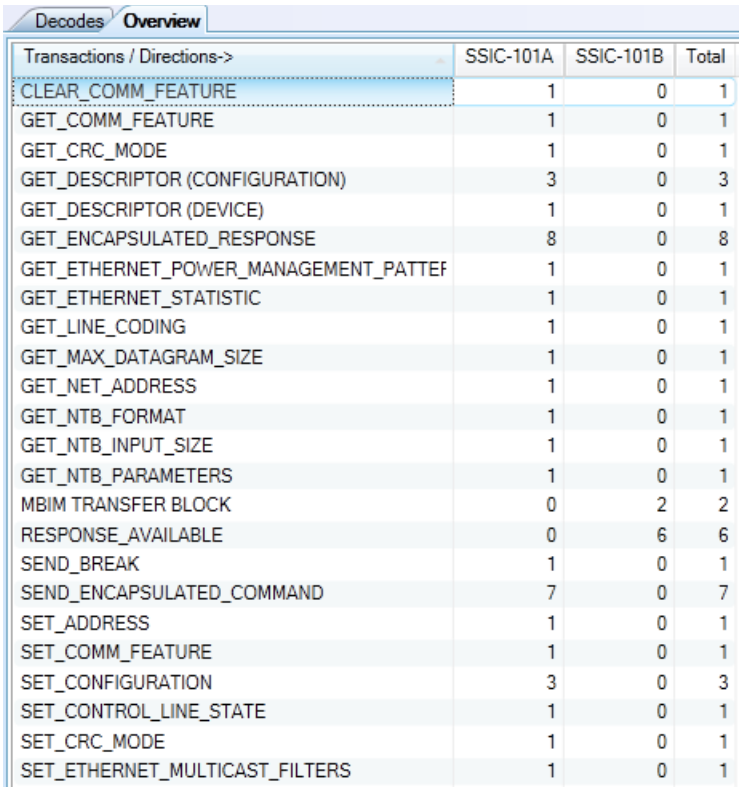
- the header for the first packet of the transaction.
- the data sent by host and device in Data Packet(s) applicable for the transaction.

A sample SSIC transaction for the SET_ADDRESS command is displayed below with all the applicable packets nested within it. The decoded data for the SET_ADDRESS request sent by host is shown in the Decodes tab on the right.



Overview tab

The Overview tab on the right side displays statistics for the computed SSIC transactions. This tab lists the transaction types (device requests) applicable for the computed transaction data and the number of events/occurrences for each transaction type in the computed data. The occurrences are also categorized based on the direction, that is host to device and device to host.



Decodes | Overview

Transactions / Directions->	SSIC-101A	SSIC-101B	Total
CLEAR_COMM_FEATURE	1	0	1
GET_COMM_FEATURE	1	0	1
GET_CRC_MODE	1	0	1
GET_DESCRIPTOR (CONFIGURATION)	3	0	3
GET_DESCRIPTOR (DEVICE)	1	0	1
GET_ENCAPSULATED_RESPONSE	8	0	8
GET_ETHERNET_POWER_MANAGEMENT_PATTEF	1	0	1
GET_ETHERNET_STATISTIC	1	0	1
GET_LINE_CODING	1	0	1
GET_MAX_DATAGRAM_SIZE	1	0	1
GET_NET_ADDRESS	1	0	1
GET_NTB_FORMAT	1	0	1
GET_NTB_INPUT_SIZE	1	0	1
GET_NTB_PARAMETERS	1	0	1
MBIM TRANSFER BLOCK	0	2	2
RESPONSE_AVAILABLE	0	6	6
SEND_BREAK	1	0	1
SEND_ENCAPSULATED_COMMAND	7	0	7
SET_ADDRESS	1	0	1
SET_COMM_FEATURE	1	0	1
SET_CONFIGURATION	3	0	3
SET_CONTROL_LINE_STATE	1	0	1
SET_CRC_MODE	1	0	1
SET_ETHERNET_MULTICAST_FILTERS	1	0	1

The following sections provide a few examples of decoded SSIC transactions of the supported device class and subclasses.

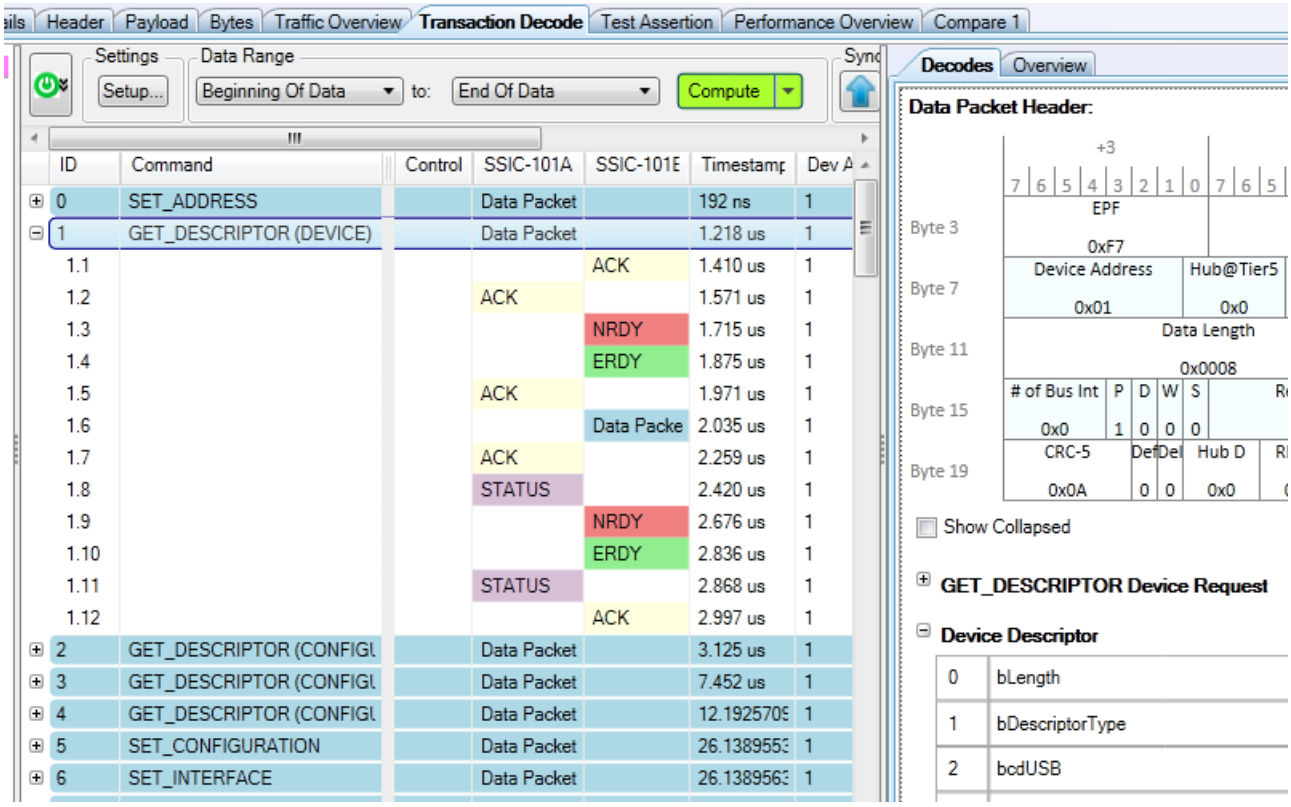
USB Standard Device Request Transactions - Example

In the following example, the decoded transaction for a **GET_DESCRIPTOR** request with the descriptor type set to Device is displayed. The first packet is a Data Packet sent by host to device to define the Device Descriptor request. After the subsequent ACKs and Endpoint Not Ready (NRDY) /Ready (ERDY) packets, the Device Descriptor is returned by the device in a Data Packet. The host then initiates the STATUS stage by sending the STATUS packet which is acknowledged by the device.

Notice that different colors have been used to represent different packet types in the transaction.

The Decodes tab on the right displays the:

- header for the first Data Packet of the transaction.
- the data for the packet defining the **GET_DESCRIPTOR** request.
- the data for the Device Descriptor returned by the device.

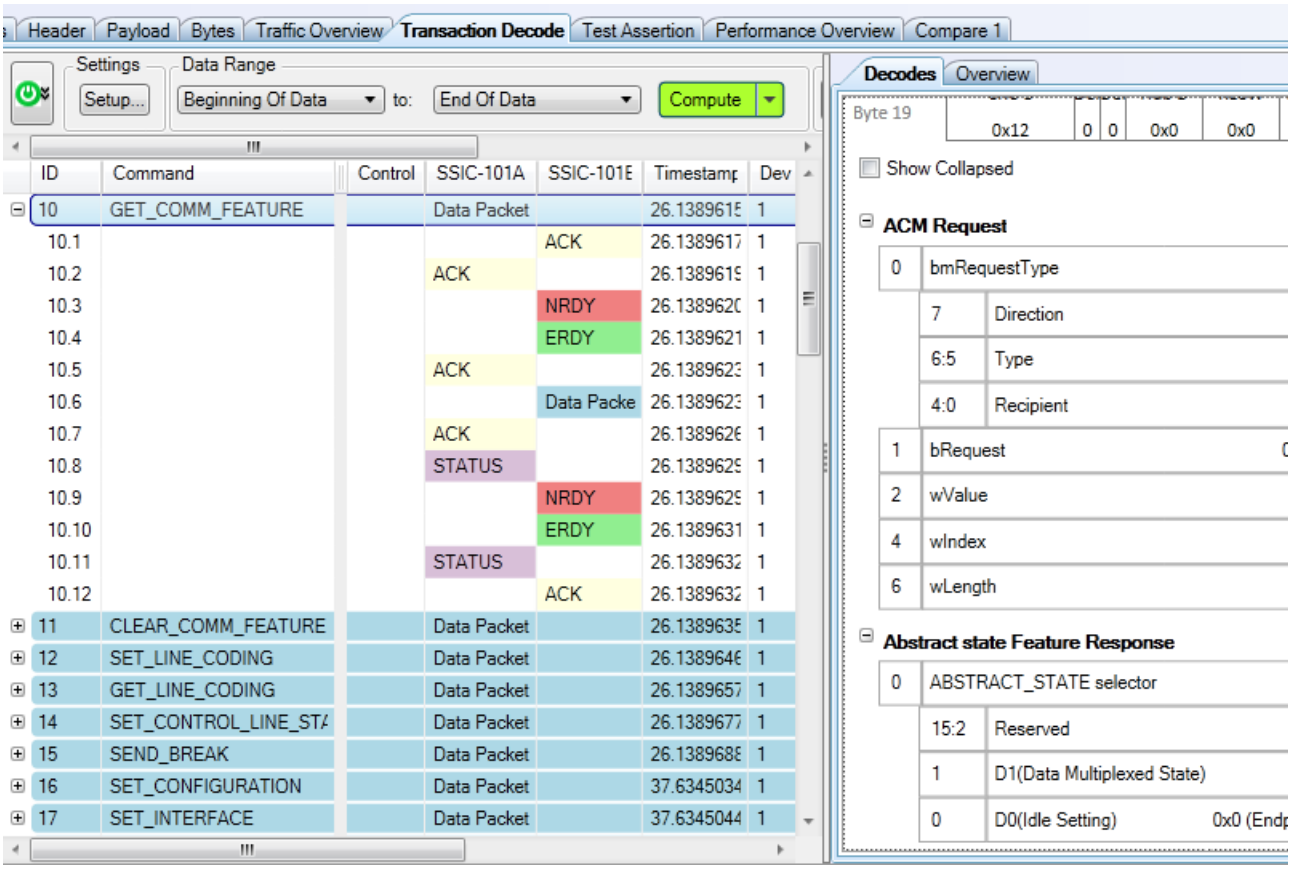


ACM Device Subclass Requests Transactions - Example

In the following example, the decoded transaction for a **GET_COMM_FEATURE** ACM request is displayed. The first packet is a Data Packet sent by host to device to get the current settings for the Abstract state communication feature. The device returns a Data Packet with the current settings for its Abstract state feature.

The Decodes tab on the right displays the:

- header for the first Data Packet of the transaction.
- the data for the packet defining the **GET_COMM_FEATURE** ACM request.
- the data for the Abstract state feature response returned by the device.

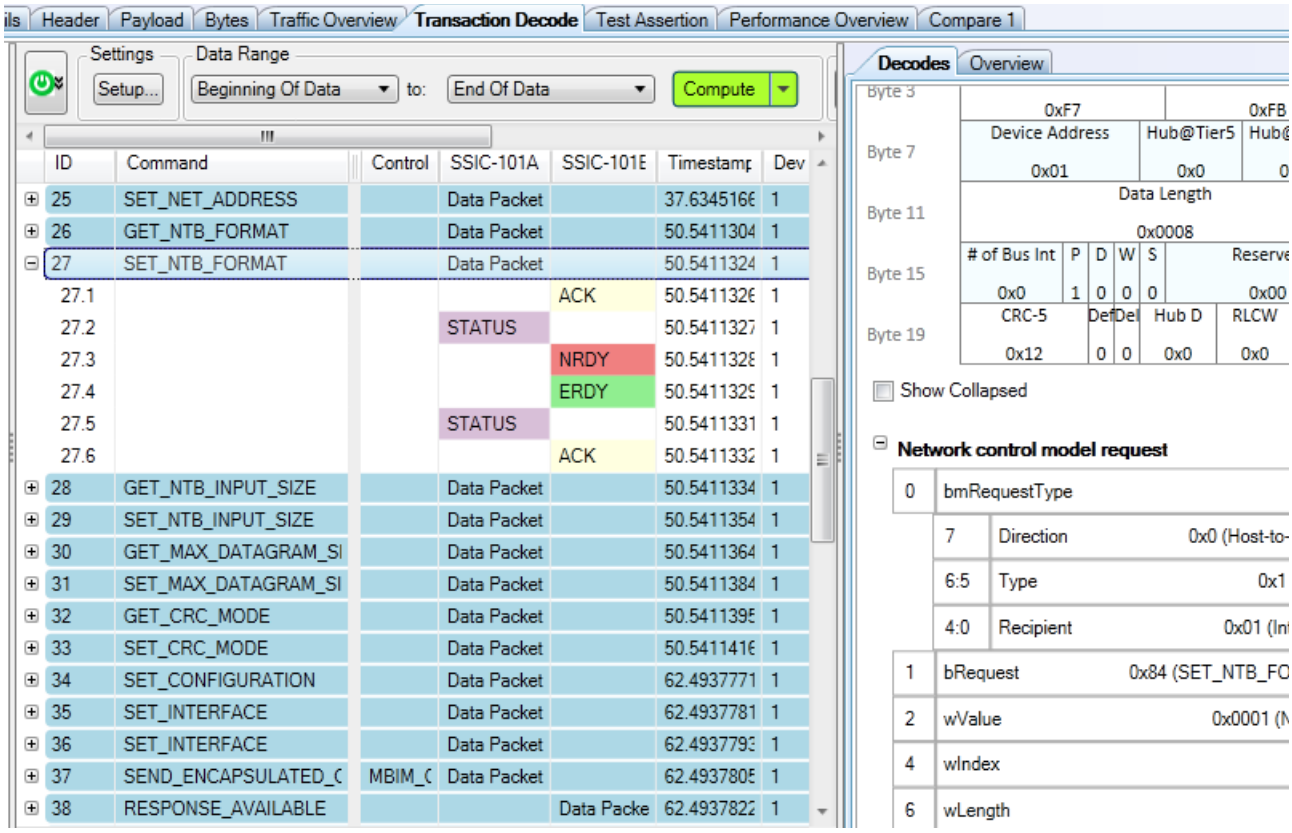


NCM Device Subclass Requests - Example

In the following example, the decoded transaction for a **SET_NTB_FORMAT** NCM request is displayed to set the device's Network Transfer Blocks (NTB) format to 32 bit.

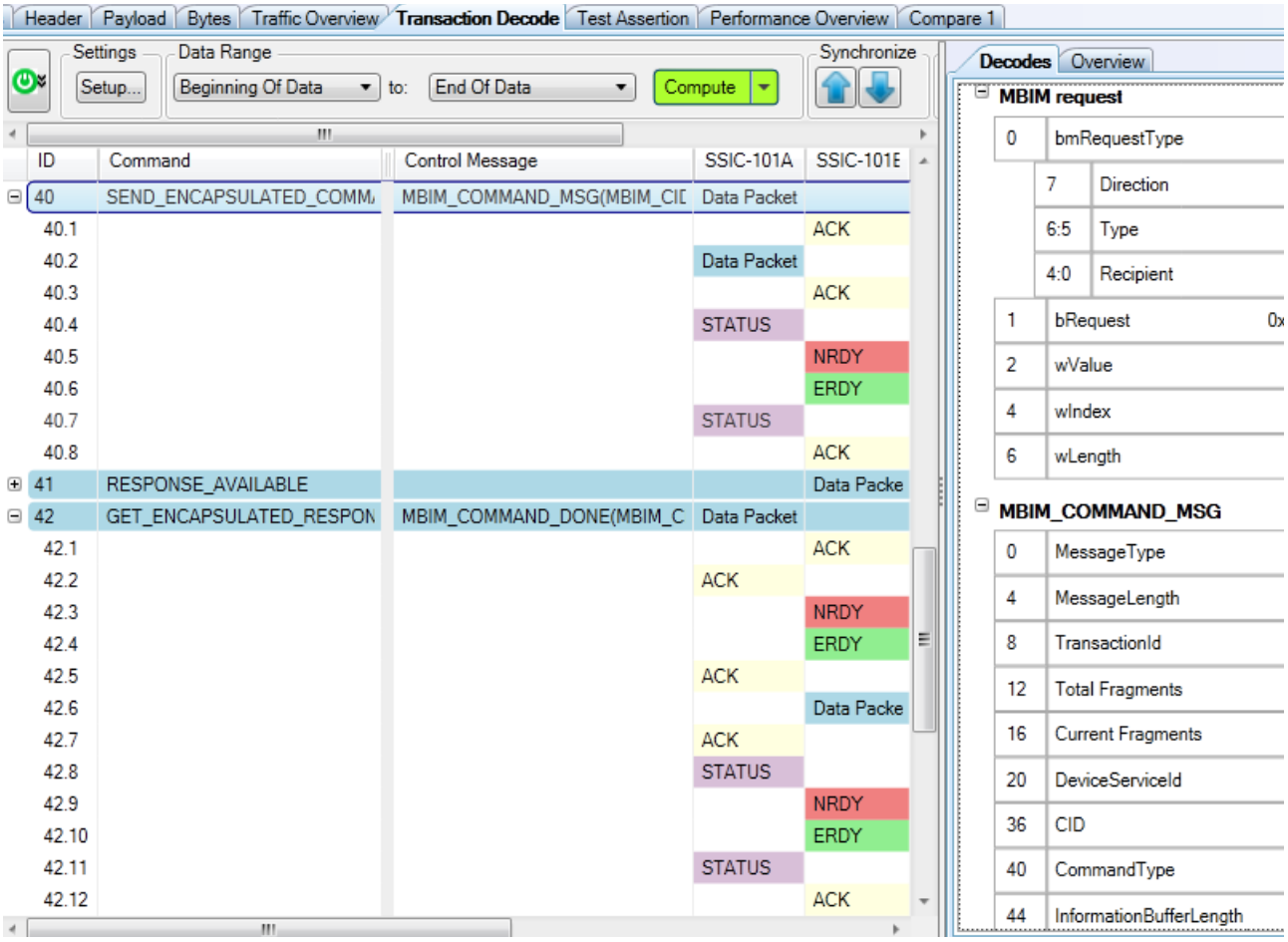
The Decodes tab on the right displays the:

- header for the first Data Packet of the transaction.
- the data for the packet setting the NTB format to 32 bit.



MBIM Device Subclass Requests - Example

In the following example, a sequence of decoded transactions for MBIM requests are displayed. The first transaction is a **SEND_ENCAPSULATED_COMMAND** request sent by host with an MBIM_COMMAND_MSG control message to request information associated to MBIM_CID_DEVICE_CAPS. The transaction is followed by a **RESPONSE_AVAILABLE** notification from device to host. The last transaction is a **GET_ENCAPSULATED_RESPONSE** transaction from the host to retrieve the MBIM_DEVICE_CAPS_INFO information in an MBIM_COMMAND_DONE control message.


Header
Payload
Bytes
Traffic Overview
Transaction Decode
Test Assertion
Performance Overview
Compare 1
Settings
Setup...
Data Range
Beginning Of Data
to:
End Of Data
Compute
Synchronize
Decodes
Overview
ID
Command
Control Message
SSIC-101A
SSIC-101E
40 SEND_ENCAPSULATED_COMM MBIM_COMMAND_MSG(MBIM_CIE Data Packet
40.1 ACK
40.2 Data Packet
40.3 ACK
40.4 STATUS
40.5 NRDY
40.6 ERDY
40.7 STATUS
40.8 ACK
41 RESPONSE_AVAILABLE Data Packe
42 GET_ENCAPSULATED_RESPON MBIM_COMMAND_DONE(MBIM_C Data Packet
42.1 ACK
42.2 ACK
42.3 NRDY
42.4 ERDY
42.5 ACK
42.6 Data Packe
42.7 ACK
42.8 STATUS
42.9 NRDY
42.10 ERDY
42.11 STATUS
42.12 ACK
MBIM request
0 bmRequestType
7 Direction
6:5 Type
4:0 Recipient
1 bRequest
2 wValue
4 wIndex
6 wLength
MBIM_COMMAND_MSG
0 MessageType
4 MessageLength
8 TransactionId
12 Total Fragments
16 Current Fragments
20 DeviceServiceId
36 CID
40 CommandType
44 InformationBufferLength

8 Computing and Running Test Assertions

Test cases are a set of predefined logical rules. Running the test cases on captured packets allows you to check whether or not in a trace the captured packets have been exchanged as per the defined rules. This chapter describes how you can run the test cases, and compute and assess the assertion results.

Test Assertion - Overview

Test Assertion is the process of running one or more test cases on the captured packets which allows you to check whether the captured packets have been exchanged as per the predefined rule. The **Test Assertion** tab in the Protocol Viewer window allows you to compute the test assertions for one or more traces.

Types of Protocols Supported

In this release, assertion of SSIC test cases is supported.

Test Assertion Tab

You use the **Test Assertion** tab displayed in the lower pane of the Protocol Viewer to view and run the test cases.

The **Test Assertion** tab consist of two panes. The left pane displays a list of test cases. When you select any test case, the sub-pane in the bottom of the left pane displays the details of the selected test case. The right pane displays the test summary of the computed test assertion(s).

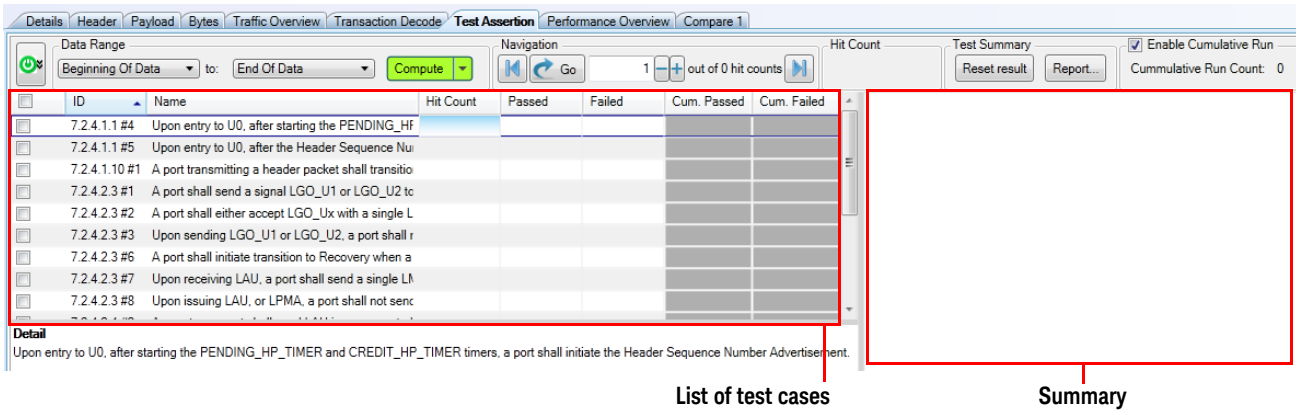


Table 1 lists the various test cases for this release.

Table 1 Test Cases

Sr. No.	ID	Description
1	7.2.4.1.1 #4	Upon entry to U0, after starting the PENDING_HP_TIMER and CREDIT_HP_TIMER timers, a port shall initiate the Header Sequence Number Advertisement
2	7.2.4.1.1 #5	Upon entry to U0, after the Header Sequence Number Advertisement, a port shall initiate the Rx Header Buffer Credit Advertisement
3	7.2.4.1.10 #1	A port transmitting a header packet shall transition to Recovery upon its PENDING_HP_TIMER timeout
4	7.2.4.2.3 #1	A port shall send a signal LGO_U1 or LGO_U2 to request a transition to a low power link state
5	7.2.4.2.3 #2	A port shall either accept LGO_Ux with a single LAU or shall reject LGO_U1 or LGO_U2 with a single LXU and remain in U0
6	7.2.4.2.3 #3	Upon sending LGO_U1 or LGO_U2, a port shall not send any packets until it has received LXU or re-entered U0

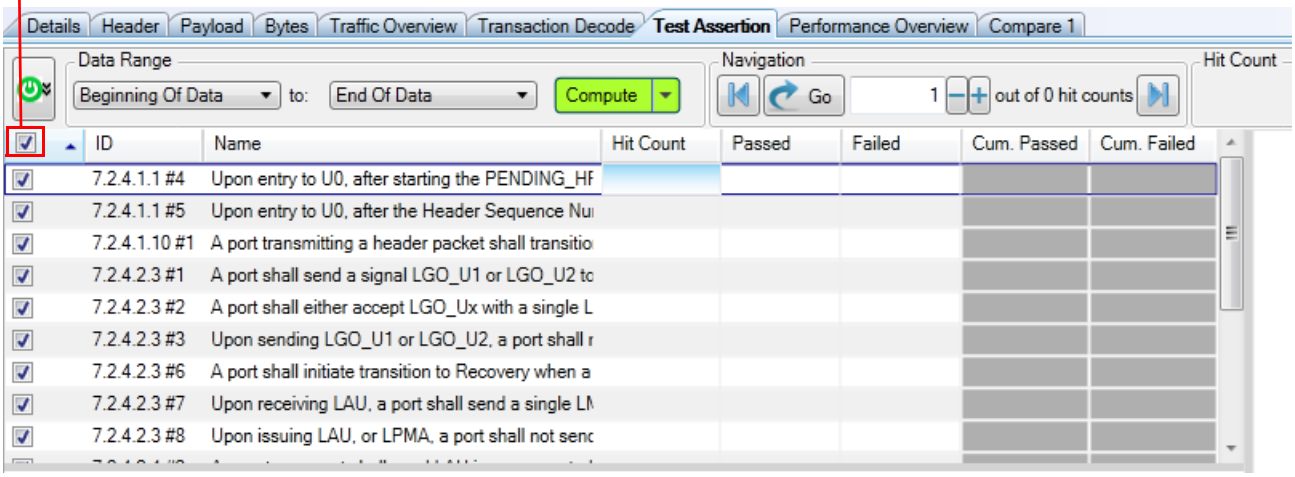
Sr. No.	ID	Description
7	7.2.4.2.3 #6	In U1/U2 entry flow, a port shall initiate transition to Recovery when a single LAU or LXU is not received upon PM_LC_TIMER timeout
8	7.2.4.2.3 #7	In U1/U2 entry flow, upon receiving LAU, a port shall send a single LPMA and then the requested low power link state should be entered
9	7.2.4.2.3 #8	In U1/U2 entry flow, upon issuing LAU, or LPMA, a port shall not send any packets or link commands
10	7.2.4.2.4 #2	In U3 entry flow, an upstream port shall send LAU in response to LGO_U3
11	7.2.4.2.4 #3	In U3 entry flow, an upstream port shall not send any packets or link commands subsequent to sending an LAU
12	7.2.4.2.4 #5	In U3 entry flow, a downstream port shall send a single LPMA and then transition to U3 when LAU is received
13	7.2.4.2.4 #7	In U3 entry flow, an upstream port shall transition to U3 when LPMA is received
14	7.3.4 #6	A port shall transition to Recovery upon its PM_LC_TIMER timeout
15	7.3.6 #1	A port shall transition to Recovery upon its PENDING_HP_TIMER timeout before the Header Sequence Number Advertisement is received
16	7.4.2#4	Upon completion of Hot Reset, the LTSSM of a port shall transition to U0
17	7.4.2#4	When a PORT_RESET is directed, when a Hot Reset fails due to a TS1/TS2 handshake timeout, a downstream port shall transition to Rx. Detect and attempt a Warm Reset

Computing the Test Assertions

1 Click the **Test Assertion** tab in the Protocol Viewer window.

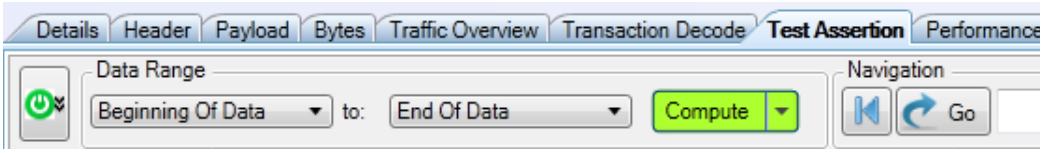
2 Select all or the individual test cases you want to assert by selecting the required checkboxes. For selecting all the test cases, you can select the checkbox as shown below.

Select this check box to select all the test cases



3 In the **Data Range** groupbox, specify the start and end points of the captured data for which you want to run the test cases. Following options are available for setting the data range.

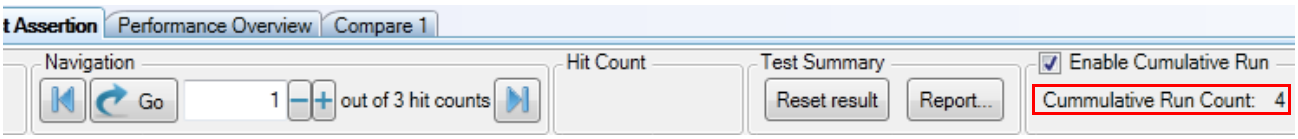
- **Beginning and End of data** - This data range selection ensures that test cases are run for the entire trace.
- **Trigger** - Selecting Trigger in the data range ensures that test cases are run from the point where the U4431 module's trigger condition was met.
- **Markers** - Selecting markers in the data range ensures that test cases are run for the specific portion of the traffic defined by markers. Refer to "Defining Markers for Setting the Computation Range" on page 145 to know more.



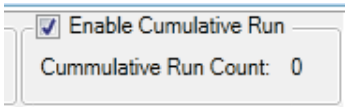
4 Click the **Compute** button displayed with the Data Range fields. The **Compute This** and **Compute All** are the two options available with this button.

- **Compute All** allows you to compute traffic overview statistics (see page 64), decoded transactions, the test assertions and offline performance summary (see page 156) for the captured packets by a single click of this button.
- **Compute This** allows you to compute only the test assertions. When you click **Compute**, then also only the test assertions are computed.

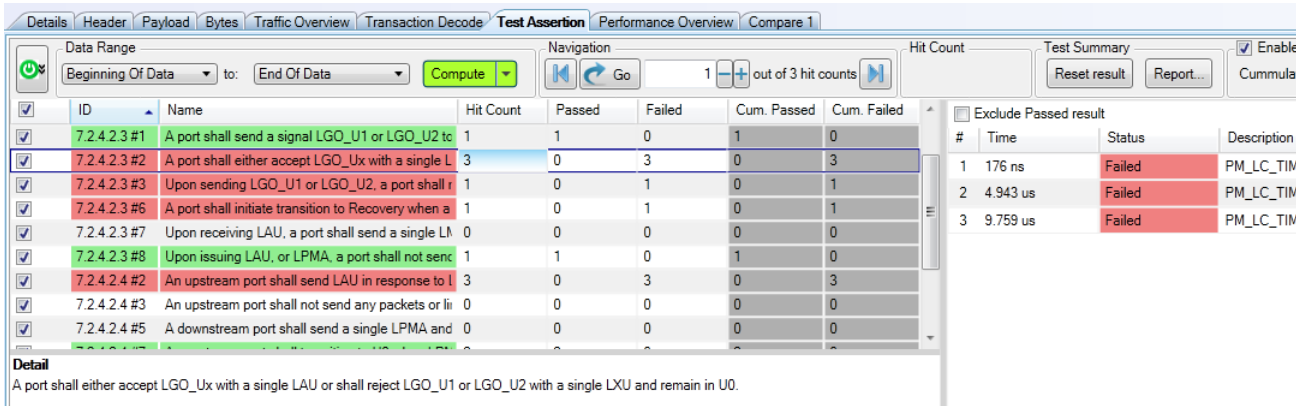
Cumulative Run Count shows the total number of compute operations performed for the trace.



NOTE The total number of compute operations are only counted when you have selected the **Enable Cumulative Run** check box. Selecting the **Enable Cumulative Run** check box also displays the **Cumulative Passed** and **Cumulative Failed** columns.



Test cases are computed and displayed for the specified data range.



NOTE You need to recompute the test assertion(s) if you want to change the data range for which test result is to be displayed.

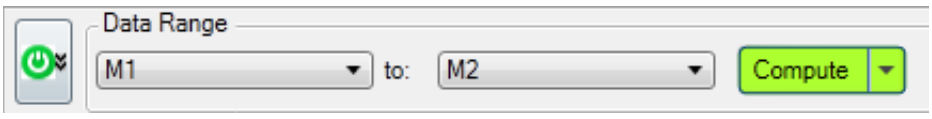
Defining Markers for Setting the Computation Range

If the captured traffic is too large and you want to view the test results from a specific portion of this traffic, then you can limit the computation range by defining start and end markers in the traffic.

To define markers

1. From the upper pane of the Protocol Viewer, right-click the row in the captured traffic that should act as the starting point for test assertion.
2. Select **Place Marker** from the displayed context menu and then select an existing marker or click **New Marker** to define a new marker at this point.

Once markers are defined, these are available for selection in the **Data Range** group box of the **Test Assertion** tab.



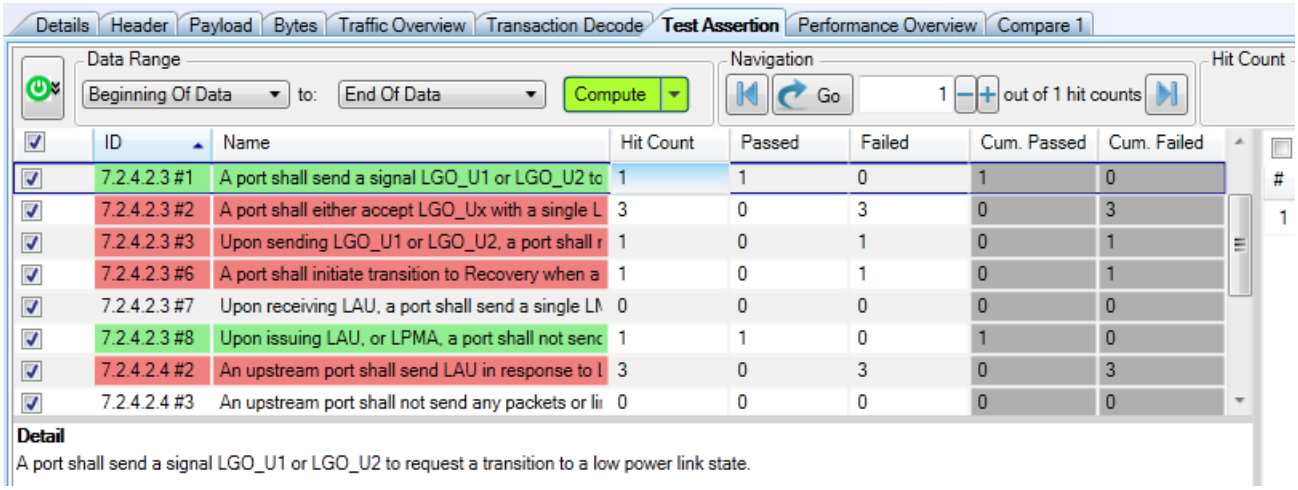
Viewing and Interpreting the Test Results

A test assertion is successful if the selected test case(s) passes all the encountered instances in the most recent compute operation. A test assertion has failed if the selected test case couldn't pass at least one encountered instance in the most recent compute operation.

After computing the selected test assertion(s), the test results are displayed in the **Test Assertion** tab.

Viewing the Test Cases Pane

In the left pane of the **Test Assertion** tab, test cases with their details are displayed. In this pane, a test case is uniquely identified by a reference ID. The description of the specific test case is also displayed. You can sort the test cases based on their IDs and descriptions.



A color coding scheme is used to clearly indicate the results of the test assertion. The background color of the selected test case:

- changes to green, if the test assertion is successful.
- changes to red, if the test assertion has failed.
- retains the default color, if the test case wasn't encountered at all in the most recent compute operation.

Based on the most recent compute operations, the results section also displays the following informations:

Column	Description
Hit Count	Displays the number of instances the selected test case was encountered in the most recent compute operation
Passed	Displays the number of instances the selected test case has passed the test assertion
Failed	Displays the number of instances the selected test case has failed the test assertion
Cumulative Passed	Displays the cumulative number of instances the selected test case has passed the test assertion over a number of compute operations
Cumulative Failed	Displays the cumulative number of instances the selected test case has failed the test assertion over a number of compute operations

Viewing the Test Summary

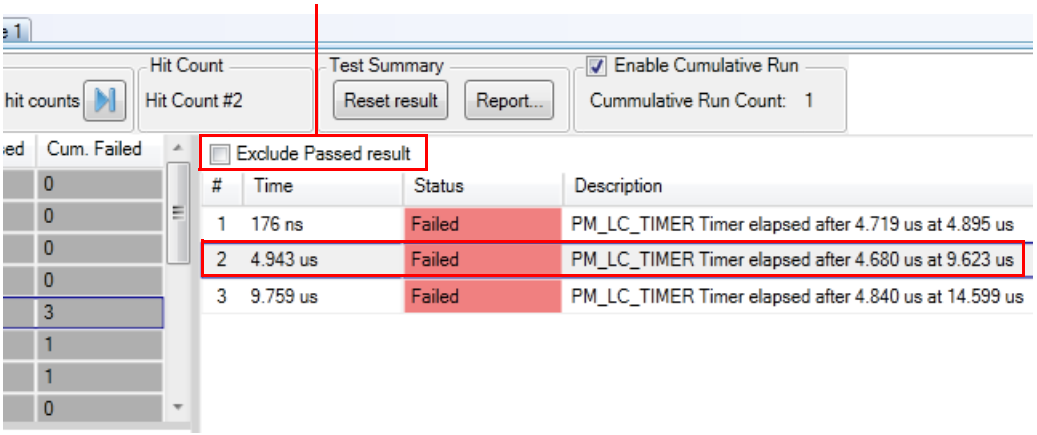
You can view the summary of the most recent test assertion or the collective summary of all the test assertions.

Viewing the Individual Test Results

You can click the individual test cases in the left pane of the **Test Assertion** tab for viewing the summary of the desired test assertions in the right pane of the **Test Assertion** tab. In the summary section, the detailed result of the specific test assertion is displayed.

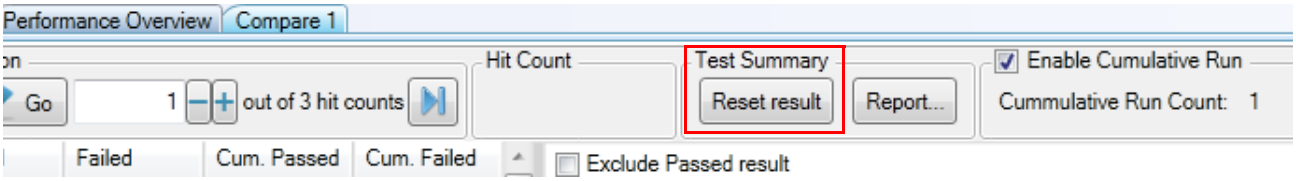
- The **Time** column displays the time of the first packet at which the selected test assertion started.
- The **Status** column displays whether the selected test case has passed or failed the desired assertion.
- The **Description** column displays the failure causes of the test assertion in that particular time interval. For example in the below screen, when the test case was encountered the second time, the test assertion failed at 4.943 us. The detailed description of the failure is also displayed.

Select this checkbox to view only the Failed assertions



You can view only the failed test results by clicking the **Exclude Passed result** checkbox as shown above.

If you want to clear the test results from the **Test Assertion** tab after a series of compute operations, click **Reset result** in the **Test Summary** section.

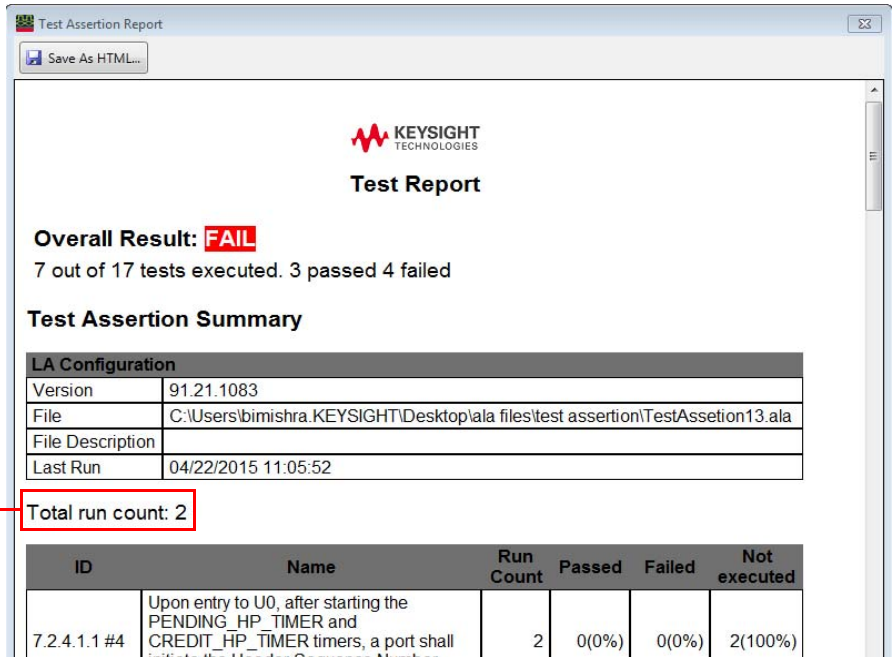


When you click **Reset result**, the **Cumulative Run Count** shows the default value “0”. You can then recompute the test cases for the most recent test result.

Viewing the Collective Summary

For viewing the test summary of all the test assertions over a number of compute operations, click **Report** in the **Test Summary** section.

The **Test Assertion Report** window is displayed.



No. of Compute Operations

The test report is a collective summary of all the individual test results displayed for a series of compute operations. In the test report,

- The **Overall Results** section displays the total number of instances the selected test cases were encountered over a series of compute operations. This section also displays whether or not the test assertion has passed over a series of compute operations.
- The **Test Assertion Summary** section summarizes the test results for a series of compute operations.

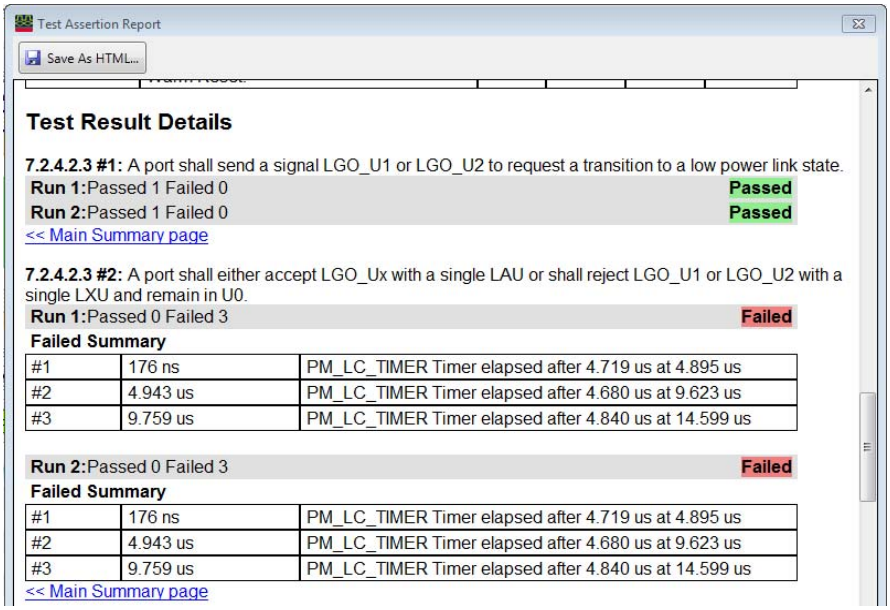
  For example in the above screen, seventeen test cases were tested twice in which seven test cases were encountered in the compute operation. Out of these seven test cases, four test cases failed the desired assertions. Since at least one test case has failed the assertion, the overall test assertion has failed.

  Overall Result: FAIL

- The **Test Result Details** shows the test result for the most recent compute operation or a collective summary of the individual test results over a series of compute operations.

NOTE You can view the detailed test report of the most recent compute operation without viewing the individual test results by deselecting the **Enable Cumulative Run** check box and then computing the required test cases.

For example in the below screen, the test assertion failed in both the instances when the test case having reference ID 7.2.4.2.3 #2 was encountered twice. You can view the individual failed summary reports for both the runs and assess the causes of failures.



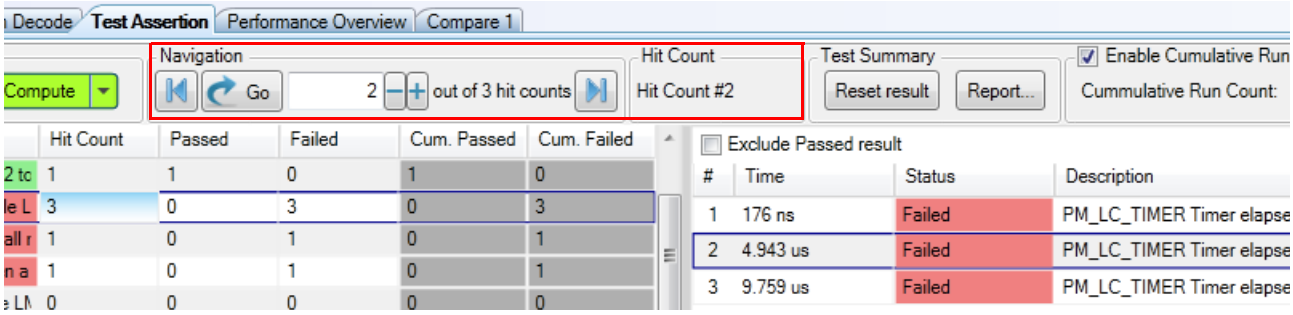
You can save the detailed result in HTML format on your computer by clicking **Save As HTML**.

Navigating Through Captured Packets from Test Results

You can navigate to the individual first captured packets which the selected test cases encountered when the test assertion started.

1 Click a test case on the left pane of the **Test Assertion** tab.

The navigation bar in the **Test Assertion** tab now displays the total number of instances the selected test case was encountered in the compute operation.

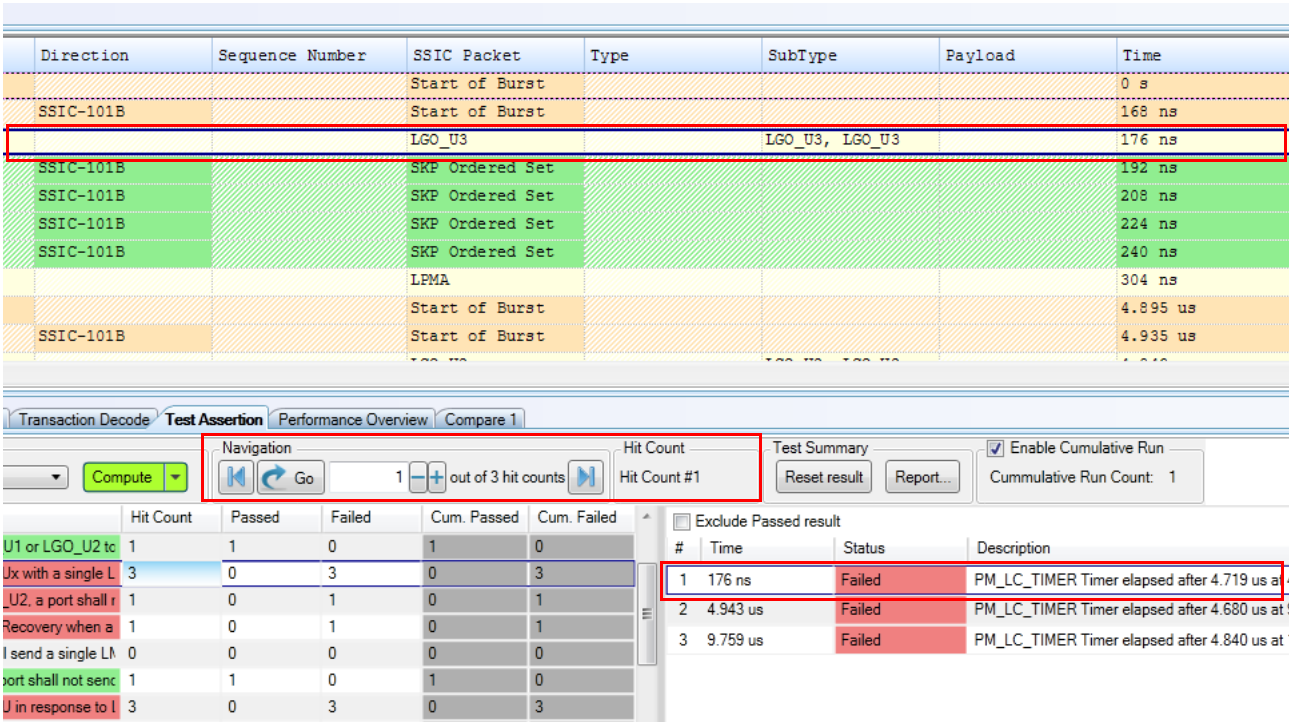


2 To view the first encountered instance of the selected test assertion in the computed data, click button in the navigation bar. Click to go to the last encountered instance of the selected test assertion.

3 To sequentially move through the encountered instances of the selected test assertion in the computed data, use the buttons in the navigation bar. The **Hit Count** section displays the current instance number of the individual test result.

4 To go to a specific encountered instance of the selected test assertion in the computed data, specify the hit count in the text box displayed in the navigation bar and click **Go**.

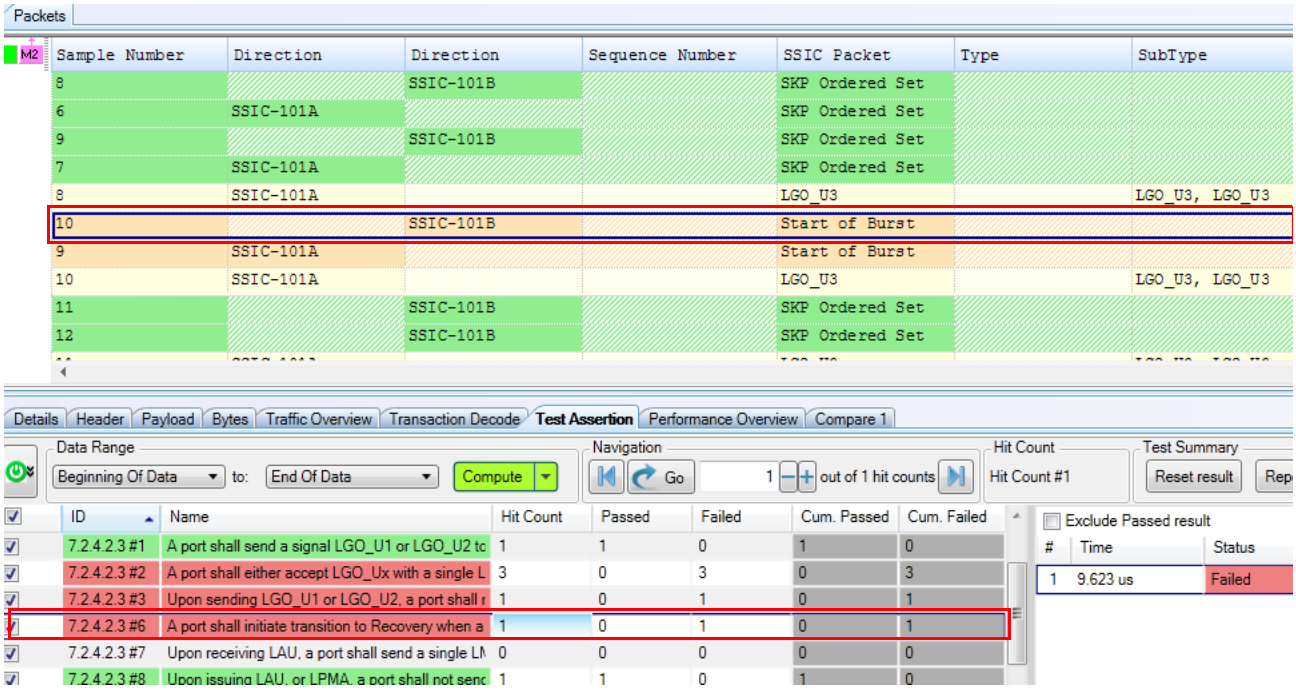
You can also double-click the required test result in the right pane of the **Test Assertion** tab to navigate to the first packet which the selected test case encountered in the Protocol Viewer in that particular time duration.



Navigating Between Test Cases and their Associated Packets

You can double-click a specific test case row in the left pane of the **Test Assertion** tab to navigate to the first packet which the particular test case encountered in the Protocol Viewer. Doing so, highlights the first captured packet encountered by the selected test case in the upper pane of the Protocol Viewer.

For example in the following screen, double-clicking the test case having reference ID 7.2.4.2.3 #6 highlights the Start of Burst packet in the Protocol Viewer, which is the first captured packet encountered by this test case.



9 Viewing Offline Performance Summary

In the Protocol Viewer window, you can generate and view the performance summary from the MPCIe and SSIC trace that you captured using the U4431A Analyzer module. This chapter describes how you can compute and view offline performance summary from the captured MPCIe/SSIC data.

Offline Performance Summary – Overview

The *Performance Overview* tab in the Protocol Viewer window allows you to perform post processing on the captured MPCIe/SSIC traffic to generate an offline performance summary of bus utilization. This tab presents statistics for various performance parameters in tabular as well as charts form.

In this tab, you define the range of captured data for which performance summary is to be generated. The software decodes transactions from this specified range of data and then computes performance statistics and charts from the decoded transactions. The specified range of trace data is sampled and statistics is computed from these samples to generate charts for each performance parameter.

Performance statistics are displayed separately for upstream and downstream link directions.

NOTE **You do not need connectivity to the U4431A hardware module to generate performance summary from a captured trace.**

Performance Overview Tab

You use the *Performance Overview* tab displayed in the lower pane of the Protocol Viewer to compute and view performance summary.

This tab displays the performance data in the following two panes.

Pane	Description
Performance Statistics pane	This is the left pane that displays a list of categories and the performance parameters for each of the categories for which statistics will be generated and displayed. For each of the parameters that you select, statistics are displayed based on the link directions (upstream as well as downstream).
Performance Charts pane	This is the right pane and displays a performance chart for selected performance parameters listed under the category.

NOTE If the Performance Overview tab is not visible, click the Pf button displayed at the top of the Protocol Viewer window.

The Overlay View - At a Glance

The information in the Performance Overview tab is presented using the Overlay view. The Overlay view allows you to select the series that you want to be displayed in the chart area. You can select the series by clicking the required performance parameter check box listed under Statistics tree/table on the left. This view displays graphs for the selected series in a single chart with a black background. It displays the Band Chart area where you can view the MSI Write, Msg Assert_INT and Msg Deassert_INT and error packets occurrence plotted in the chart. It also displays multiple Y-axis which you select in the **Setup** dialog box. To know more about how to select and interpret multiple Y-axis, see "Showing/Hiding multiple Y-axis" on page 158. Following screen displays the Overlay view after you select and compute a range:

In the above screen, graphs for the Update_FC_P series (represented by dotted lines) in both directions are displayed. The blue dotted line represents the graph for Update_FC_P series in upstream, whereas, the red dotted line represents the graph for Update_FC_P series in the downstream direction.

Configuring and Computing Offline Performance Summary

Before you Start

- Ensure that the data in the required direction(s) is already captured and available in the Logic and Protocol Analyzer GUI for performance summary computation. You may save the captured data in a Logic Analyzer configuration (.ala) file and access this data offline for performance summary computation.

Computing Offline Performance Summary

1 Click the **Performance Overview** tab in the Protocol Viewer window.

2 In the **Data Range** groupbox, specify the start and end points of the captured MPCIe/SSIC data for which you want to compute performance summary. Only the specified range of data is analyzed to compute performance. Following options are available for setting this data range.

- **Beginning and End of data** - This data range selection ensures that performance summary is computed for the entire trace.
- **Begin Extent and End Extent** - The Extent markers indicate the pan/zoom extents. The software automatically places extent markers on the beginning and end of the current pan/zoom extents defined in charts. When you change the pan/zoom extents in charts, the extent markers are automatically moved to changed extents.
- **Trigger** - Selecting Trigger in the data range ensures that performance summary is computed from the point where the U4431 module's trigger condition was met.
- **Markers** - Selecting markers in the data range ensures that performance summary is computed for the specific portion of MPCIe traffic defined by markers. Refer to "Defining Markers for Setting the Computation Range" on page 157 to know more.

3 For generating performance statistics, only complete transactions from the captured trace are used. Select the **Assume ACK** checkbox to instruct the software to assume ACKs for the transactions in which only ACK is missing. Assuming ACKs, therefore ensures that the transactions with missing ACK are considered complete and used in performance summary computation. You may want to use this Assume ACK option in situations such as when you have filtered ACKs while data capture to make more memory available to TLPs.

4 For producing the performance statistics, transaction of only error free packets from the captured trace are taken into account. The packets containing error are ignored while generating the performance statistics. Select the **Include Errors** check-box to let the software include error packets and compute the performance statistics including these packets. Selecting Include Errors checkbox increases the computation time of Performance Overview. It creates a series that is displayed in the Band Chart with an element for each packet containing error in the list.

5 Click the **Compute** button displayed with the Data Range fields. The **Compute This** and **Compute All** are the two options available with this button. **Compute All** allows you to compute traffic overview statistics, decoded transactions, and offline performance summary for the captured packets by a single click of this button. **Compute This** allows you to compute only offline performance statistics from the captured packets. When you click **Compute**, then also compute only offline performance statistics.

On clicking Compute, first the transactions are decoded from the specified data range of MPCIe trace. Then, statistics and charts are computed from these decoded transactions and results are displayed.

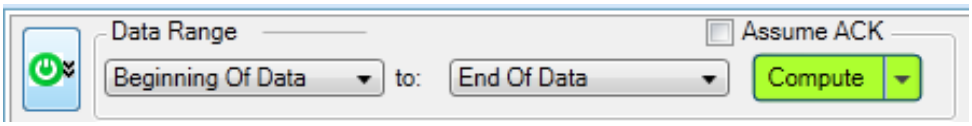
Defining Markers for Setting the Computation Range

If the captured MPCIe/SSIC traffic is too large and you want to view performance summary for a specific portion of this traffic, then you can limit the computation range by defining start and end markers in the traffic.

To define markers

1. From the upper pane of the Protocol Viewer, right-click the row in the captured traffic that should act as the starting point for performance summary computation.
2. Select **Place Marker** from the displayed context menu and then select an existing marker or click **New Marker** to define a new marker at this point.

Once markers are defined, these are available for selection in the **Data Range** group box of the **Performance Overview** tab.



Saving the Computed Performance Summary Data

Once you computed the performance summary data, you can save the performance configurations along with the captured traffic in a logic analyzer .ala configuration file. On saving, the settings such as zoom, pan, sample rate, markers, chart type, and chart order that you configured in the Performance Overview tab are also saved in the .ala file along with the trace data. Loading this .ala file retrieves these settings and computes and displays performance summary based on the saved settings.

To save the performance summary settings

1. Click **File** > **Save as**.
2. In the **Save As** dialog box, specify the name of the file.
3. Ensure that the **Standard Configuration (*.ala)** option is selected as the file type and **All Data and Setup** is selected in the **File Options** group box.
4. Click **Save**.

To access and view previously saved performance summary settings

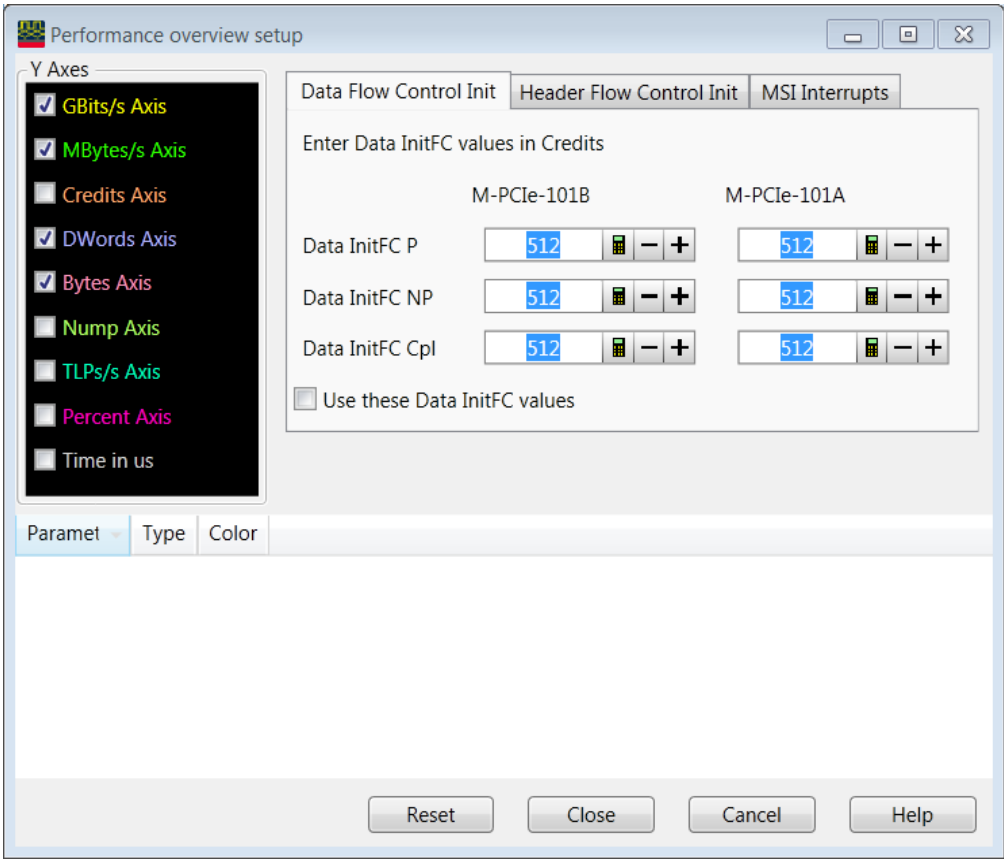
1. Click **File** > **Open**.
2. In the **Open** dialog box, navigate to the **Standard Configuration (*.ala)** file in which you saved the data.
3. Click **Open**.

Defining Chart Settings

You can define settings for the overlay chart in the **Setup...** tab of the **Settings** section and change the appearance of the displayed chart.

To access the Setup dialog box

Click the **Setup...** button under the **Settings** section. The Setup dialog box appears as shown below.

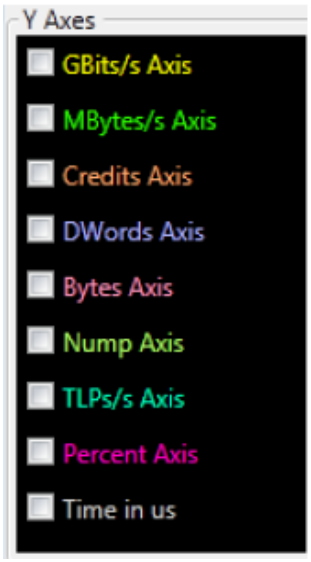


The setup details that you specify in this dialog box helps you change the way series and axes are displayed in the overlay graph. For example- you can change the graph display type and color of the displayed series or you can show or hide the Y-axis by selecting or de-selecting the respective Y-axis checkbox.

The following is the list of tasks that you can do using the **Setup** dialog box:

Showing/Hiding multiple Y-axis

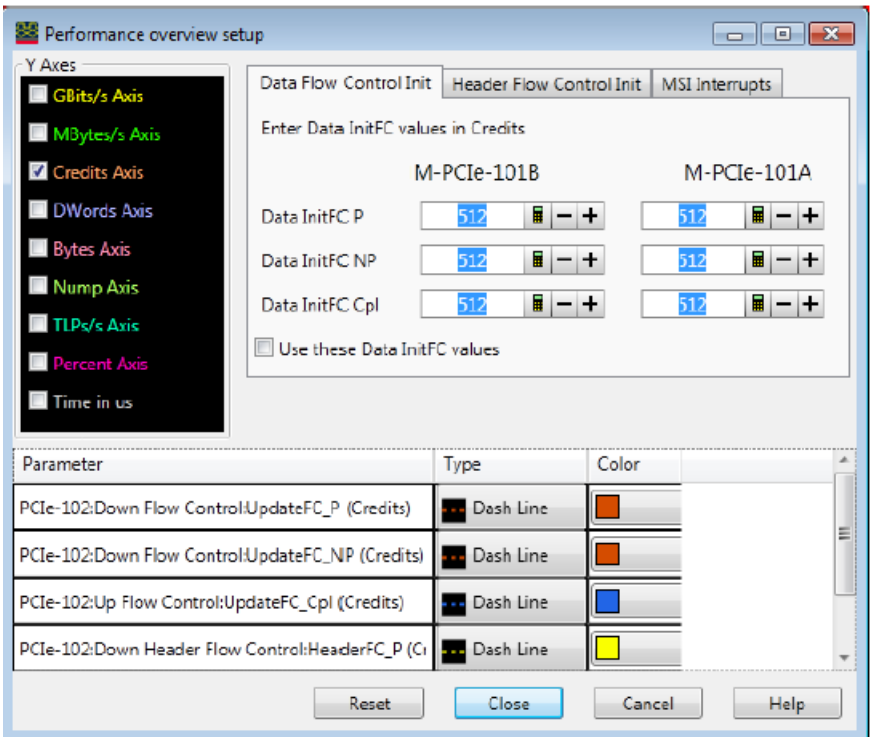
Select or de-select the Y-axis checkbox to show or hide the Y-axis displayed in the chart.



NOTE Showing or hiding a particular Y-axis does not affect how series are computed or displayed in the chart.

Specifying InitFC values

You can initially define the maximum flow control credit limits by entering InitFC values for posted, non-posted, and completion transactions in both directions (Upstream and Downstream). You can do this separately for Data and Header flow control Init values.



Click button displayed with the InitFC P, InitFC NP, and InitFC Cpl fields to open the calculator and input values. Select the checkbox '**Use these InitFC values**' to use the InitFC values defined by you when you are computing and interpreting offline performance summary. In case InitFC values are not defined or '**Use these InitFC values**' field is not selected, the minimum and maximum credit levels for the captured trace is automatically computed and flow control mechanism works on the basis of those calculated values. The minimum and maximum credit levels varies from one trace to other depending upon the size of the trace.

NOTE InitFC values are applicable only for Flow Control performance statistics category. You can use InitFC values for posted, non-posted and completions transactions only when you are computing flow control credits.

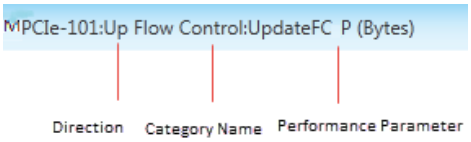
Specifying Min/Max value for MSI and MSI-X Address

You can specify the minimum and maximum values for the Message Signaled Interrupt (MSI/MSI-X) address in the **MSI and MSI-X Address** row for which you want Interrupt to be generated and plotted in the Band Chart.

Selecting this checkbox will generate interrupts for the specified address which you can view in the Band Chart.

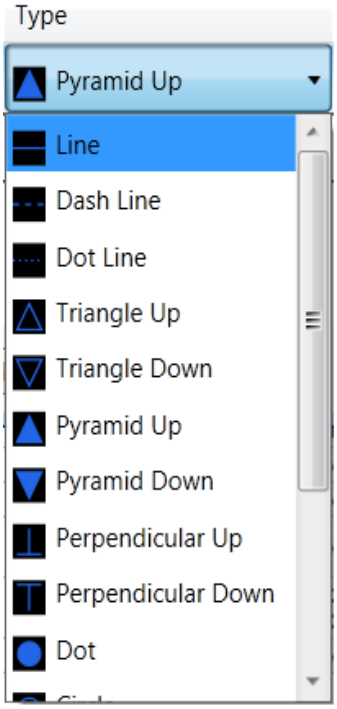
Understanding Parameter Column

The series which you select by clicking their respective check-box in the Statistics Tree/Table of the Performance Overview Tab is displayed here in the **Parameter** column of the **Setup** dialog box. The Parameter is the full name of the series and is comprised of the Direction (Up or Down), the Category name picked from the Performance Statistics tree/table, and the checked Series name.



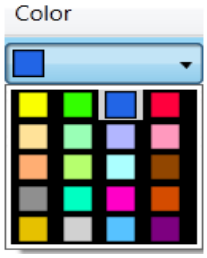
Changing the appearance of the graph

You can change the appearance of the graph representing the respective series listed in the **Parameter** column of the **Setup** dialog box. To do so, click the drop-down in the **Type** column and select the graph type you want to change with.



Changing the color of the Graph Line

You can also change the color of the graph line by picking color of your choice from the color palette. To do so, click the **Color** drop-down to display the color combo box consisting of different colors, and select the color of your choice.



- Once all the settings are defined in the Setup dialog box, click the **Close** button to apply those settings and exit from the dialog box.
- Click the **Reset** button to restore the last settings done previously when you opened the Setup dialog box. This resets the Axis, Types and Colors to their previous state.
- Clicking the **Cancel** button allows you to exit from the dialog box without making any changes.

Interpreting the Performance Summary Results

Performance Statistics panes

The **Performance Statistics** pane displays a hierarchical list in which various performance statistics are categorized in groups and sub-groups.

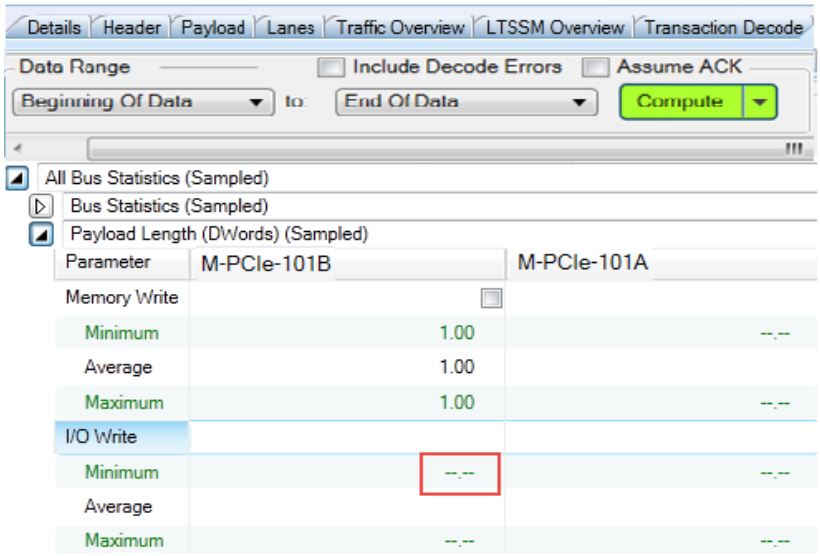
Clicking a group or category in this pane displays the link-wise statistics for associated performance parameters. The following table briefly describes the performance statistics displayed for categories

.

Category	Performance Statistics Displayed
Flow Control	Tracks and computes the available flow control credits for a bidirectional data trace and helps in analyzing the flow of data. To know more refer to the topic "Flow Control.
Bus Statistics	▪ **Bandwidth (GBits/s):** Shows the number of non- idle symbol bits transferred per second. ▪ **Data Throughput (MBytes/s)**: Shows the number of TLP payload bytes transferred per second. ▪ **Payload Length (DWords):** Shows the minimum, maximum, as well as average payload size for TLPs. ▪ **TLP Count (TLPs/s)**: Shows the number of TLPs transferred per second. ▪ **Link Efficiency (%):** Shows the efficiency of the link. It is calculated as Symbol time for payload / (Symbol time for all DLLP + Symbol time for all TLP + Symbol time for OS). ▪ **Link Utilization (%)**: Shows the percentage of non- idle symbols in total number of symbols transferred. ▪ **TLP Utilization (%):** Shows the percentage of TLP symbols in total number of symbols transferred.
Transaction Performance	▪ **Non- Posted Transactions**: Shows the minimum, maximum, and average values of the following performance parameters for all non- posted transactions. ▪ **Response Time (ns)**: It is same as the duration of the transaction. ▪ **Latency (us)**: It is the time duration between the end of a request transaction and the arrival of its first completion. ▪ **Throughput (MBytes/s)**: It is the length of complete data divided by the response time. ▪ **Posted Transactions**: Shows the minimum, maximum, and average values of the following performance parameters for all posted transactions. ▪ **Response Time (ns)**: It is same as the duration of the transaction. ▪ **Throughput (MBytes/s)**: It is the length of complete data divided by the response time.

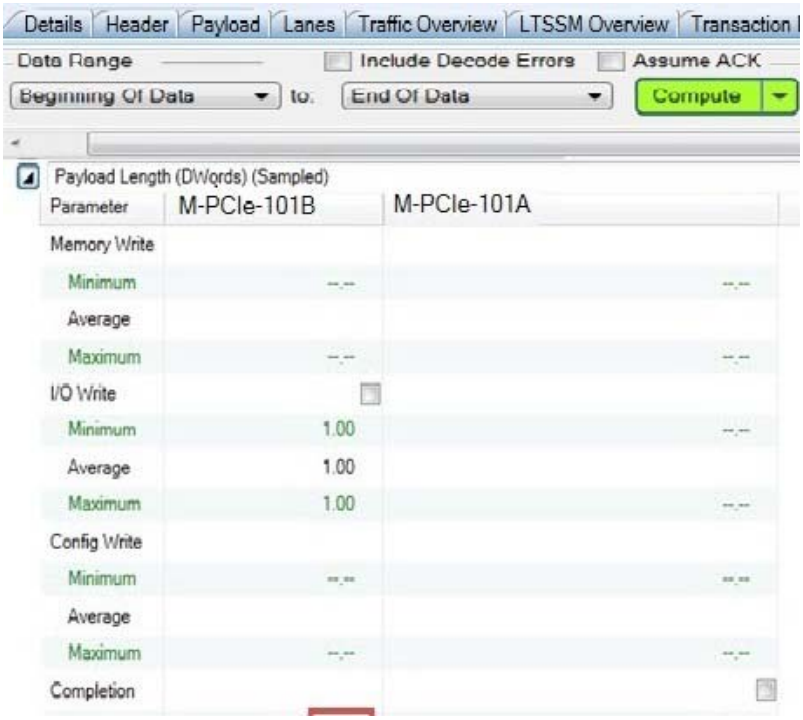
Some important points while viewing and interpreting performance summary:

- A --.-- value displayed in a statistics tree/table for a performance parameter indicates that the packet(s) required to generate that statistics is not found in the trace. It signifies 'no data' received for generating the statistics. Consequently, the chart corresponding to such a performance parameter contains no data and is therefore not displayed in the charts pane. Such a chart gets displayed only when the required packets are found while computing performance summary to generate data for the chart.



- For calculating the statistics for the **Completion** parameter, only *Completions with Data* packets are included. Any *Completions without Data packets* found in the trace are ignored for this parameter's statistics calculations.

  For instance, in the following screen, the *Completion with Data* packets are used for calculating the *minimum*, *maximum*, and *average* payload size for Completions. In the absence of *Completion with Data* packets in the trace, the *Completion* statistics is shown --.--.



- Some performance parameters are displayed in blue color. This indicates that the navigation to the associated MPCIe packet is applicable for that performance parameter. To know more, refer to "From the Performance Statistics pane" on page 173.

Parameter	M-PCIe-101B	M-PCIe-101A
UpdateFC P (Bytes)	☑	☑
Minimum	0.00	0.00
Average	4444.51	4571.28
Maximum	7712.00	7272.00

Charts pane

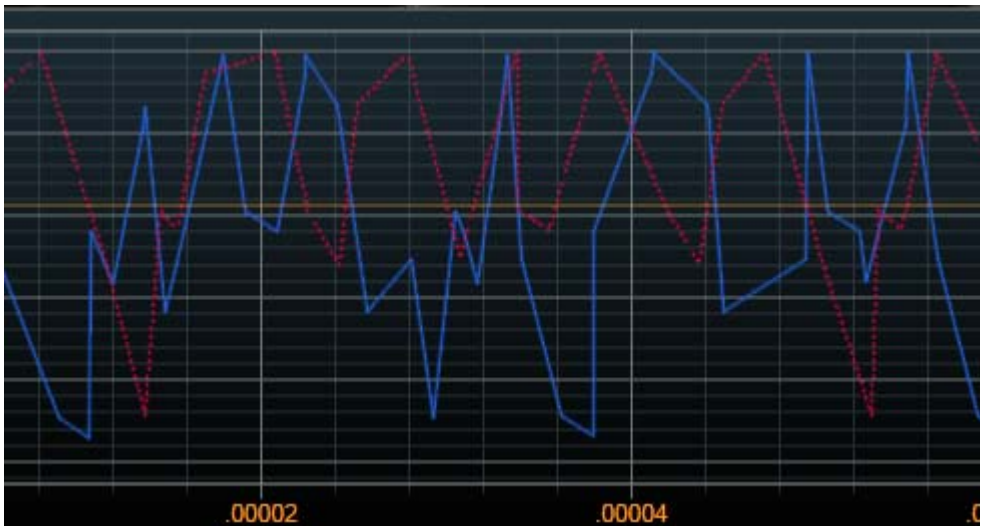
This pane displays charts for all the selected performance parameters listed in the statistics tree table.

Viewing Multiple Y-axis in the Chart Pane

The chart pane displays multiple Y-axis if you select multiple Y-axis checkbox in the **Setup...** dialog box. To know more, see "Showing/Hiding multiple Y-axis" on page 158.

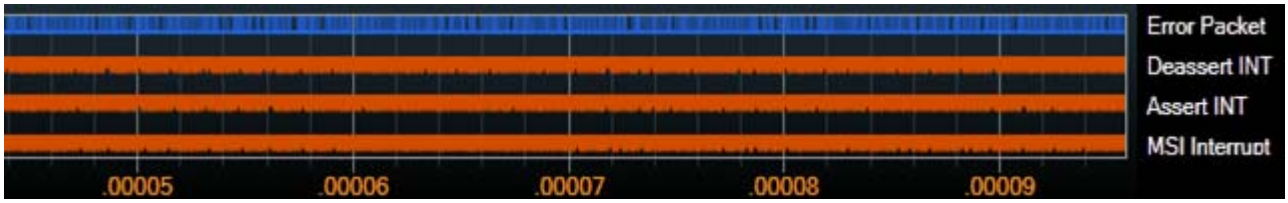
Viewing Overlay Charts and defining Color code for graphs

The Chart pane displays Overlay chart for selected performance parameters listed in the categories pane. You can define different color codes to display the performance data for downstream and upstream directions in charts. To accomplish this, click the **Setup...** button under the **Settings** section. Then, click the **Color** drop-down to select the color from the color palette. To know more, refer "Changing the color of the Graph Line" on page 161.



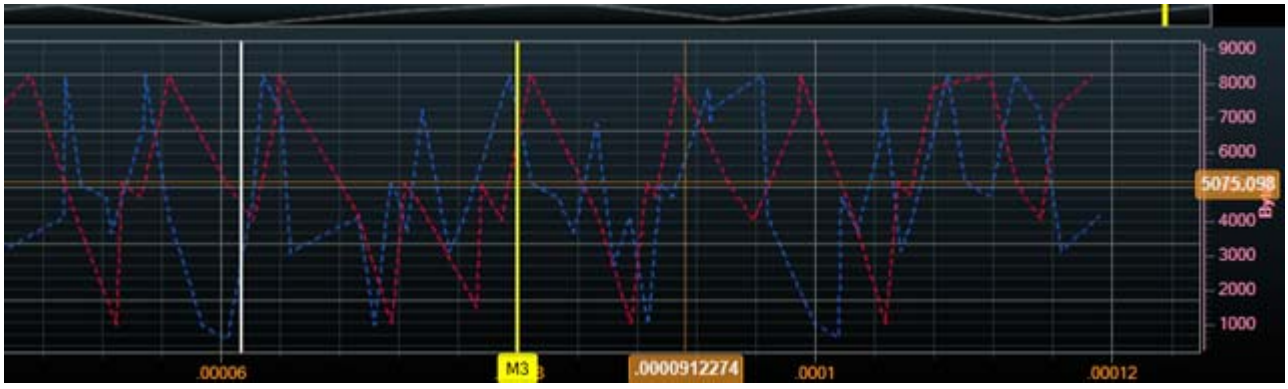
Viewing Band Chart

The lower section of the Chart pane displays the Band Chart. You can view MSI Write, Msg Assert_INT and Msg Deassert_INT and error packets in the Band chart area of the Chart pane.



Viewing X and Y axes values in a chart

Hovering the mouse over a chart location displays the applicable values of X-axis and Y-axis for that location.



Flow Control

Flow Control computes and displays the available flow control credits from the data trace that has a bidirectional traffic. It presents flow control statistics and graphs and helps in analyzing the flow of data. Flow control credit level is determined by the Writes and Completions in one direction and the UpdateFC packets in the other direction. The data series created for the flow control charts contain a point for every packet that is a part of that series. This is in contrast to all other charts which represent a group of packets in a sample period. Flow control credit levels are represented by UpdateFC series. The performance parameters for the Flow Control are explained below:

- **UpdateFC_P (Bytes)** - Shows the minimum, maximum, as well as average flow control credit levels for posted transactions.
- **UpdateFC_NP (Bytes)** - Shows the minimum, maximum, as well as average flow control credit levels for non-posted transactions.
- **UpdateFC_Cpl (Bytes)** - Shows the minimum, maximum, as well as average flow control credit levels for Completions data.
- **Memory Read (Bytes)** - Charts the size of Memory Read requests in each direction (X and Y axes).
- **Memory Write (Bytes)** - Maps the size of Memory Write and MsgD packets in each direction.
- **Memory_CplD (Bytes)** - Charts the size of Memory completion with data packets in each direction.
- **Inst. CplD and Inst. Write (Mbytes/s)** - The instantaneous completion with data (Inst. CplD) and instantaneous write (Inst. Write) show Instantaneous Bandwidth in each direction. Instantaneous Bandwidth is defined as the number of data bytes (in MegaBytes) in the packet divided by the time (in seconds) since the end of the previous data carrying packet to the end time of the current data carrying packet.

  Each packet carrying data is represented by a point in these series.
- **Interrupts**- Shows the location in the graph where MSI Write, Msg Assert_INT and Msg Deassert_INT occurred. This is displayed in the Band chart area of the chart pane.
- **Error Packet**- Shows the location in the graph where packets with any error occurred. This is displayed in the Band chart only when the Include Errors checkbox is enabled under Data Range heading.

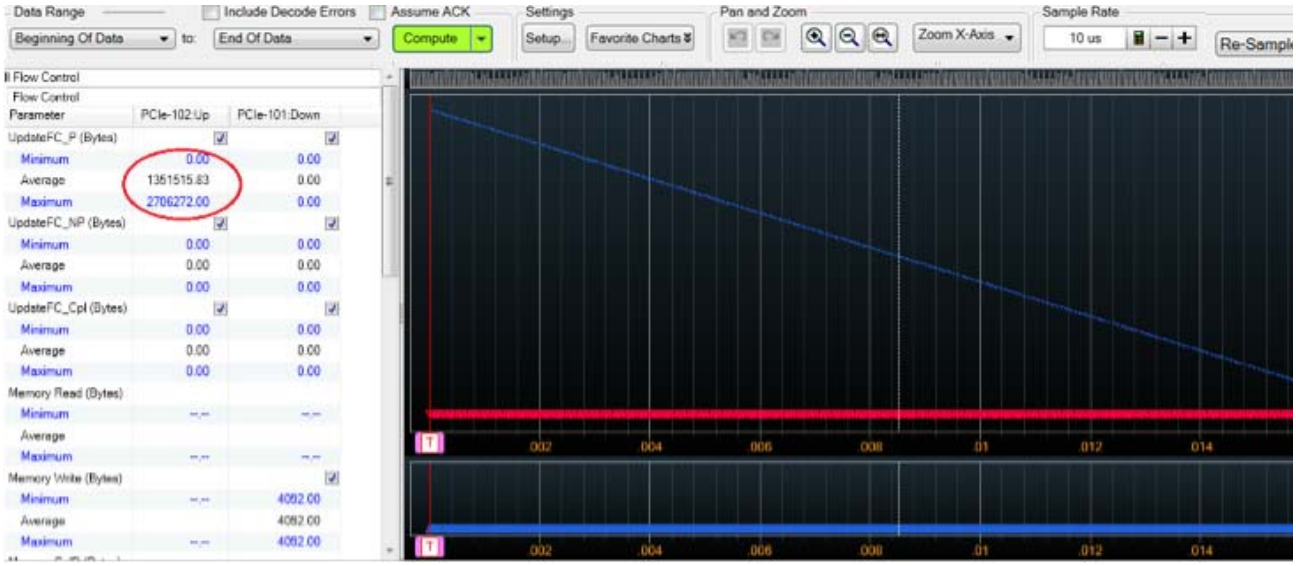
Flow Control- Examples

The following examples elaborate the usage of Flow Control feature:

Example-1

The flow control charts represent the computed flow control credit level, which is meant for how many bytes the receiver is ready to receive. When the data is sent in a higher volume than the receiving capacity of the receiver, there will be a continuous slow movement of data from transmitter to receiver. This sometimes also leads to data loss. In such scenarios, you can use the Flow Control feature to display the credit limits of the data to be sent and the data received by the receiver.

In this example, 4092B (1023 DW) packets of data are being sent in the downstream direction and the link partner is updating credits in increments of 4096B. In the following screen, a steep decline in the blue line clearly indicates the situation that FC credit is being updated at higher rate than the rate at which data is being posted. The blue line represents the computed UpdateFC_P regulating against the red line representing the corresponding Memory Write packets in the trace.

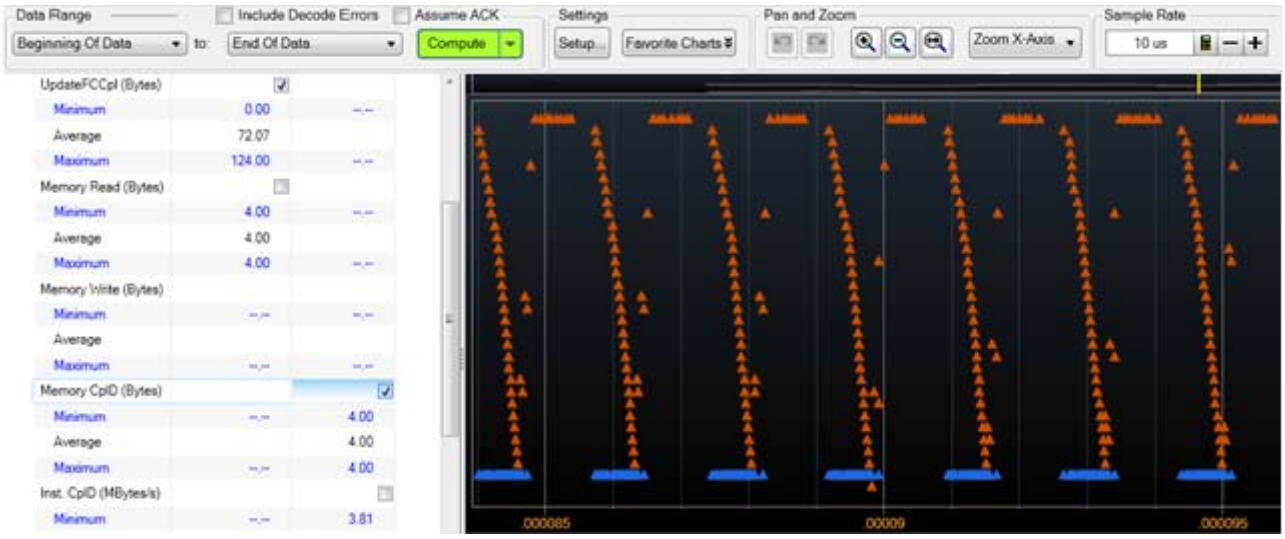


The UpdateFC_P Flow Control credit level is reduced by Memory Write packets coming from the root occurred in the trace. As displayed in the above graph, the Maximum is near the beginning of the trace and the Flow Control credit level continuously drops down and reaches zero on the right side of the trace.

This too many credits difference may indicate problem in the credit update behavior of the DUT.

Example-2

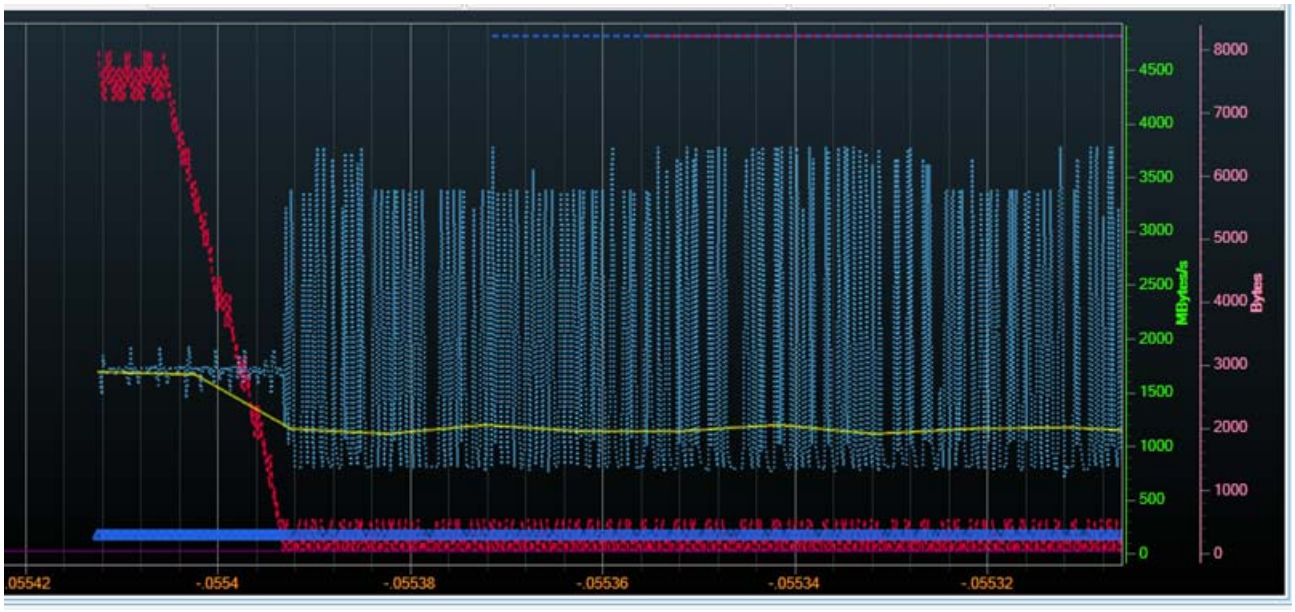
In the following screen, when memory completion data (Memory CplD) is being sent, the UpdateFC_Cpl performance parameter computes and displays minimum, maximum, as well as average flow control credit levels for Completions data. The blue and orange Pyramid Up chart line represents Memory CplD and UpdateFC_Cpl respectively. The UpdateFC_Cpl credit starts from an initial counter value and keeps on updating its credit limit till all the packets are sent and received.



As displayed in the above graph, the UpdateFC_Cpl graph line keeps showing the credit fluctuation regulating against the corresponding Memory Completion packets coming from root that occurred in the trace. The drop in credit level shows how many packets are left to be transmitted and helps in regulating the flow of data packets. When all packets are transmitted and received, the FC level changes to zero counter value. It remains constant till any further credits are added into account. Once the FC level is incremented with some value, the same mechanism is repeated till all the packets are transmitted and received and FC level again comes to a zero value.

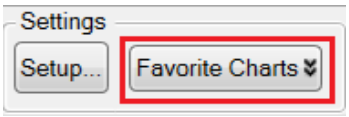
Example-3

In the start of the DMA buffer transfer, the link credits are quickly exhausted and reaches to zero counter value because the buffer transfer runs at a higher rate in comparison to the rest of the data transfer. The quick drop in credit level is caused by the chain of link buffering. In the start, the data write or completion capacity remains higher than the sink capacity but lower than the link capacity. The data transfer at higher rate only happens while the link buffering capacity can sustain it.



Saving the Favorite Overlay Chart

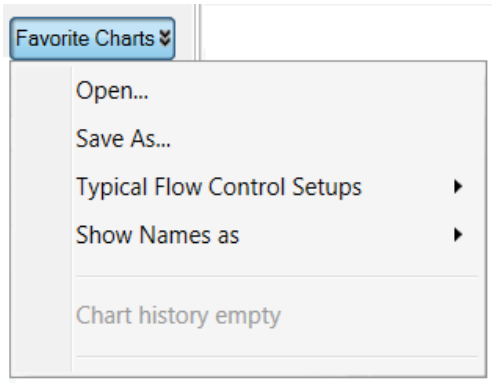
In the Performance Overview tab, the **Settings** section has a **Favorite Charts** button which allows you to save your favorite charts and open the previously saved overlay charts.



The overlay chart is saved along with all the settings that you made to the graph displayed in the chart pane of performance tab (such as selecting or deselecting various performance parameters in the Categories pane, zooming or panning the chart, making changes in the Setup dialog box etc.).

To save the Overlay Chart

1 Click **Favorite Charts** button> **Save as**.



2 In the Save As dialog box, specify the name of the chart. Ensure that ***.cfv** (Chart Favorite) option is selected as the file type.

3 Click **Save**.

To view previously saved Overlay chart

1 Click **Favorite Charts** button> **Open**.

2 In the Open dialog box, navigate to the *.cfv file name by which you saved the Overlay chart.

3 Click **Open**.

Navigating Through the Performance Summary Results

Navigating Through a Chart

By Using Pan option

To navigate through a chart horizontally, that is X-axis, click the **Pan X-Axis** button displayed at the top of the charts pane.

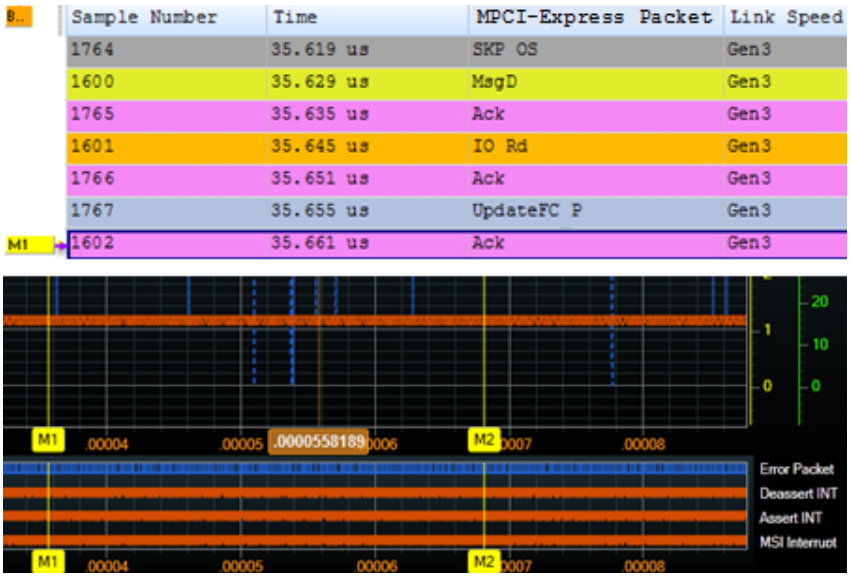
To navigate through a chart horizontally and vertically, that is both axis, click the **Pan Both Axis** button displayed at the top of the Charts pane. You can then drag the chart up, down, left, and right.

Navigating Between Performance Statistics and Associated MPCIe Data

By placing markers in charts

You can place markers in charts and use these markers to navigate to the MPCIe packet associated with the chart location at which you placed a marker. Markers placed in charts are correlated to markers displayed in the trace data in the upper pane of the Protocol Viewer.

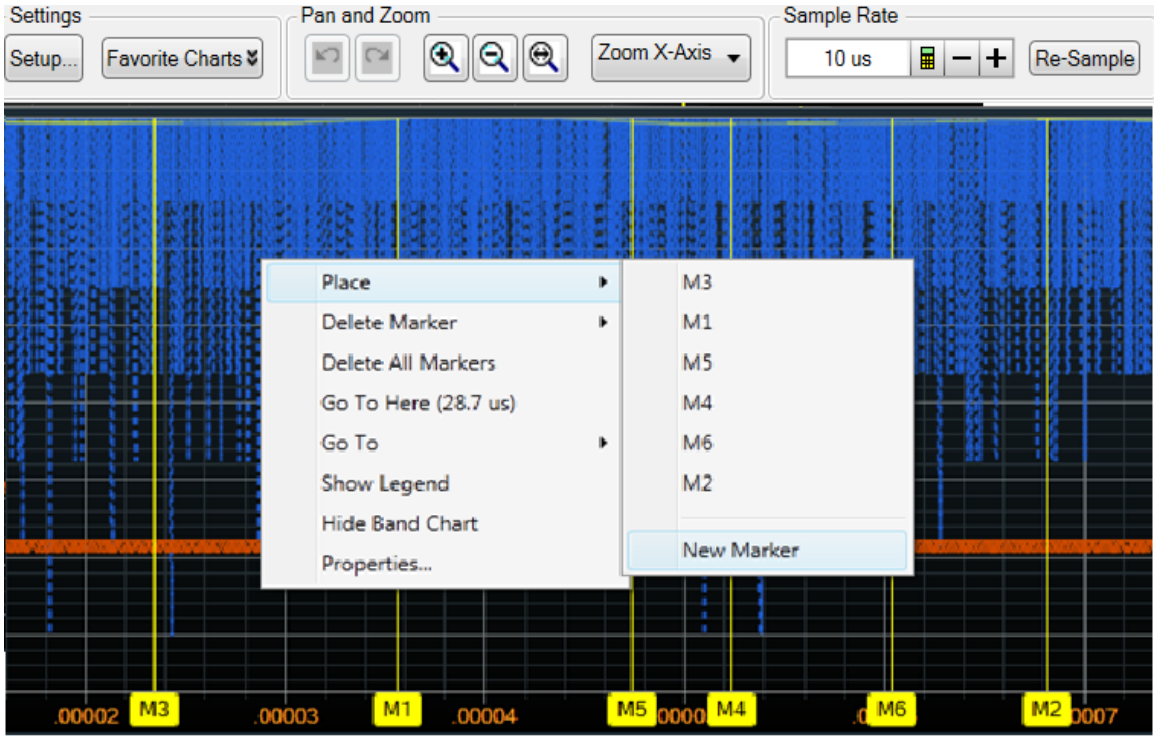
This type of navigation is particularly useful when you notice a sudden variation in a chart and want to navigate to the exact trace position that corresponds to that chart location.



To place a marker in charts

1 Double-click the location in the chart at which you want to place a marker. A new marker is added to that chart location and the corresponding trace location in the upper pane (trace view).

Alternatively, right-click the chart location where you want to place a marker. Then select **Place** > **New Marker** or select an existing marker to place that marker at the current location.

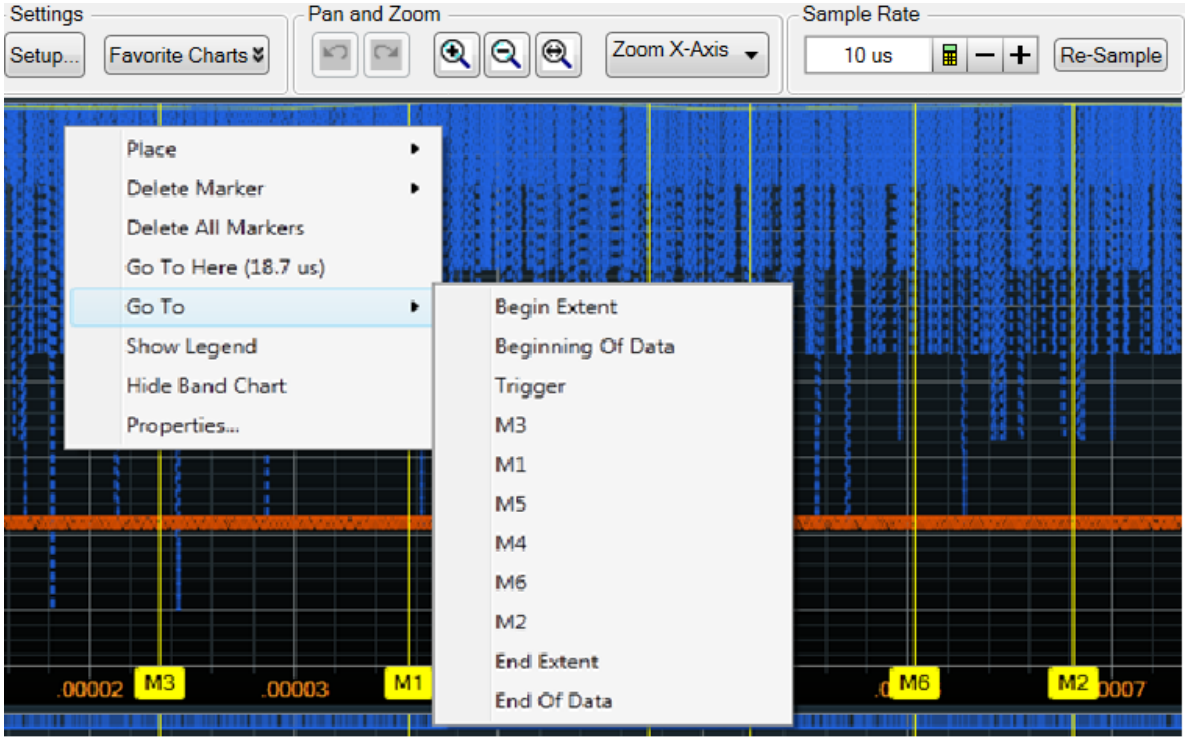


To navigate to a particular marker placed in charts

In situations when you have placed multiple markers in charts, you may want to navigate to a particular marker and its associated trace position in the upper pane. To do so, right-click anywhere in a chart, select **Go To** and then select the marker to which you want to navigate.

On doing so, the chart display moves to the point at which the selected marker is located. Also, the trace position corresponding to the selected marker is highlighted in the upper pane.

NOTE If the markers are not displayed in charts, click the Show Markers button at the top of the charts pane.



By using Extent Markers

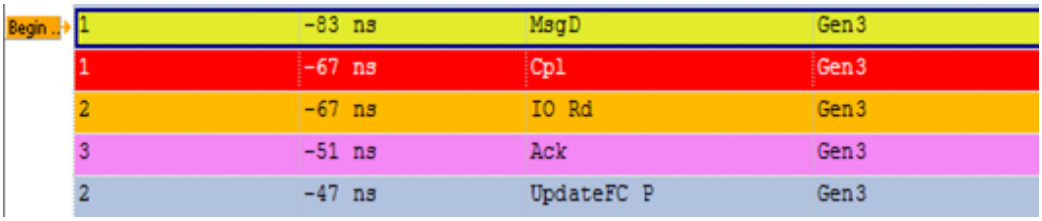
When you pan/zoom a defined area in charts, Extent Markers are automatically placed at the beginning and end of this defined pan/zoom extent in charts. On changing the pan/zoom area, these markers are automatically moved. There may be situations when you zoom a specific area in charts and then want to navigate to the MPCIe data associated with the chart's zoomed area in a Waveform Viewer or a Listing. To accomplish this, you can navigate using Extent Markers.

To navigate using Extent markers:

1 Right-click anywhere in the zoomed area in a chart and select **Go To**.

2 Then select **Begin Extent** or **End Extent** to navigate to the data associated with the beginning or end of the zoomed area.

The applicable data is highlighted in the upper pane of Protocol Viewer as well as in Waveform and Listing viewers.



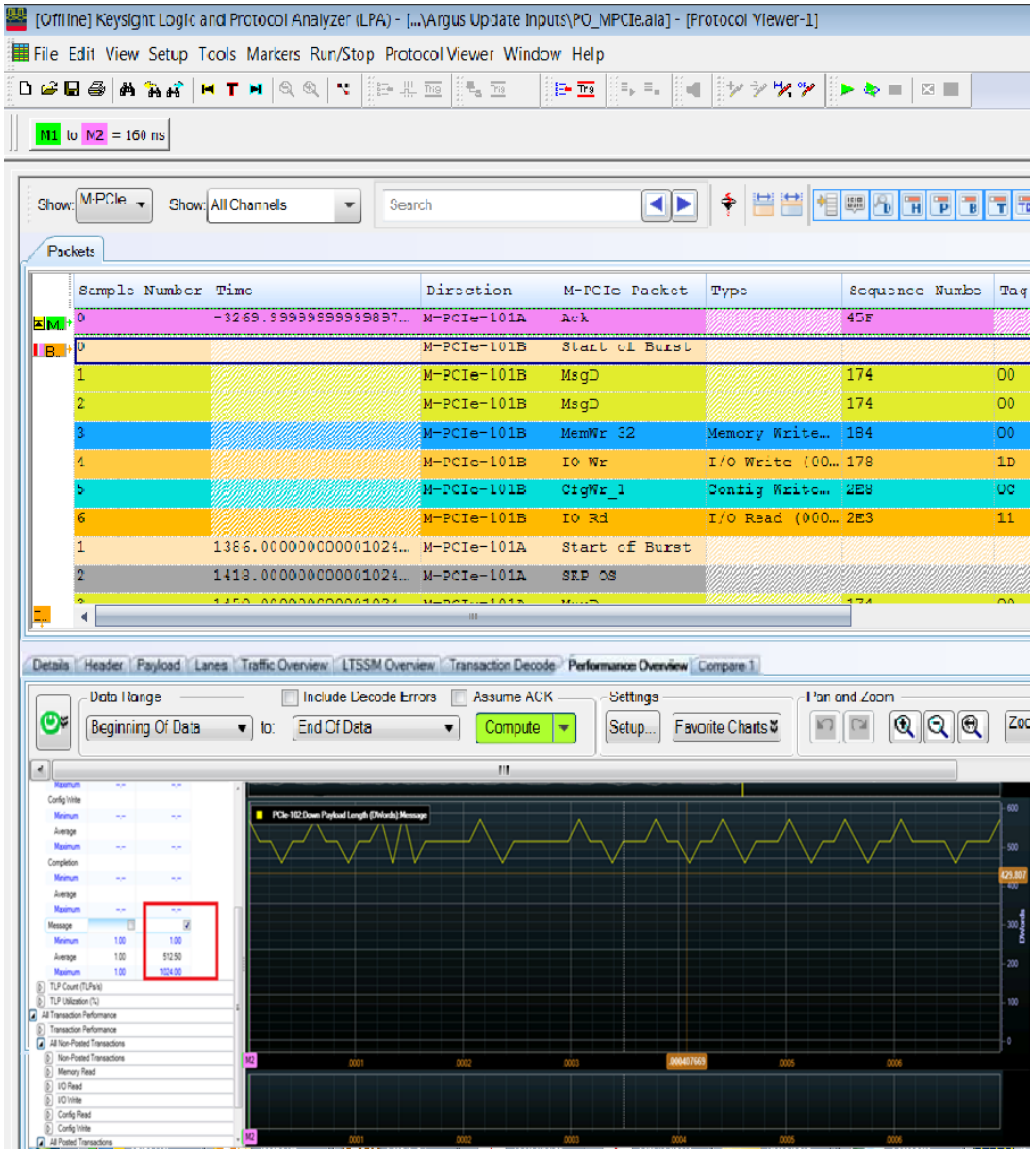
NOTE If the Extent Markers are not visible, then click the **Extent Markers** button at the top of the charts panes to display these markers in trace view as well as other viewers in Logic and Protocol Analyzer application.

From the Performance Statistics pane

For some performance parameters such as **Minimum** and **Maximum**, it is possible to navigate directly to the MPCIe/SSIC packet that accounted for the generation of a particular statistical value of that performance parameter.

Such performance parameters for which navigation to the packet is applicable are displayed in blue color.

In the following screen the “Message” parameter in the downstream direction has a Min of 1.0 and Max of 1024 displayed in blue color. Clicking these numbers, navigate you to MsgD packets with lengths of 1 and 1024 (DWords) that is accounted for the minimum and maximum value respectively in the upper pane.



NOTE

The chart pane does not show the exact value of the data packets as displayed in the upper pane of the Protocol Viewer window. This is because the chart pane displays the graph of sampled series that represents an average value of all the packets transmitted during a particular micro second as defined in the sample rate field.

Customizing Charts

Changing the Sampling Rate and Re-sampling the Chart

Sampling is a process by which captured data trace is sampled on the basis of specified time period to create charts. The trace data is sampled to compute statistics from these samples for charts generation. This sampling is done as per the *Sample Rate* set for charts. By default, the sample rate is set to 10 microseconds. You can change this sample rate and can re-sample the data trace by specifying time slices. By re-sampling the data you can regenerate charts based on the changed sampling rate.

To change the sample rate of charts

1 Access the **Performance Overview** tab.

2 Click the  button displayed with the **Sample Rate** field in the Charts pane on the right.

3 In the **Time** dialog box, specify the value and unit for the sample rate. The permissible range for sample rate is 1 us to 100 ms.

4 Click **OK**.

To regenerate charts based on the new sample rate

1 Click the **Re-Sample** button displayed with the **Sample Rate** field at the top of the Charts pane.

NOTE Sampling is done on the basis of specified time period defined as per the sample rate. It is not applicable for Flow Control category because Flow Control statistics is not time dependent and is based on the flow control credits of the data transmitted and received, The series listed under the Flow Control category have packet granularity whereas sampling is significant for all other categories listed in the Statistics tree/table of the Performance Overview tab.

When you capture a data trace and sample it at defined sample rate to generate charts, there may be repeated scenarios when no message packets are received during a particular slice of time while sampling the entire trace. Such scenarios are represented in the graph by plotting glyphs at zero entry. The glyphs plotted at zero, which you see in the chart, signifies no data packet received at that time.

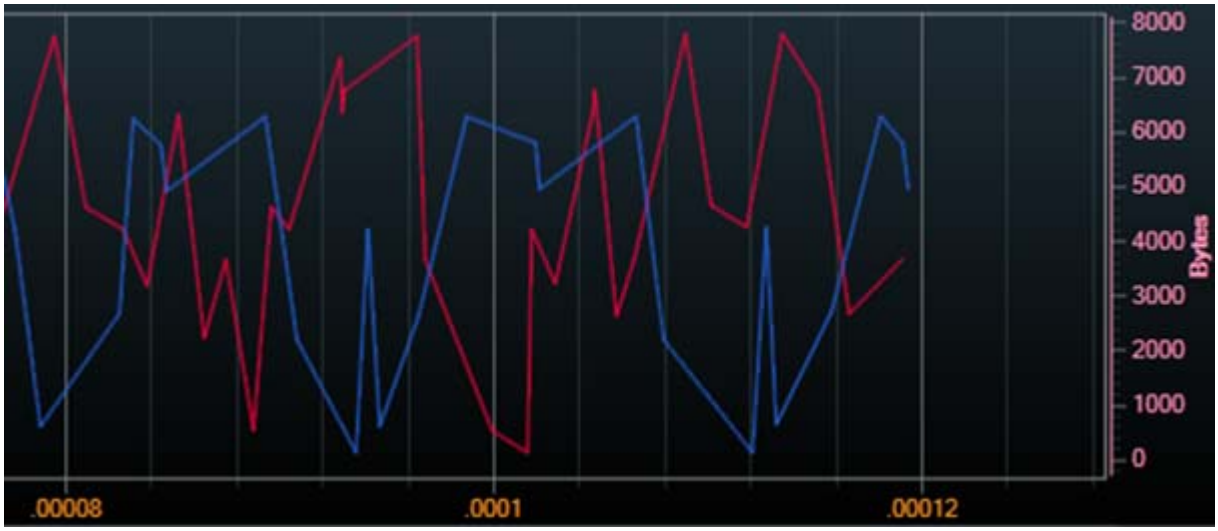
NOTE Message packet displaying a zero/hyphen value in the chart does not mean it has zero value, rather it means no relevant data received at that time.

Changing the Chart Display

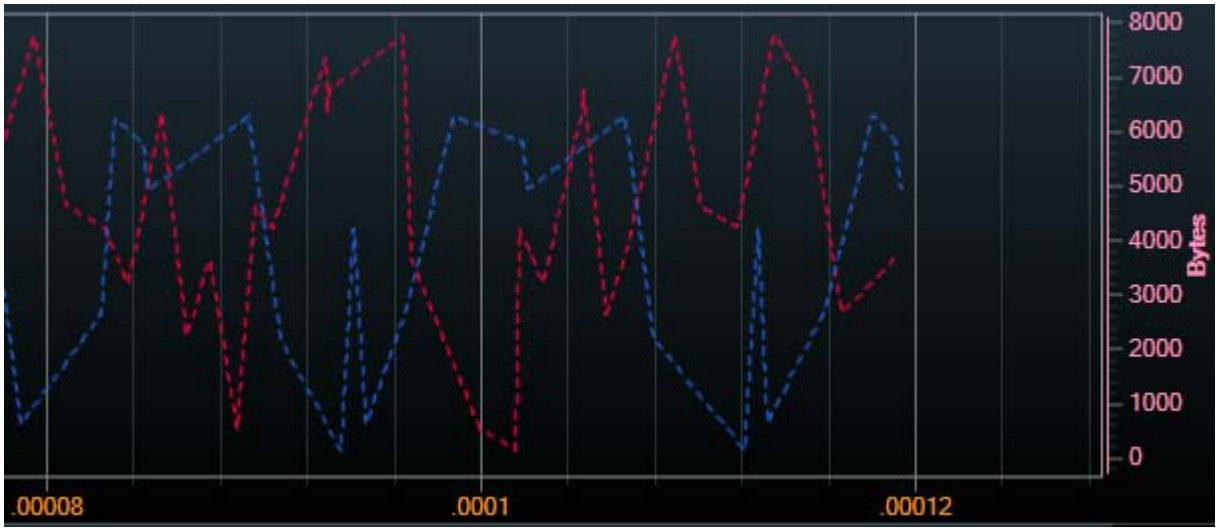
You can change how data is presented in a performance chart. To know, how to change the chart display, see "Changing the appearance of the graph" on page 160.

By default, data is presented as Dash Line chart type. The following chart type options are available.

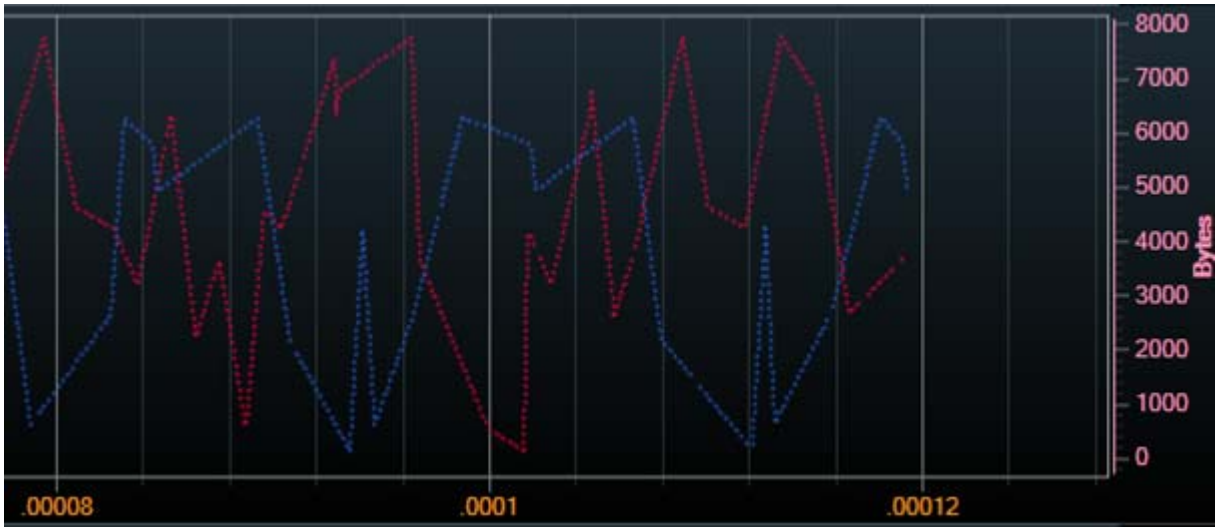
- Line -



- Dash Line-

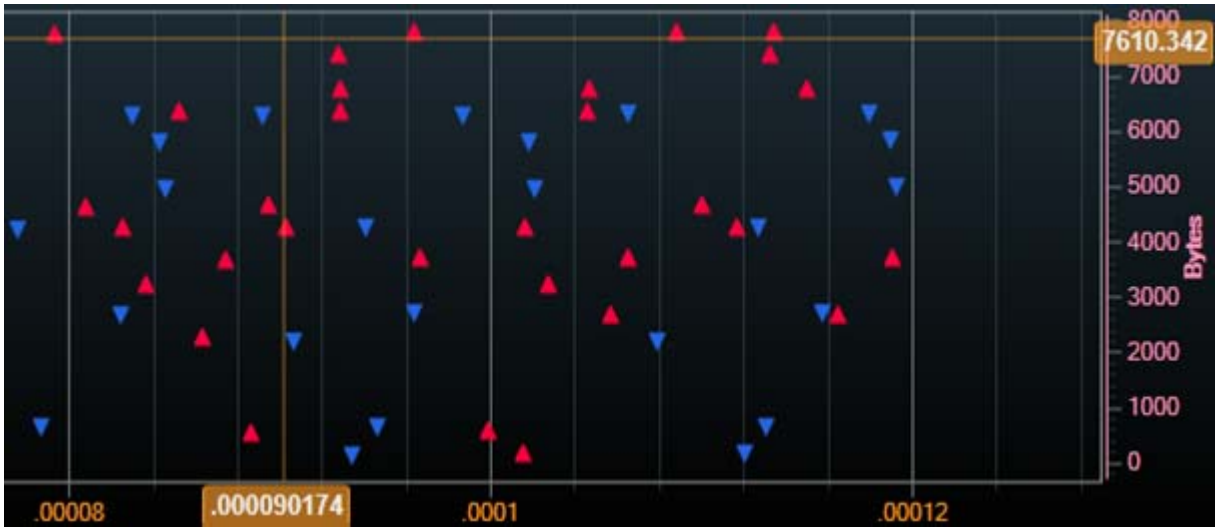


- Dot Line -

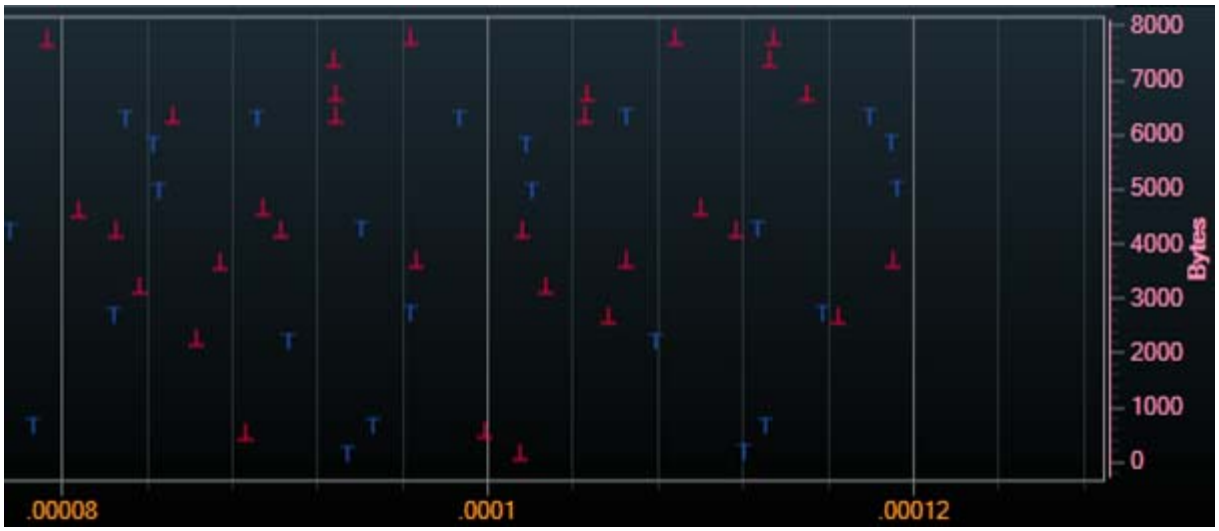


- Triangle Up/Down -

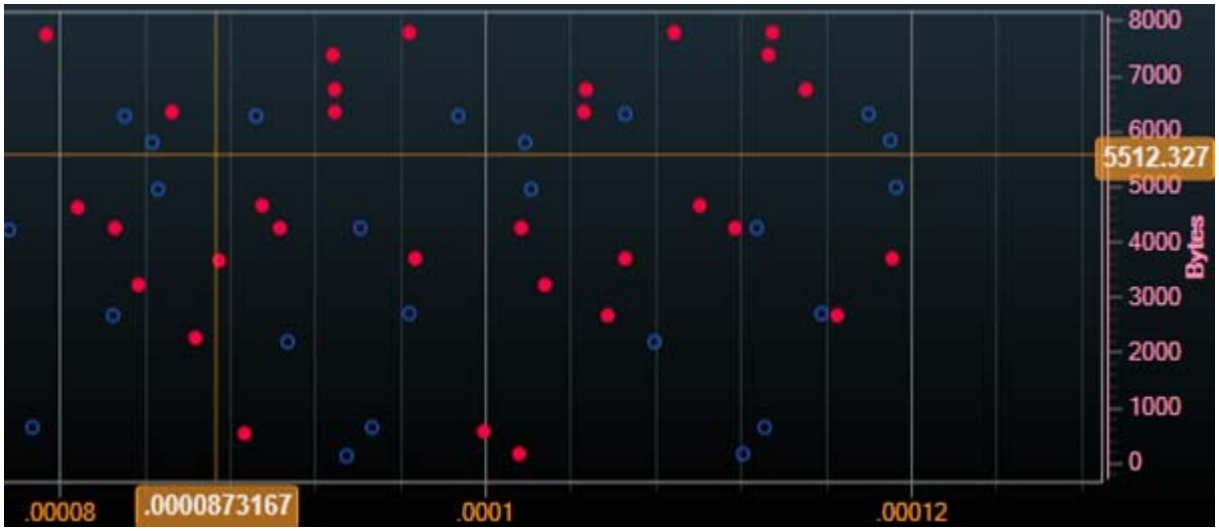
- Pyramid Up/Down -



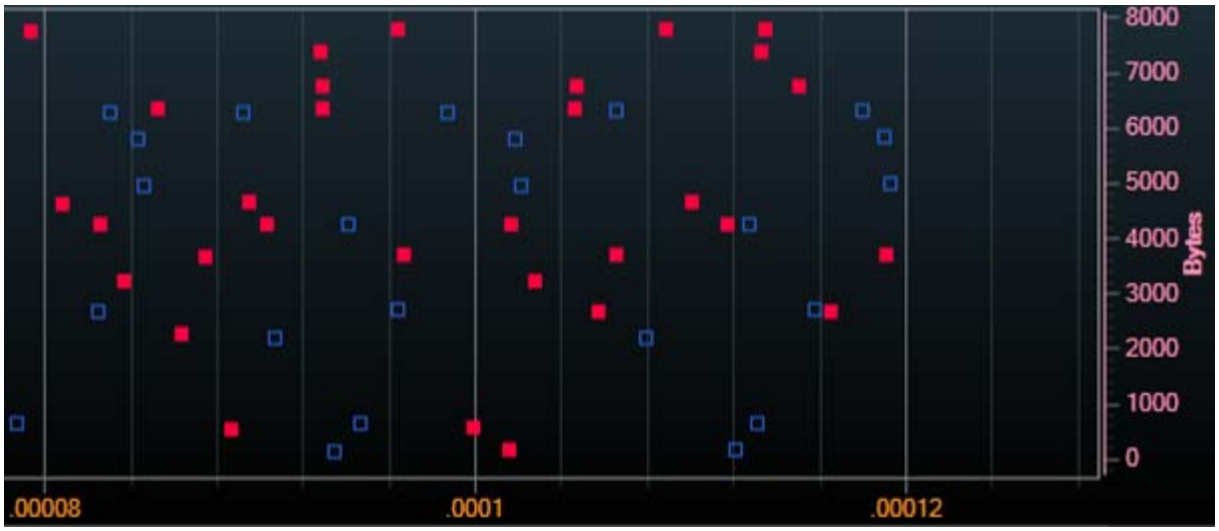
- Perpendicular Up/Down-



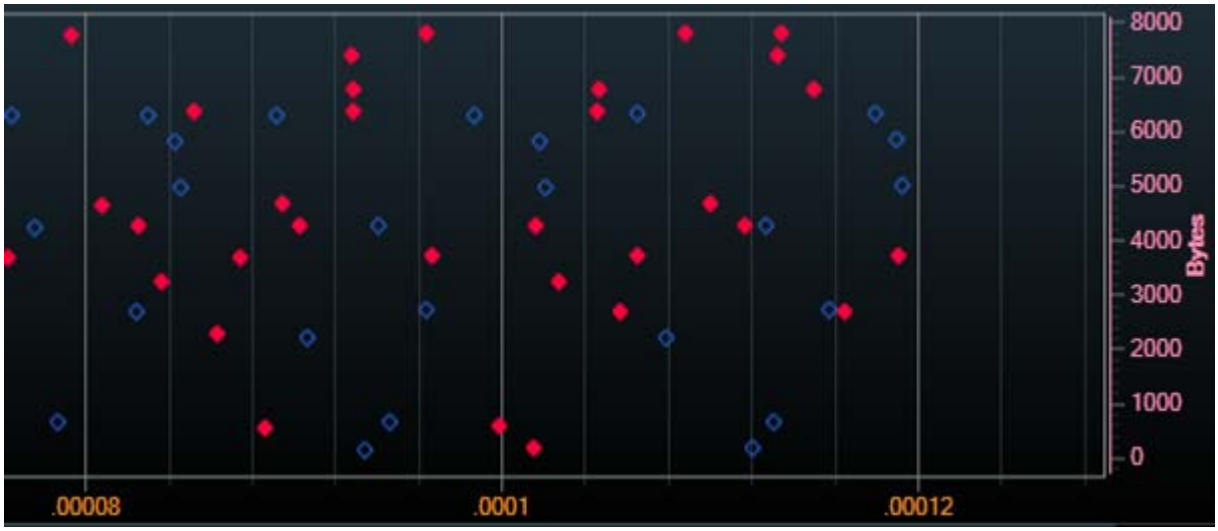
- Dot/Circle -



- Box/Square -



- Diamond/Filled Diamond -



To change chart type for a performance chart

Click the drop-down in the **Type** column from the **Setup...** dialog box and select the type you want to change with.

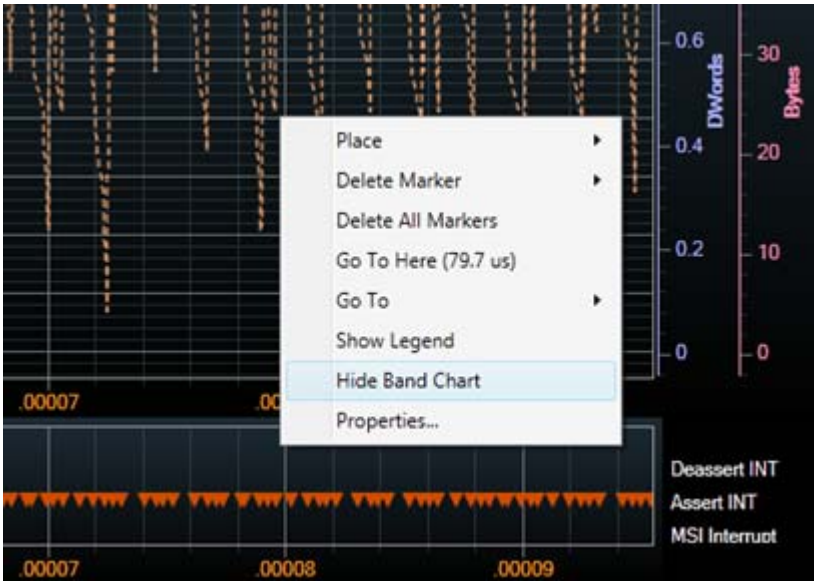
The chart is displayed as per the changed chart type.

Showing/Hiding the Band Chart

You can show or hide the Band Chart displayed in the lower pane of the chart area.

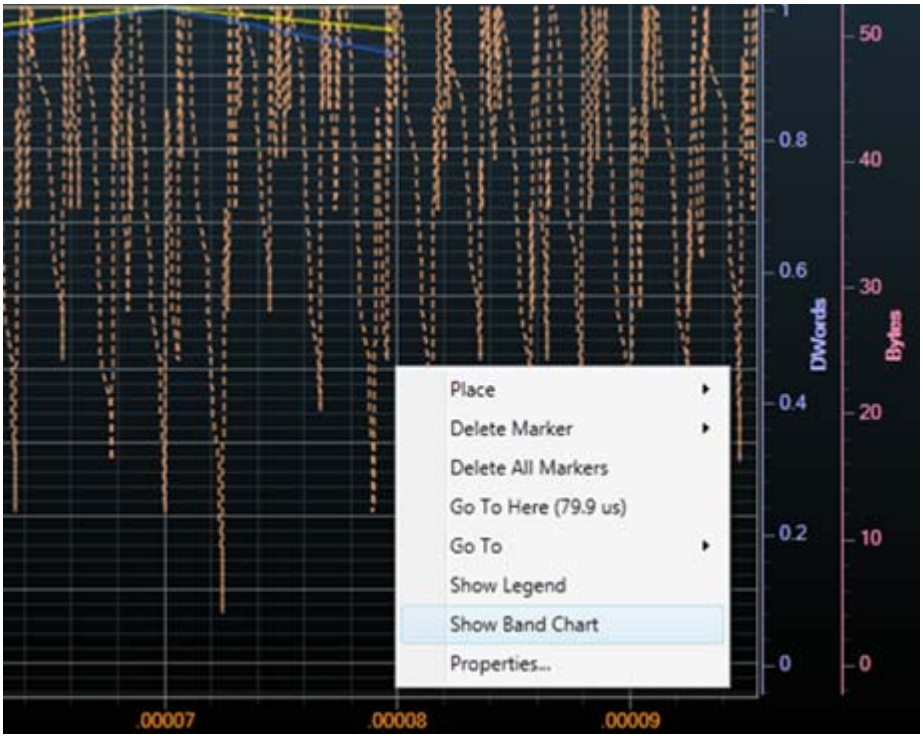
To hide the Band Chart

Right-click anywhere in the chart location and then select **Hide Band Chart**.



To show the Band Chart

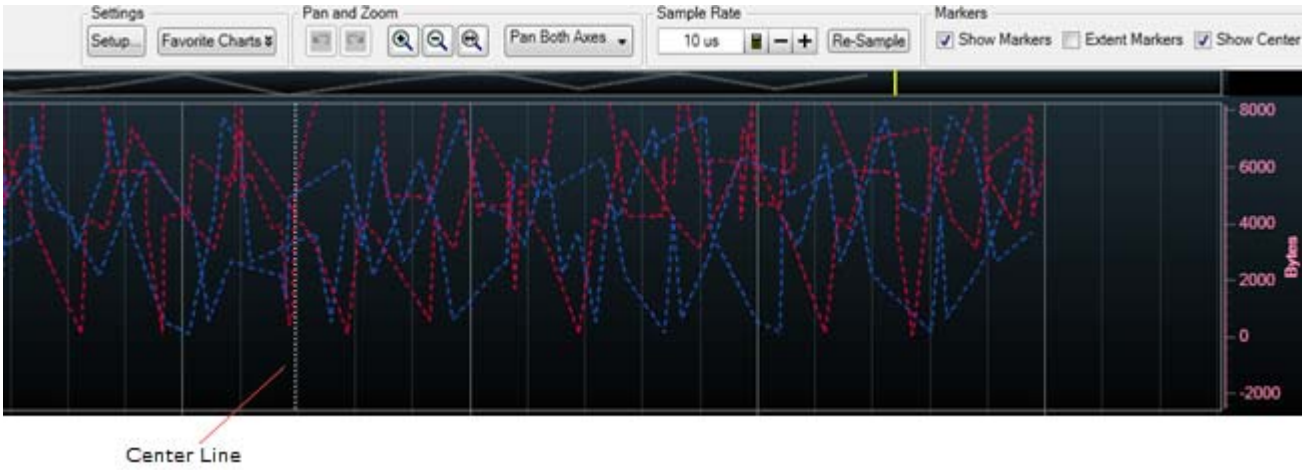
Right-click anywhere in the chart location and then select **Show Band Chart**.



Viewing the Center of the Chart

If the captured traffic is too large and defined pan/zoom extent in the overlay chart is outsized, there may be situations when you want to view where the center of the chart's zoomed area lies. You can do so by using **Show Center** check-box.

To view the center of the chart's zoomed area, click the checkbox **Show Center**. Selecting this checkbox displays a white colored vertical dotted line at the center of the Overlay Chart. It makes the center of the chart apparent as displayed in the following screen:

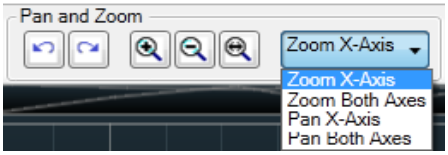


Zooming In/Out Charts

You can zoom in or zoom out a defined area in the chart or the complete chart.

To zoom X-Axis for a defined area in the chart

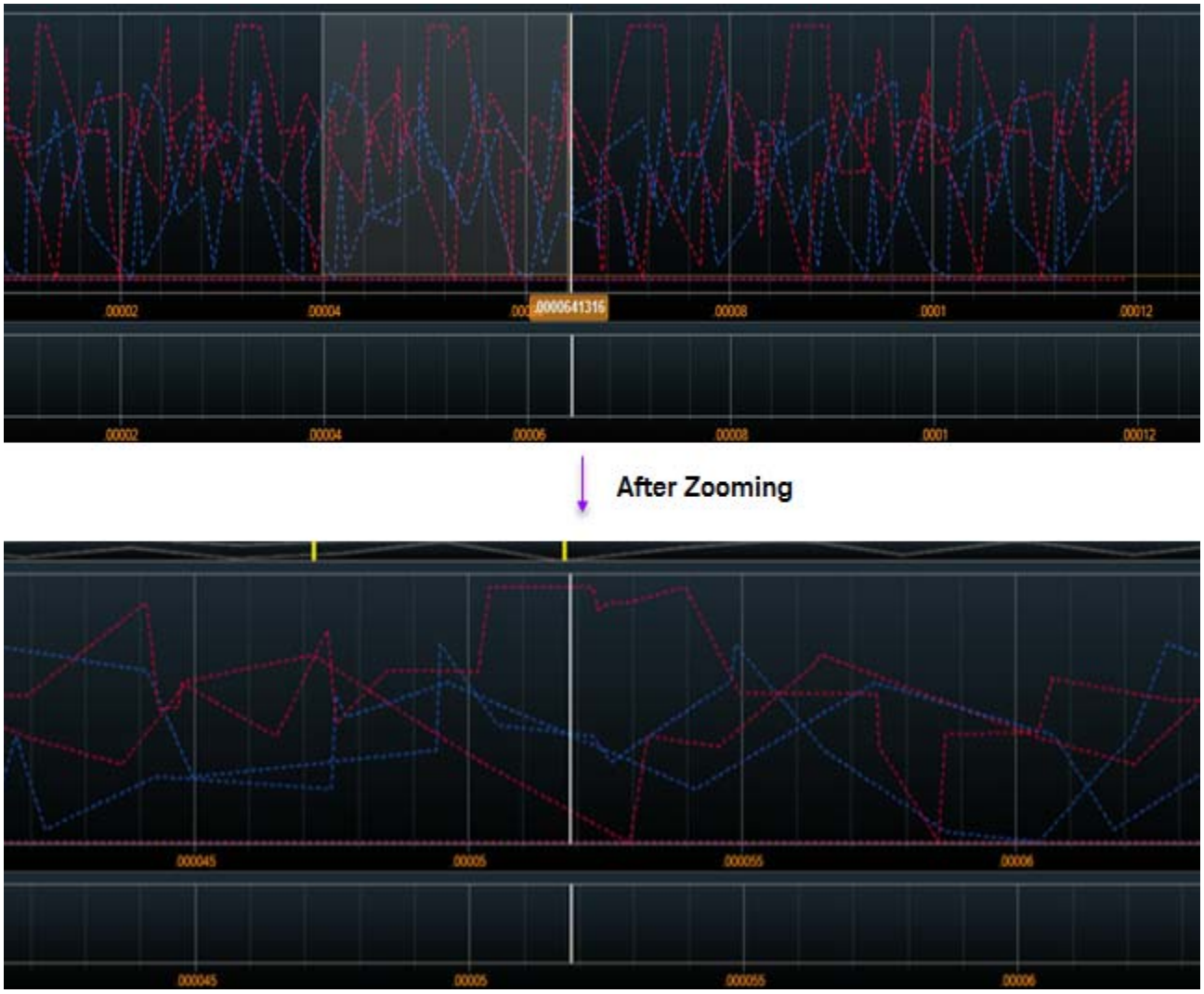
1 Click the **Zoom X-Axis** option from the combo box displayed in the **Pan and Zoom** section of the charts pane to make it active.



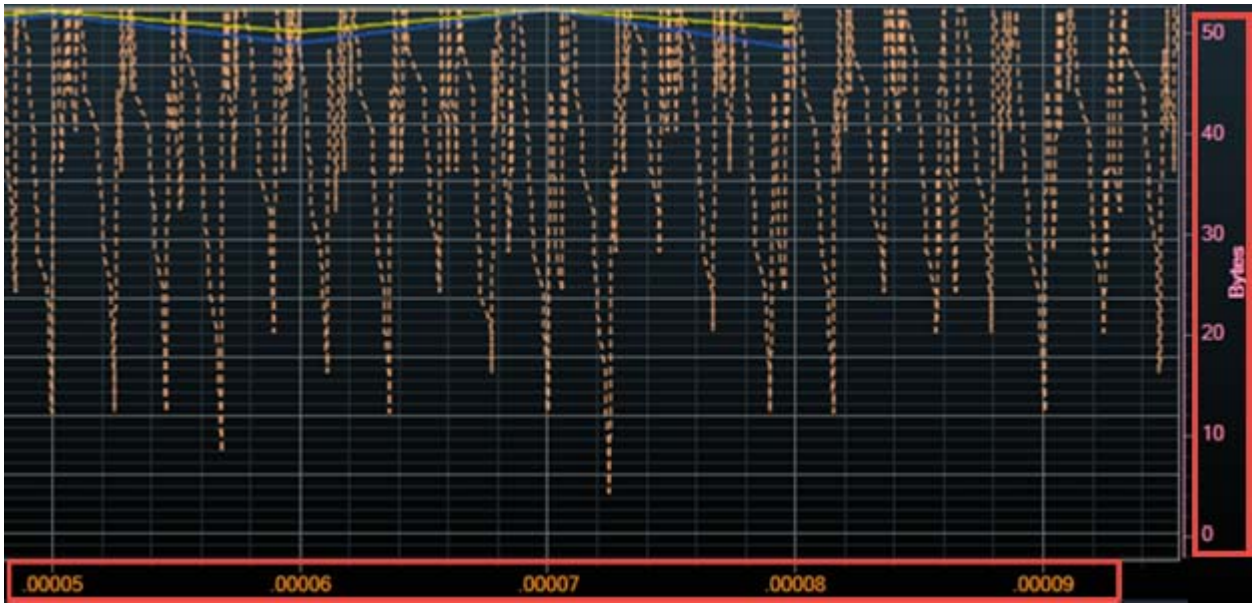
2 Move the mouse pointer to the chart location from which you want to begin zooming.

3 Left-click at this location and while keeping the left mouse button pressed, drag the mouse to the chart location till which you want to zoom the display. As you move the mouse, the zooming extent is defined in chart and highlighted with grey.

When you release the left mouse button, the defined X-axis area is zoomed for all the displayed charts. Following screen displays the chart before and after zooming in Overlay view:



Similarly, you can zoom both X and Y axes of the defined area in the chart by selecting the **Zoom Both Axes** option from the combo box displayed in the **Pan and Zoom** section of the performance charts pane.



NOTE The X-axis zoom applies to all the displayed charts where as the Y-axis zoom (in both axis zoom) applies only to the chart in which you define the area to zoom.

You can also zoom in or zoom out complete charts. To do this, use the following buttons in the **Pan and Zoom** section of the charts pane.

- - **Zoom In** magnifies the center 50% of the chart to the full width of the chart.
- - **Zoom Out** doubles the time displayed in the full width of the chart.
- - **Zoom Out Full** displays the entire range of Computed data across the full width of the chart.

NOTE You can undo and redo zooms by clicking the and buttons in the **Pan and Zoom** section of the charts pane.

Index

T

U

V

W

Z

This information is subject to
change without notice.
© Keysight Technologies 2014-2015
May 2015
U4431-97002

KEYSIGHT
TECHNOLOGIES

www.keysight.com