STX SERIES

SATA Serial Bus Protocol Analyzer

User's Manual

Compatible with software version 1.x

CATALYST ENTERPRISES INC.

February

P/N 400-0078-005

Trademarks and Servicemarks

Microsoft, Microsoft Windows 2000, and Windows XP are either registered trademarks or trademarks of Microsoft Corporation

Intel and Pentium are registered trademarks of Intel Corporation

AMD Duron and AMD Athlon are trademarks of Advanced Micro Devices Inc.

All other trademarks and registered trademarks are property of their respective owners.

THE SPECIFICATIONS AND INFORMATION REGARDING THE PRODUCTS IN THIS MANUAL ARE SUBJECT TO CHANGE WITHOUT NOTICE. ALL INFORMATION, EXAMPLES AND RECOMMENDATIONS IN THIS MANUAL ARE BELIEVED TO BE ACCURATE BUT ARE REPRESENTED WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED. USERS ARE FULLY RESPONSIBLE FOR THEIR APPLICATION OF ANY PRODUCTS.

THE SOFTWARE LICENSE AND LIMITED WARRANTY FOR THE ACCOMPANYING PRODUCT ARE SET FORTH IN INFORMATION THAT SHIPPED WITH THE PRODUCT AND ARE INCORPORATED HEREIN BY THIS REFERENCE. IF YOU ARE UNABLE TO LOCATE THE SOFTWARE LICENSE OR LIMITED WARRANTY, CONTACT LeCroy FOR A COPY.

---

http://www.lecroy.com

LeCroy Corporation.
3385 Scott Blvd.
Santa Clara, CA. 95054
1- (800)-909-7112 or (408) 653-1260
psgsupport@lecroy.com

Table of Contents

Introduction

What's In this Manual

This manual describes the installation and operation of your Catalyst Enterprises **SATA Analyzer: Models STX-460 (4-Port), STX-430/431 (4-Port), STX-430/431 IB* (One x4 Wide Port SAS)** or **STX-230/231 (2-Port)**. Examples of some typical applications are included.

* STX-430IB is functionally identical to STX-430 except that it provides an alternative Front Panel 4x Port connection.

Note: Unless stated otherwise references to STX-230 apply to STX-230 and STX-231 and references to STX-430 apply to STX-430, STX-430IB, STX-430-IBP, STX-431 and STX-431IB*.

Various Available Models

STX products are available in three models, STX-230/231, STX430/431, STX-430/431IBP & STX_460 with major differences summarized below:

Model Major Differences

	STX-230/231	STX430/431	STX-430IB/430IBP/431IBP	STX-460
Ports	1 or 2	4	4	4
Interconnect	SAS	SAS	x4 SAS	Mini SAS
Analyzer	1 or 2	4	4	4
Initiator Emulation	1	2	2	-
Target Emulation	1	2	2	-

Analyzer Overview

The **SATA Analyzer** is a serial bus analyzer that is capable of analyzing Serial ATA data transfers. The analyzer is based on the STX hardware platform that performs serial bus analysis for SATA when controlled by the SATA analyzer software.

The analyzer supports the following:

- Capture and Trigger of Serial ATA packets
- Generation of bus traffic with the **Host Emulator** while monitoring and analyzing the result
- Run a Bus Performance Analysis
- Run a Pattern Generator
- TX Vout on transmitters for test and characterization

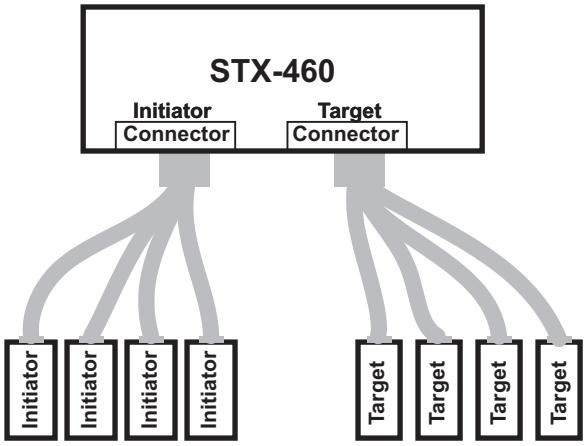


Figure 1 Typical SATA Test Setup of Analyzer

The analyzer provides for bi-directional trigger and capture of commands, primitives and all bus conditions.

STX Interface

- Power In
- USB Port
- Ethernet Port
- SAS/SATA Ports (2 Ports for STX-230 and 4 Ports for STX-430 and STX-430IB)
- External trigger Input
- External trigger Output
- Stacking connections

LEDs

Each link is supported by 5 LEDs with the following functionality:

Green This LED is illuminated during the OOB sequence before the link is established and after link is established it indicates traffic on the bus.

Orange This LED is illuminated as follows:

Speed	Initiator	Target
1.5G	Off	Off
3.0G	On	Off
6.0G	On	On

Yellow This LED is illuminated when a link is established.

Red This LED illuminates when an error occurs.

Blue This LED is illuminated when a trigger occurs.

Receiving Your Analyzer

Your analyzer package includes the following components:

Carrying Case

STX unit identified in the packing list

SATA software on CD ROM

One USB 2.0 Cable 1.8 meter

x1 SATA Cable (2 for STX230, 4 for STX430), 1 Meter

Two SAS x4 Cable (Crossover) (for STX430IB and STX430IBP), 1 Meter

One Stacking cable (For STX-430 and STX-460 Units)

Two External trigger cables

Two iPass to iPass 1/2 meter cables (for STX-460)

Two iPass to IB 4X 1 meter cables (for STX-460)

One iPass to SATA 1 meter octopus cable (Straight) (for STX-460)

one iPass to SATA 1 meter octopus cable (Crossover) (for STX-460)

Ethernet Cable

User's Manual

Unpacking Your Analyzer

Inspect the received shipping container for any damage. Unpack the container and account for each of the system components listed on the accompanying packing list. Visually inspect each component for absence of damage. In the event of damage notify the shipper and Catalyst Enterprises. Retain all shipping materials for shipper's inspection.

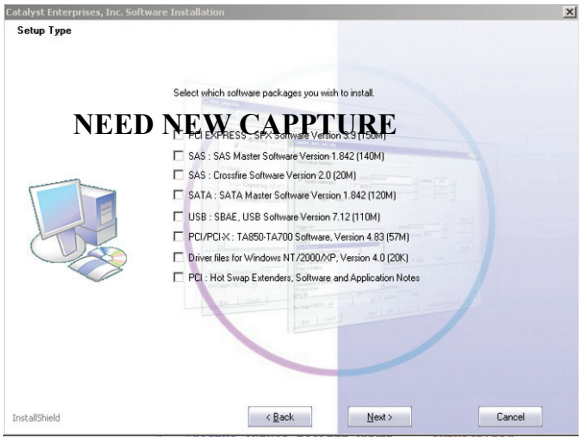
Installing Your Analyzer

Software Installation

On systems operating under Windows® 2000 and Windows® XP.

Do not connect STX Do not connect the STX to your host system until the software installation is complete.

1. Insert the CD ROM with the SATA software in the CD ROM drive.
2. The installation will automatically start the setup, unless the Auto Run is turned off. In this case select the CD ROM from "My Computer" and click setup.exe.
3. After the warning to close all other programs and before starting the installation, the Install component selection will open as shown below**.**



4. Select the desired components for installation.
5. Click **Next** to complete the installation.

System restart You must restart your computer before you can use your Analyzer software.

Error Message If you get an error message during installation of the drivers for Windows 2000 or XP consult your system administrator. Your system may be setup to only allow an administrator level to copy such driver files.

Hardware Setup

Separate Systems When using the SATA analyzer, it is recommended to use a system to generate bus traffic and a second system to run the SATA software in order to avoid characterization of analyzer traffic.

Connecting the STX Connect the STX as shown in Figure 2.

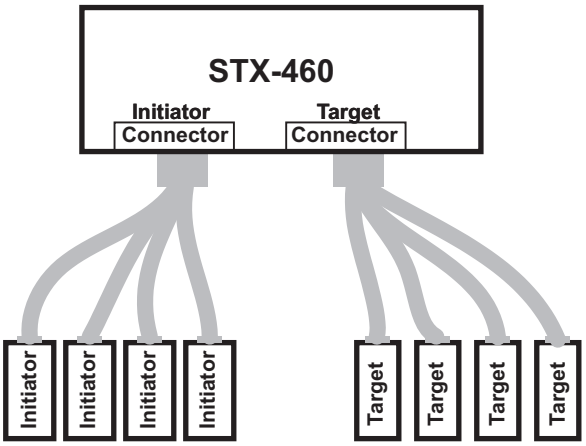


Figure 2 Analyzer Connections

Connecting the STX-430 IB

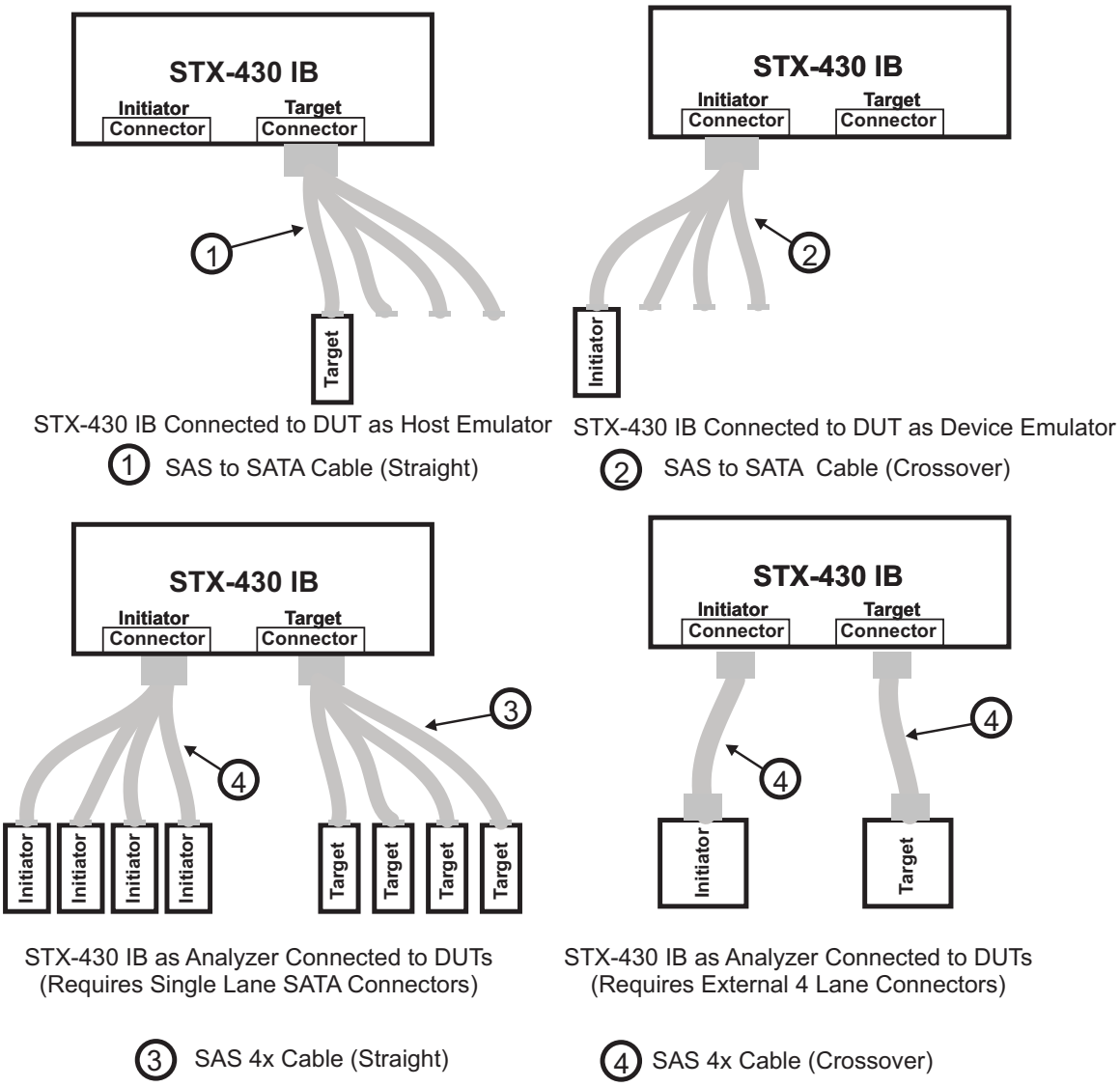


Figure 3 Connecting the STX-430 IB

Connecting the STX-430 IBP

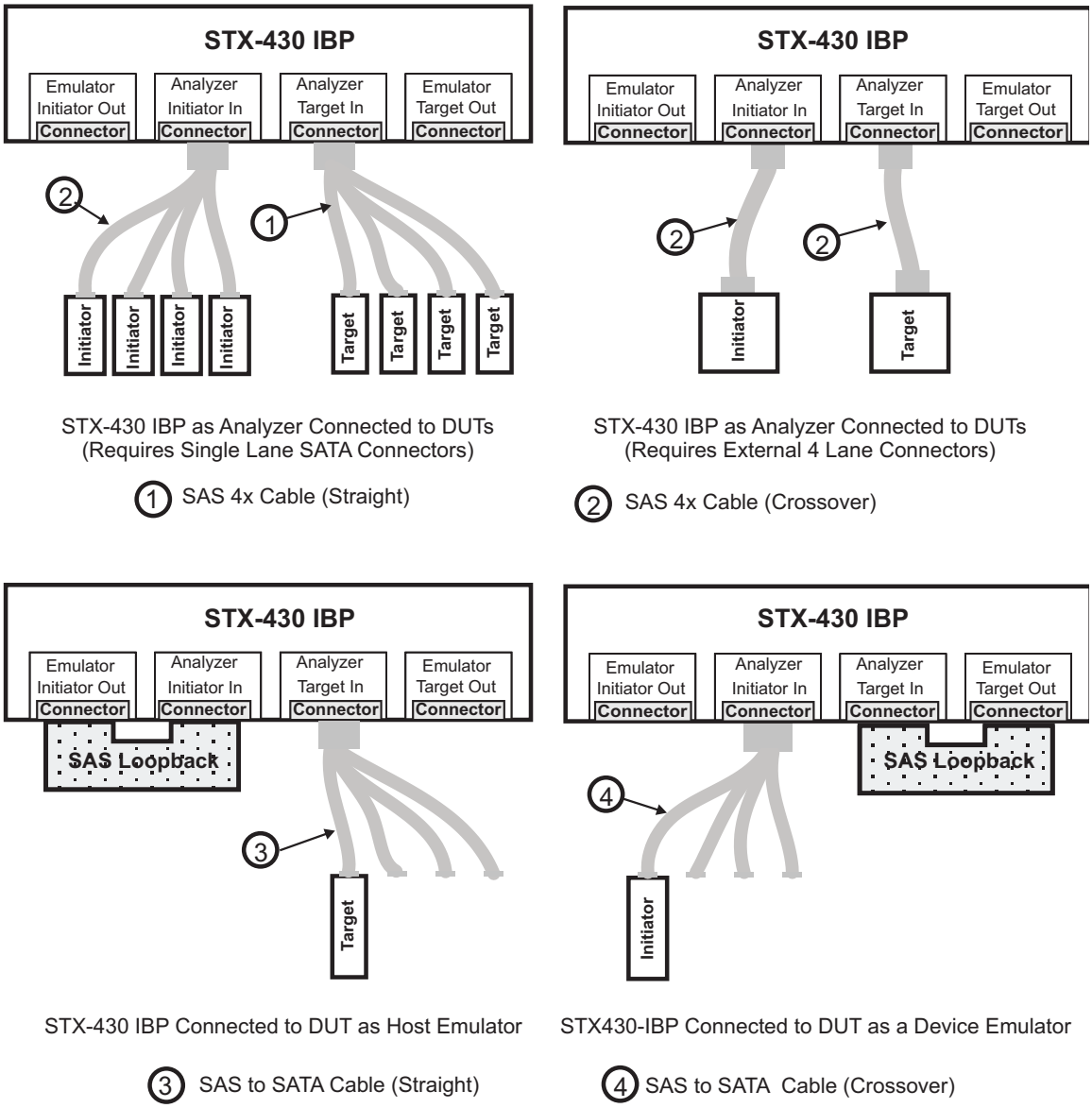


Figure 4 Connecting the STX-430 IBP

Expandability

STX units may be expanded for wider lane analysis. This may be accomplished by daisy chaining the units through the provided interface in the back. The “Out” connectors should be connected to the “In” connectors of the next unit in the chain for both, the signal and the clock interfaces.

The user needs to provide external hubs for connecting the host USB or Ethernet to these units.

Cascading STX-430’s and STX-460’s

Up to 8 STX-430 and STX-460 units may be cascaded and connected to a Host PC using USB or Ethernet cables.

To set up the STX units in a cascade:

1. Connect all of the units to be cascaded to the PC using either a USB cable or an Ethernet cable. You may use hubs (USB or Ethernet) to connect up to 8 units to a single PC.
2. Locate the Expansion ports on the back of each unit.

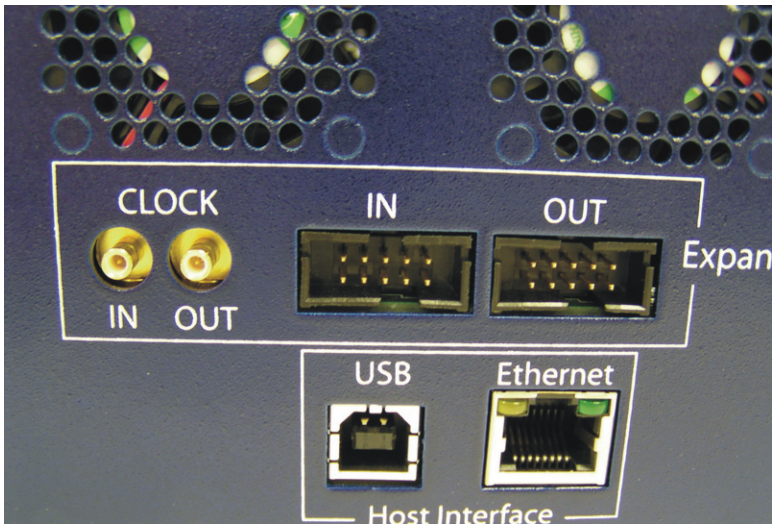


Figure 5 STX-430 Expansion Ports

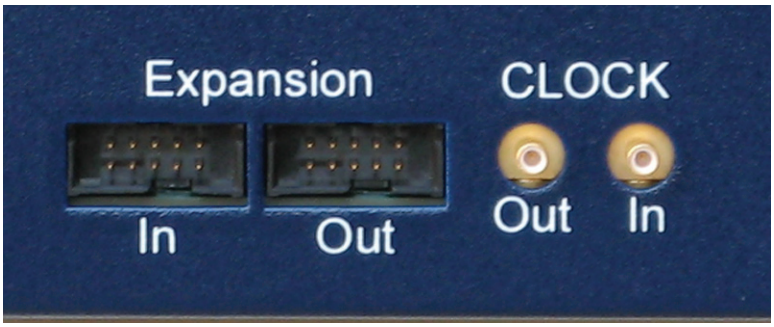
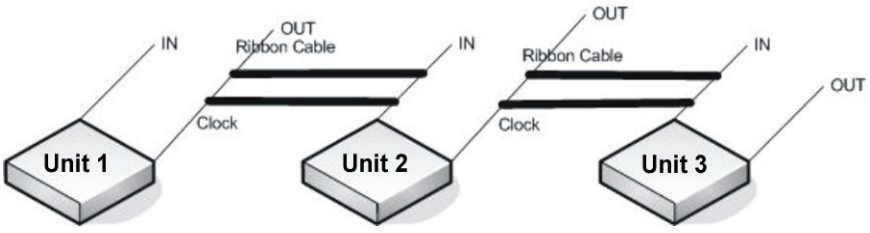


Figure 6 STX-460 Expansion Ports

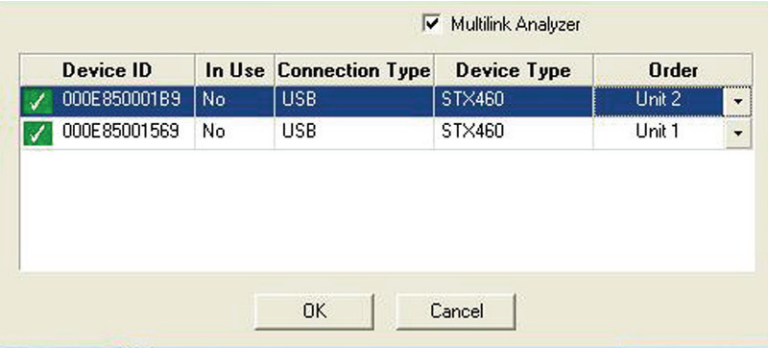
3. Connect the OUT clock connector of Unit 1 to the IN clock connector of Unit 2 using the supplied BNC Coax SMA to SMA cable.
4. Connect the OUT 10 pin connector of Unit 1 to the IN 10 pin connector of Unit 2 using the supplied 10 pin Ribbon Cable.
5. Similarly connect additional units up to a total of 8.
6. Arbitrarily designate one of the units as Unit 1.



7. Make a note of the last 4 digits of the MAC address of the unit designated as Unit 1 and the last 4 digits of the other units in the order of connection.

8. Make sure that all of the units are powered up and start the STX software.
9. The software will search for and display all of the connected units.

Note: Be sure to check the Multilink Analyzer check box.



10. Compare the MAC addresses, which are displayed in the field titled Device ID to those noted as they were connected, then click on the pull down tab under the heading “Order” on the right side of the menu and select the Unit numbers i.e. 1 for Unit 1, 2 for Unit 2 making sure your pre determined sequence for the Units matches with the MAC address for each Unit.
11. Click **OK** and let the STX software initialize so you can start capturing traces.

The units are now cascaded together.

Connecting via Ethernet

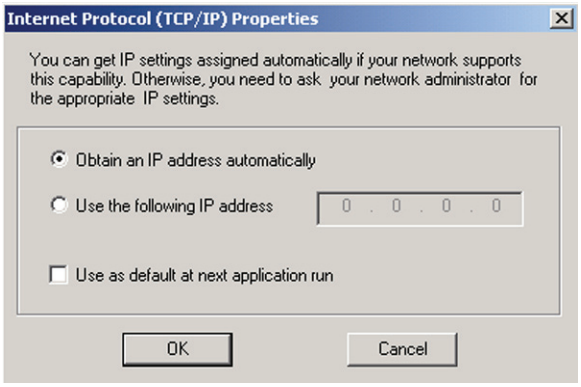
You may use the ethernet connection using any one of the following 3 supported configurations:

1. The STX connected to a network via a hub, switch, or similar device.
2. The STX connected to the host computer (machine running the application software), via a hub, switch or similar device.
3. The STX connected directly to the host computer using a crossover cable.

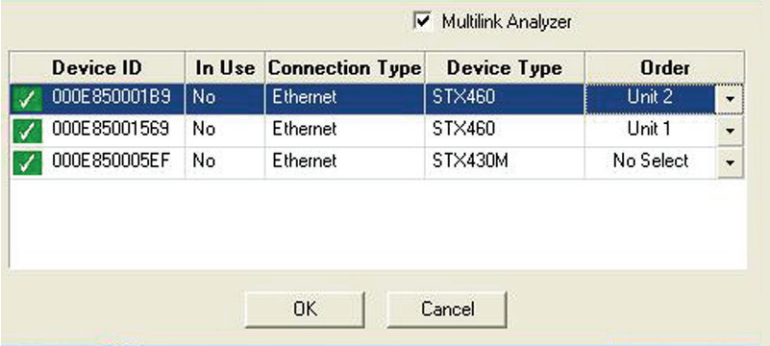
Connecting to a Network

When connected to a network the STX must communicate with the DHCP server to establish a connection. The DHCP server will continually send the next available IP address to the STX until the STX software is started.

When the user starts the software, the user may be prompted if they wish the software to automatically use the offered IP address or if they wish to assign a specific IP address (the assigned IP address needs to be on the same network segment as the host computer). The menu also allows the user to save the selected option (automatic or specific address). If the assigned IP address is not available, the OS will notify the user of an IP address conflict.



After the user clicks ‘OK’ the software will search for all STX units connected to the network, and will display a list of available STX units. After the user selects the desired STX unit, the software will assign the IP address to the selected unit, completing the connection and will launch the software.



Connecting via Hub, Switch or Similar device

When connected to the host machine via a hub, switch or other similar device or directly using a crossover cable the Catalyst board must communicate with the host computer to establish a connection. The host computer will continually broadcast the next available IP address to the Catalyst Board until the Catalyst software is started.

When the software starts, the user may be prompted if they wish the software to automatically use the offered IP address or if they wish to assign a specific IP address (the assigned IP address needs to be on the same network segment as the host computer). The menu also allows the user to save the selected option (automatic or specific address). If the assigned IP address is not available, the OS will notify the user of an IP address conflict.

After the user clicks 'OK' the software will search for all Catalyst boards connected to the network, and will display a list of available Catalyst boards, after the user selects the desired Catalyst board, the software will assign the IP address to the selected board, completing the connection and will launch the software.

Remote Operation

In order to operate your STX remotely you must install the Remote WAN Feature as described in Appendix B.

Note 1. When using the remote option, the software cannot detect the power cycle on the board.

Note 2. In the event that the software cannot connect to the server with an error message, you must exit and re-run the software.

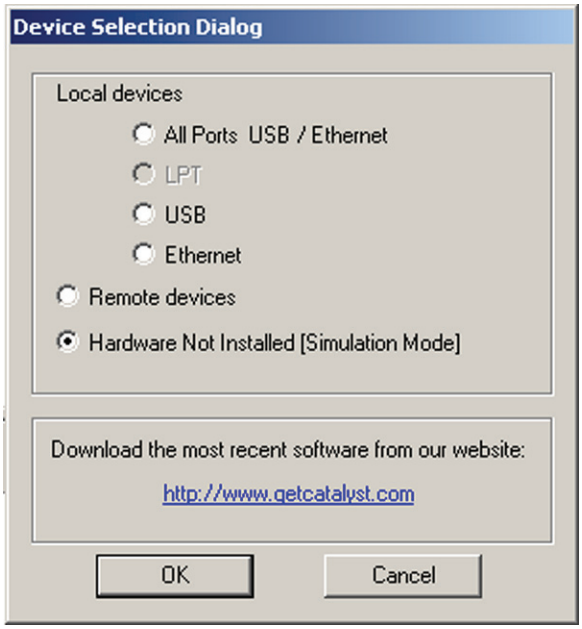
Launching Your Analyzer

SATA Analyzer

To launch the **SATA Analyzer** software, double-click the SATA Button on your Windows desktop. You can also launch the SATA Analyzer software from the start menu by choosing Programs and then the SATA folder.

The first time you run your software it will search for a default host interface and if it is found, the software will launch. If no interface is found the software will launch in simulation mode.

Establish Interface If no interface is detected initially, then establish an available interface and relaunch the software.



Click **OK** and the Analyzer will launch and display the Analyzer Tool bar for the Analyzer Software launched.

Software Launched

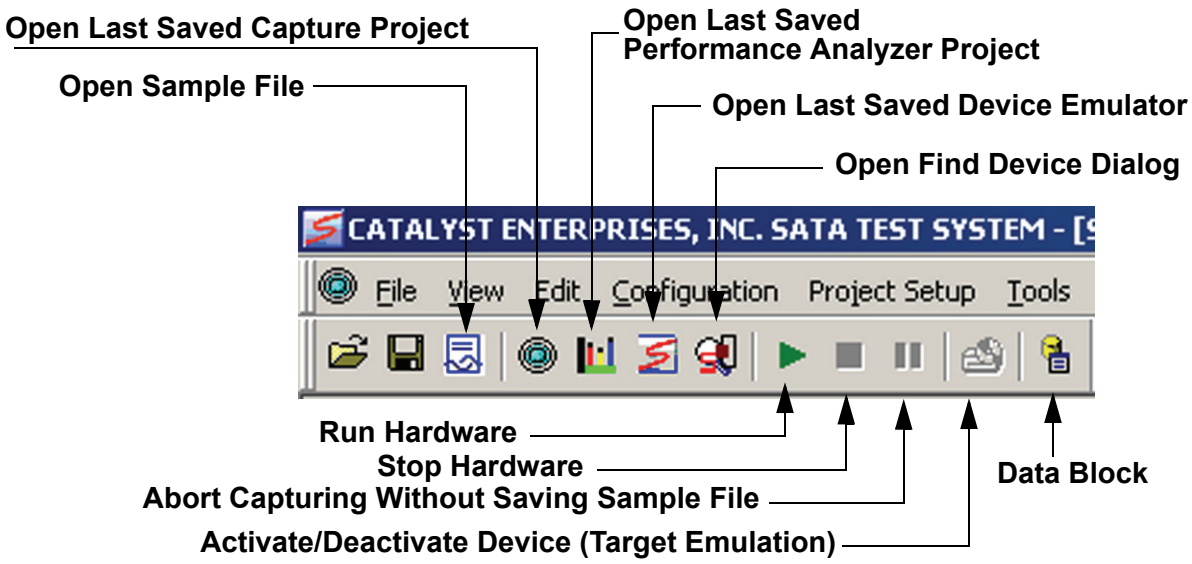


Figure 7 SATA Analyzer Toolbar

Operating in Simulation Mode

Your system will operate in the Simulation Mode by default if the hardware is not detected, however, you may operate in Simulation Mode directly without installing the Analyzer hardware. To operate without hardware, select **Hardware Not Installed (Simulation Mode)** in the Port Setting dialog box and click **OK.**

The Analyzer software will launch and display the appropriate tool bar, but with the limitation that the Analyzer will operate only on static, previously captured, bus data.

Limitations

The Simulation mode lets you try all of the available functions, but keep in mind that **the system is not capturing any real data and is displaying only pre-captured results.**

Protocol Analysis

Protocol Analysis is performed by defining and running an analysis project. An analysis project definition includes: defining what will be captured, what the analyzer will trigger on and capture memory settings. Defined projects are saved as project *.stc files for later use.

Easy Mode (Pre-Defined Setups)

This mode allows you to operate the STX with a minimum of setup. In this mode you may perform a Trigger and Data capture only or program the Host Emulator to generate bus traffic for triggering and data capture.

Quick Start

To get a comprehensive overview of your analyzer's capabilities:

1. Install the SATA Analyzer software. See "Software Installation" on page 5 for software installation instructions.
2. Set up your STX. See "Hardware Setup" on page 6
3. Launch your SATA Analyzer software. See "Launching Your Analyzer" on page 14 for launching instructions.
4. On the Analyzer Menu Bar click **File**, **New** and choose **Protocol Analyzer** to open an Analysis Project dialog.

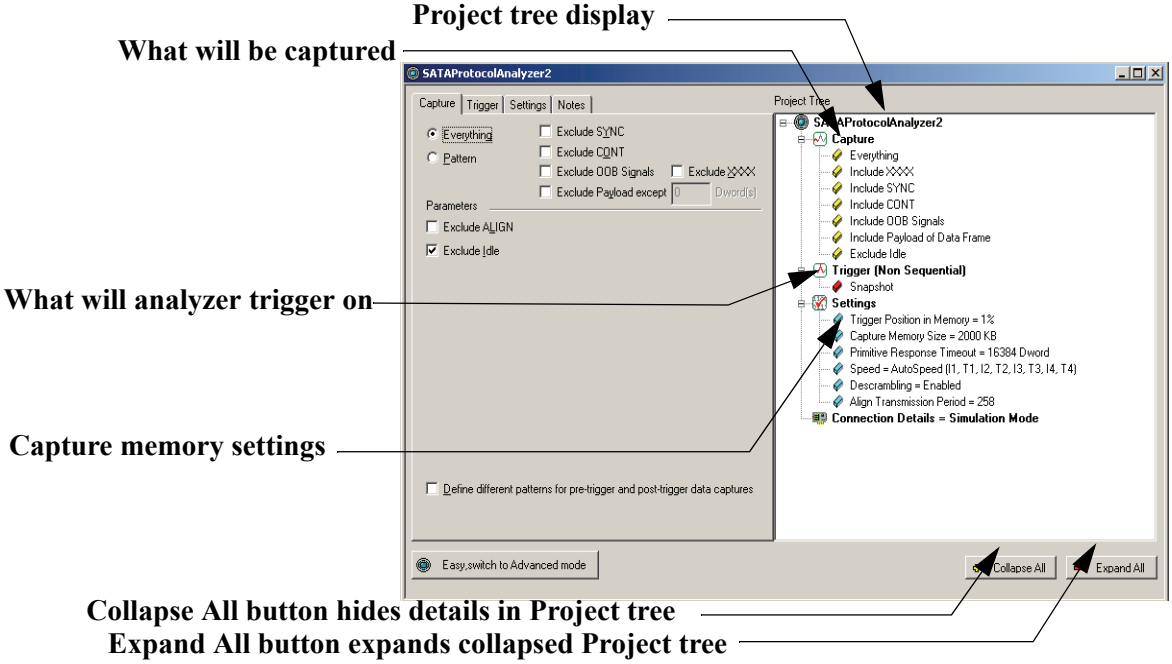


Figure 8. New Project Dialog

The New Project dialog opens with default settings to capture **Everything** on the bus and to **Trigger On Snapshot.** (Meaning that the analyzer captures everything immediately without triggering on anything in particular).

Project Overview

A comprehensive tree structured overview of the project is displayed in the **Project Tree display**. The project tree shows what is to be captured, what the analyzer will trigger on and capture memory settings.

5. To get an immediate overview of the bus traffic to and from your SATA Analyzer click the  **Run Hardware** button on the toolbar. After a short time a Packet view display opens.

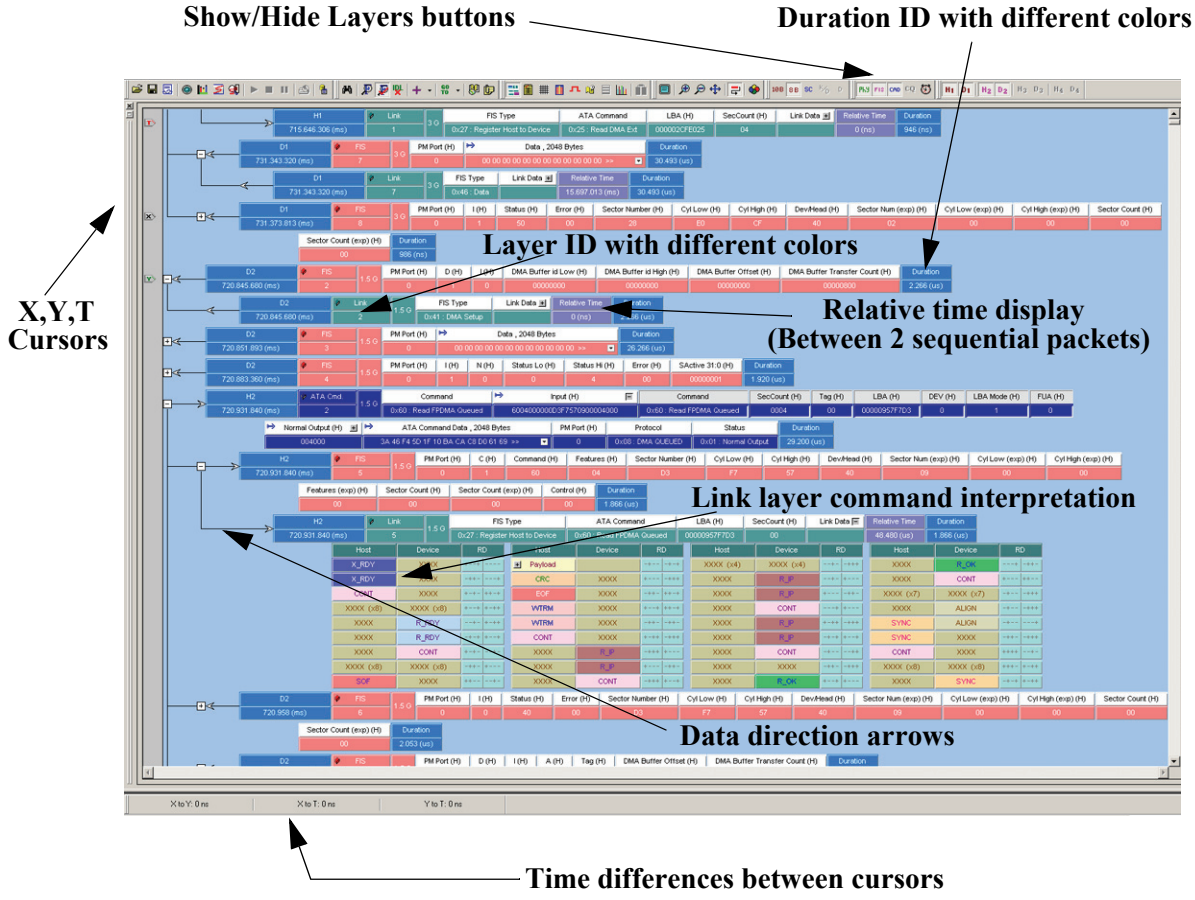


Figure 9. Typical Packet View Results Display

The results display shows each transaction for every layer identified in a different color and the data direction identified with data direction arrows. Device to Host traffic is identified with the arrow from right to left. This arrow direction ⇐ indicates Device to Host traffic. Host to Device traffic is identified with the arrow left to right. This arrow direction ⇒ indicates Host to Device traffic.

You may hide any layer by clicking the corresponding **Show/Hide** button on the menu bar. All captured data is retained, but the display is limited to the layer data of interest for simpler viewing.

The New Project dialog offers you a comprehensive set of choices to create a trigger and capture project to satisfy a specific need. You may set the Analyzer to:

- Capture specific patterns.
- Capture different patterns pre and post trigger.
- Exclude parameters from capture.
- Trigger on a Pattern or a sequence of patterns. (see "Triggering Setup" on page 33)
- Configure trace capture memory.
- Select file to save trace capture in memory.
- Select Pattern Generator file to run.
- Capture at appropriate speed.

Run a Sample Project Before setting up your own custom project you may wish to run one or more of the sample projects included with your analyzer software.

Example Projects

Your SATA Analyzer includes a number of pre-defined example projects that you may use to perform an immediate analysis without any setup.

The SATA Analyzer system software comes with a pre-defined folder (Directory) structure for storing all files. All pre-defined example files are stored in the following Folder:

c:\program files\catalyst\sata1.xx\examples

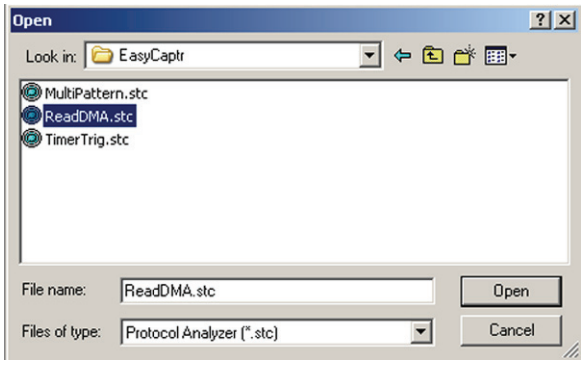
It is strongly recommended that you open some of these files to get an introduction to the types of projects that can be created for the SATA Analyzer.

Project file type definition

*.asl	Decoding Script File
*.stc	Data Capture file
*.sts	Sample file
*.stp	Performance Analyzer Project file
*.paf	Performance Analyzer Output file
*.spg	Pattern Generator Program file
*.cfg	Viewer Configuration file
*.tfl	Filter File
*.tsf	Search File

Run an Example Analysis Project

1. Click **File** on the main menu bar and then choose **Open.**



2. Locate available example analysis projects by looking in: c:\program files\catalyst\sata1.xx\examples.
3. Choose an example *.stc file and click **Open** to display the example project dialog.

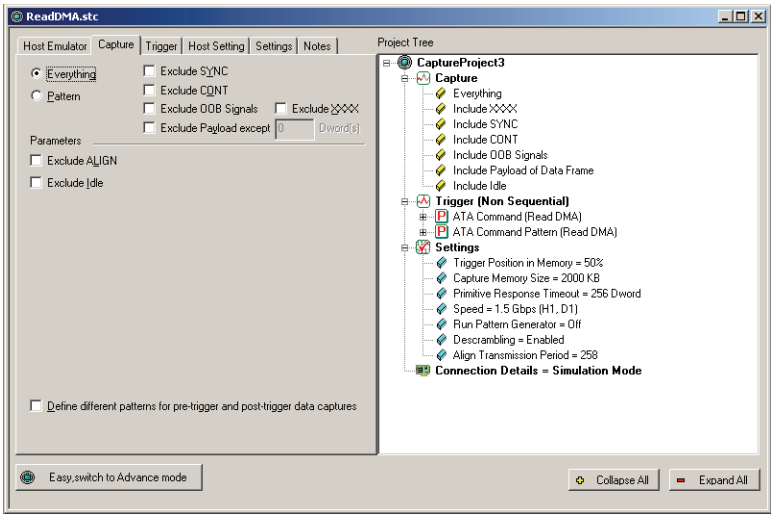


Figure 10. Example Protocol Analysis Project

4. Click the  **Run Hardware** button on the toolbar to execute the pre-defined example.
5. Once the project runs you will see an analyzer trace capture display similar to the one shown in Figure 11..

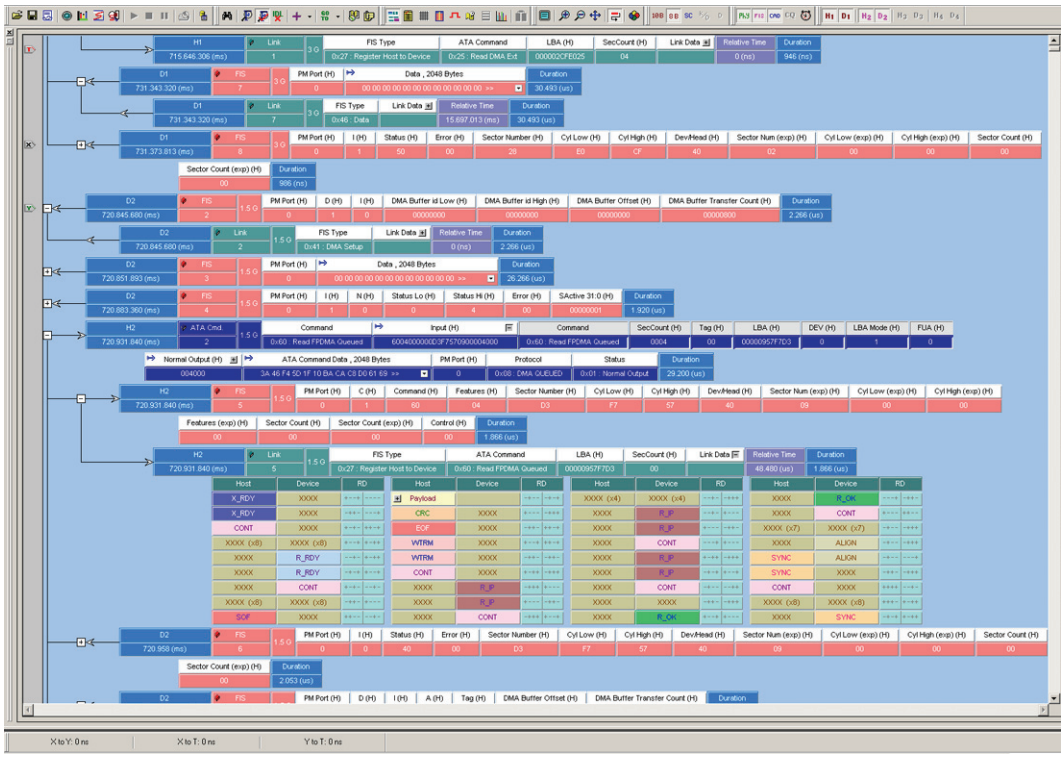


Figure 11. Analyzer Trace Capture Display

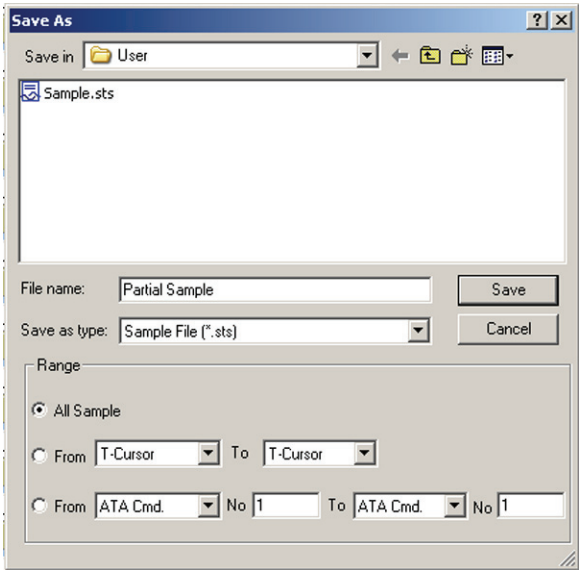
See section “Display Manipulation” on page 99 and section “Display Configuration” on page 133 for details about the results display.

Saving a Trace Capture

You may save a Trace Capture for review at a later time.

The **Save As** dialog offers you the ability to limit the range of the saved file.

You may save All Samples, a range between selected cursors or a range between selected commands.



Analysis Project Setup

You may define a new project starting with the default project definition, or modify the settings for the last project run.

Last Project Clicking the **Green** button opens the last project run. You may modify this project as required.

Click the **Green** button on the main menu bar to open the last project run dialog.

New Project To start a **New** project, click **File** on the main menu bar, choose **New**, and select "Protocol Analyzer".

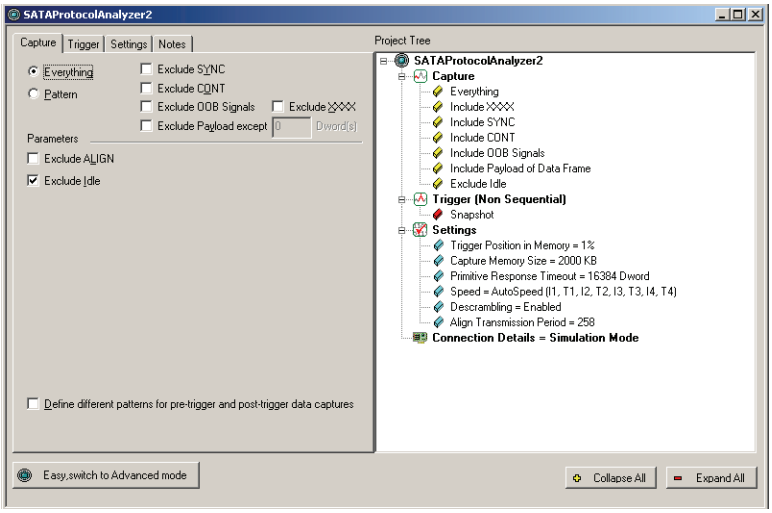


Figure 12. New Capture Project Dialog

Data Capture Setup

Capture Everything

The default Capture Tab opens with capture **Everything** selected and the corresponding default Trigger On tab with **Don't care (Snapshot)** selected. Clicking run with these default settings will immediately start a data capture to give the user a quick view of bus activity.

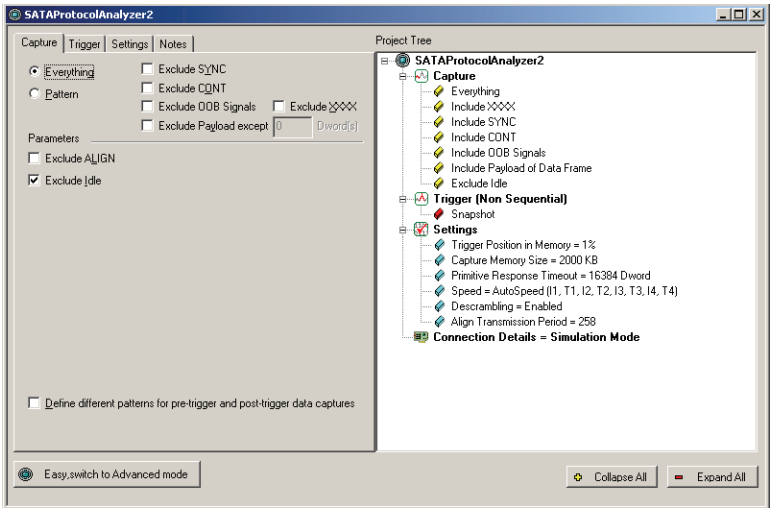


Figure 13. Capture Everything

Exclude SYNC Check this to exclude SYNC from the data capture.

Exclude OOB Signals Check this to exclude OOB Signals from the data capture.

Exclude CONT Check this to exclude CONT from the data capture.

Exclude XXXX Check this to exclude XXXX patterns from the data capture.

Exclude Payload Check this to exclude Payload from the data capture. (You may set except the first # of Dword(s)).

Exclude Align Check this to exclude Align Primitives from the data capture.

Exclude Idle Check this to exclude idles from the data capture.

Exclude patterns Check this box to allow for the capture of everything **except** the patterns that have been added to the Project Tree. When this box is checked, primitive category is added to the parameter window.

Pre and Post Trigger Capture You may define one set of patterns for capture prior to the occurrence of a trigger and another set of patterns for capture after the occurrence of a trigger. The setup of capture procedure is identical for both captures. To define different patterns for pre-trigger and post-trigger data capture check the **Define different patterns for pre-trigger and post-trigger data captures** check box to enable the Pre-Trig Capture tab.

Capture Patterns

To define specific patterns for capture, click the **Pattern** button.

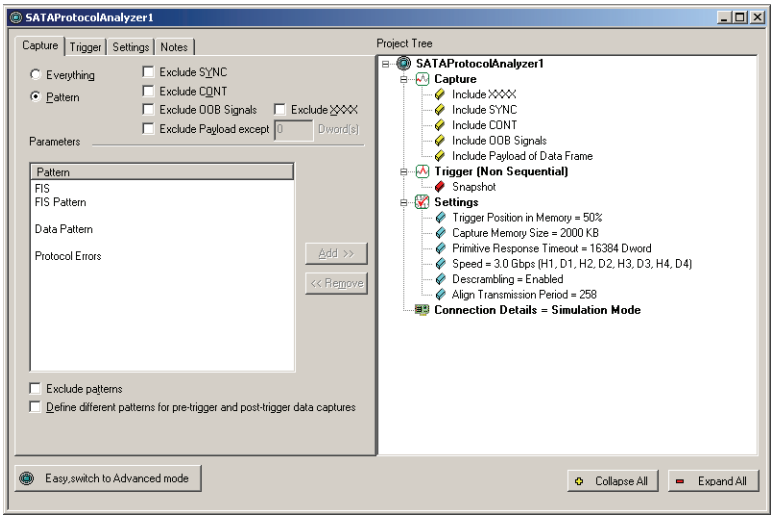


Figure 14. Choosing Capture Patterns

The **Parameters** window displays the following pattern capture choice categories:

- FIS
- FIS Pattern
- Data Pattern
- Protocol Errors

Choose a parameter To choose a parameter for capture from any of these categories, highlight the category in the parameter window and click the **Add>>** button. This will open selection dialogs for each of the categories displaying all of the parameters for that category. All of the patterns added will appear in the project tree.

Exclude SYNC Check this to exclude SYNC from the data capture.

Exclude OOB Signals Check this to exclude OOB Signals from the data capture.

Exclude CONT Check this to exclude CONT from the data capture.

Exclude XXXX Check this to exclude XXXX patterns from the data capture.

Exclude Payload Check this to exclude Payload from the data capture. (You may set except the first # of Dword(s))

Exclude patterns Check this box to allow for the capture of everything **except** the patterns that have been added to the Project Tree. When this box is checked, primitive category is added to the parameter window and Exclude Idle choice is enabled.

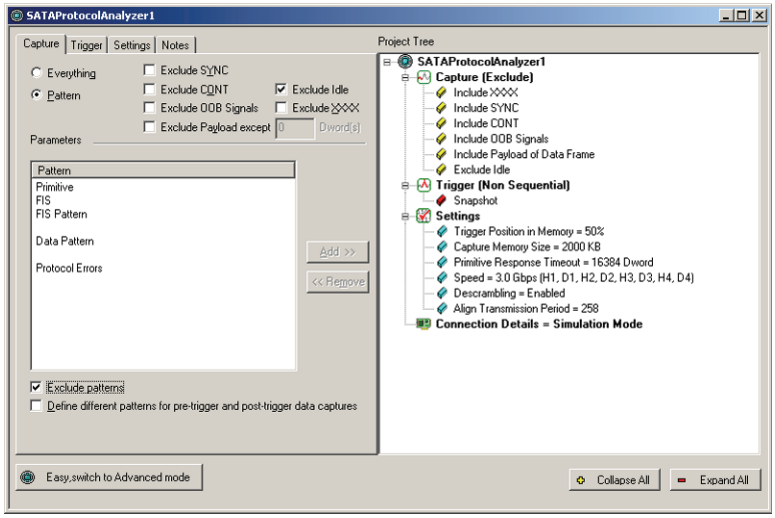


Figure 15. Exclude Patterns Checked

To remove an item from capture, highlight it in the Project tree and click the **<<Remove** button.

Pre and Post Trigger Data Capture

You may define one set of patterns for capture prior to the occurrence of a trigger and another set of patterns for capture after the occurrence of a trigger. The selections and setup procedure is the same for both, the Pre-Trigger capture and the Post-Trigger capture. Check Define different patterns for pre-trigger and post-trigger data capture to enable the Post-Trigger tab.

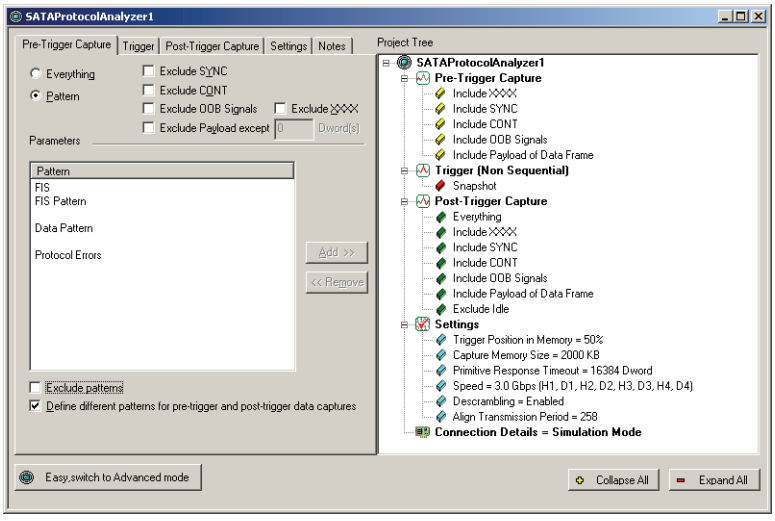
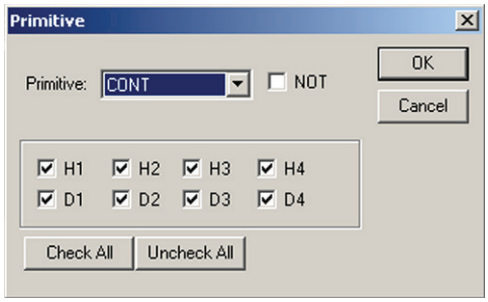


Figure 16. Post-trigger Capture Dialog Enabled

Defining Patterns

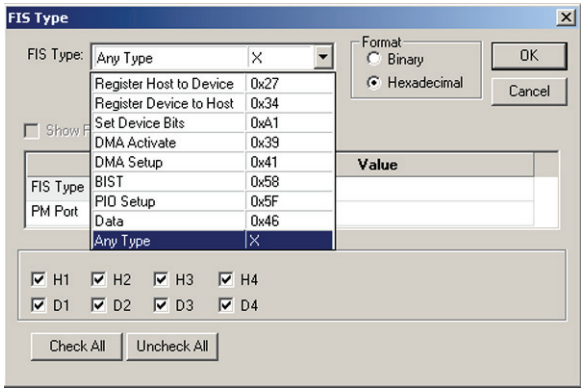
To select an item for capture, either highlight the category and click the **Add>>** button or double-click the category to open a corresponding definition dialog.

Primitive Double-click **Primitive** (Available only if Exclude Patterns is checked) to open the Primitive selection dialog.



Select the desired primitive and click **OK**.

FIS (Frame Information Structure) Double-click **FIS** to open the FIS Type selection dialog.



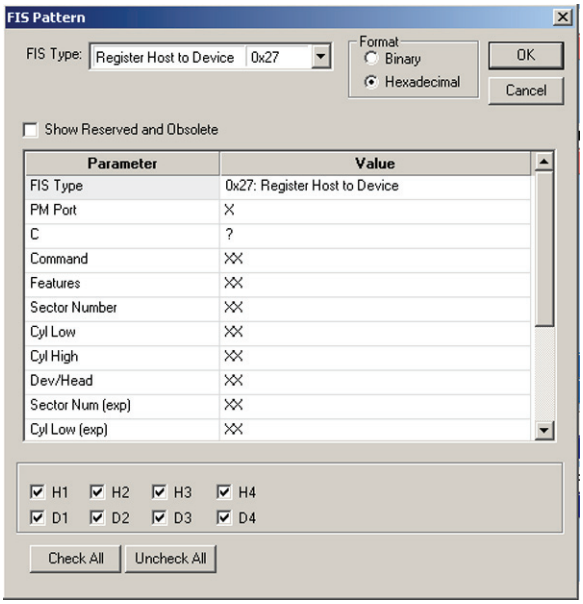
Click the down arrow next to the Type drop-down list box, choose a FIS type to capture and click OK. Repeat for additional types.

Available FIS Types:

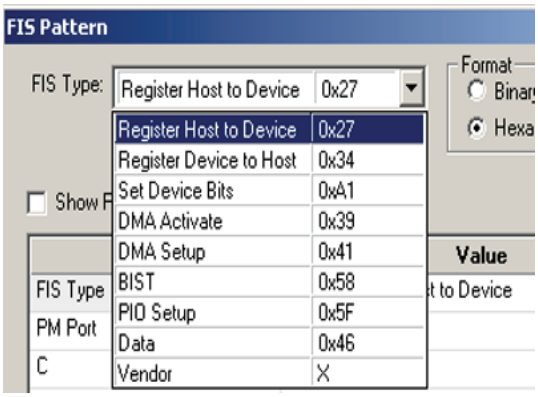
- Register Host to Device
- Register Device to Host
- Set Device Bit
- DMA Activate
- DMA Setup
- BIST
- PIO Setup
- Data
- Any Type

FIS Pattern

Double-click **FIS Pattern** to open the FIS Pattern selection dialog.



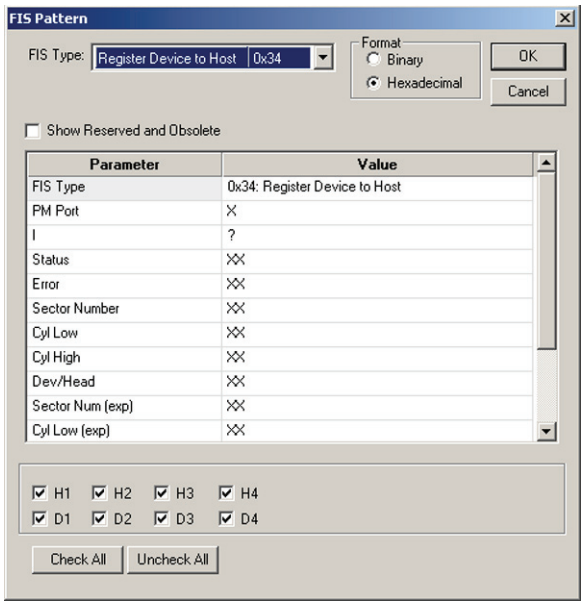
The FIS Pattern dialog opens with the default FIS Type as **Register Host to device.** To choose another available FIS Type click the down arrow next to the FIS Type list box.



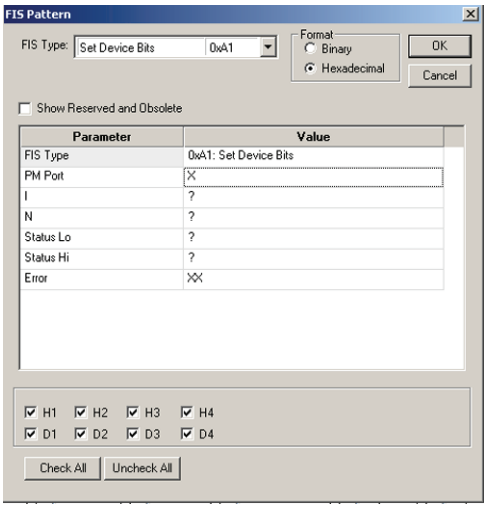
Choose the desired FIS Type and complete the corresponding dialog.

FIS Types

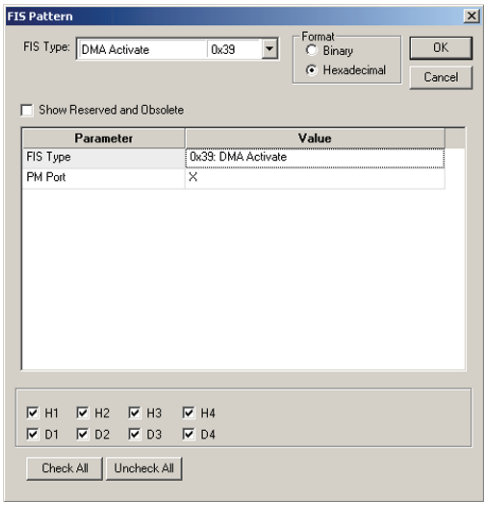
Register Device to Host



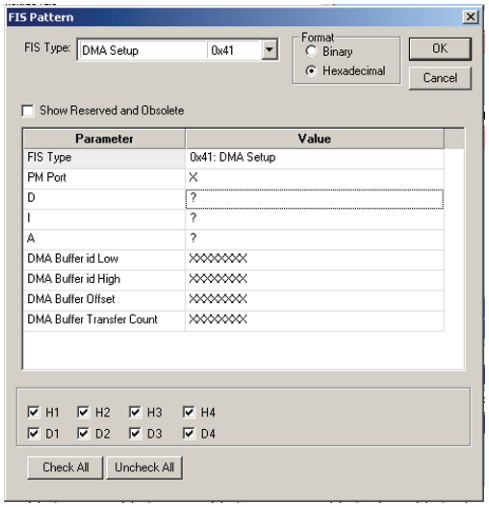
Set Device Bits



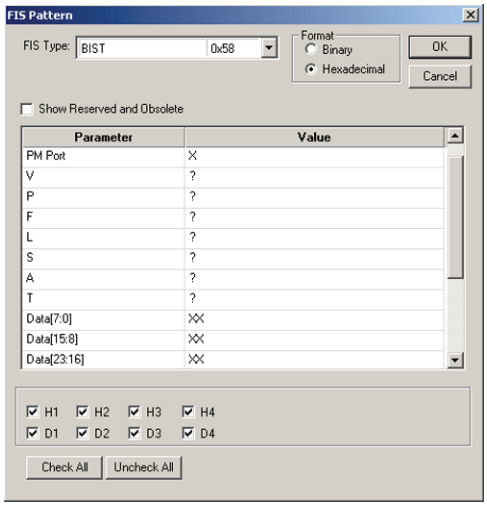
DMA Activate



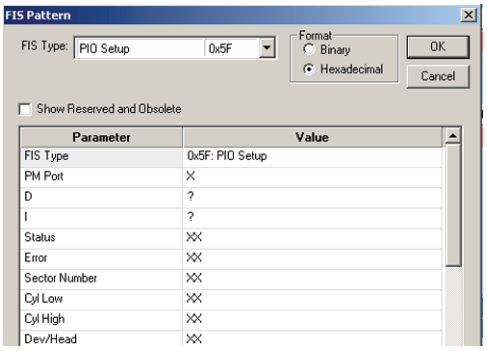
DMA Setup



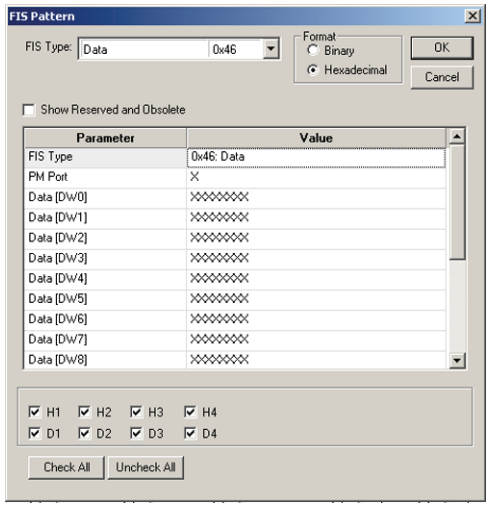
BIST



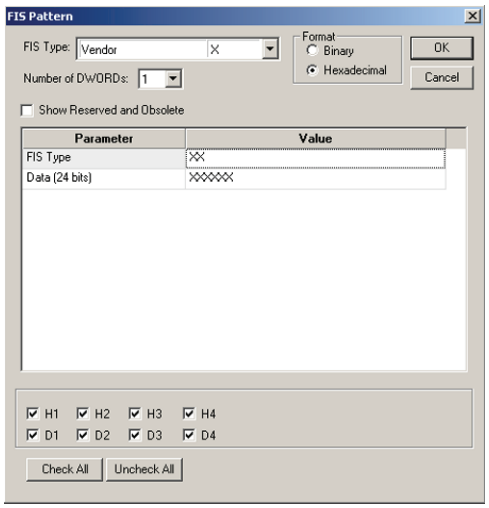
PIO Setup



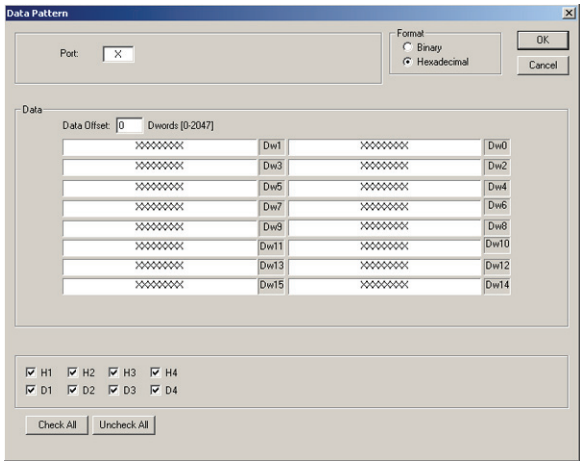
Data



Vendor



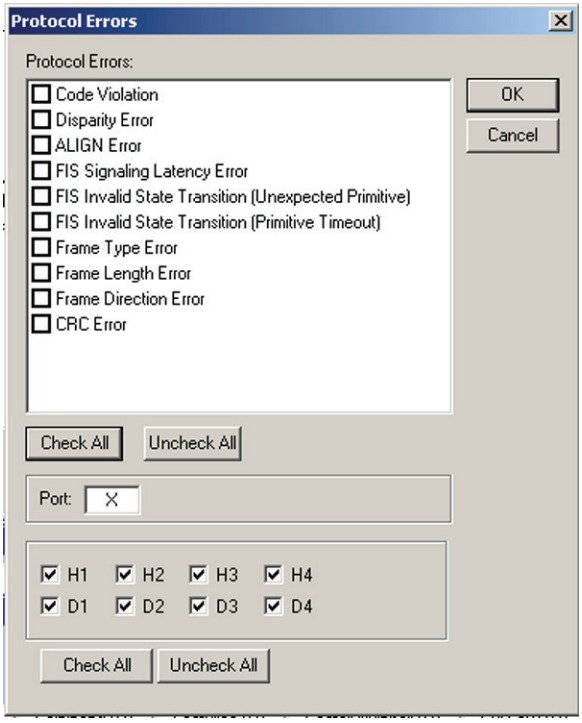
Data Pattern Double-click Data Pattern to open the Data Pattern definition dialog.



Enter the data pattern(s) for capture and click **OK**.

Protocol Errors

Double-click Protocol Errors to open the Protocol Errors selection dialog.



Check the desired Protocol Error(s) for capture and click **OK**.

Triggering Setup

The **Trigger** tab, in the analysis project dialog, allows the user to specify when the analyzer will complete a data capture. Three trigger modes are available. A default **Don't Care (Snapshot), Manual Trig** and **Pattern.**

When a data capture is started with **Don't Care (Snapshot)** selected, the analyzer will trigger on the first data sample collected. Starting a data capture with **Pattern** selected the analyzer will trigger when specific pattern(s) are detected in the captured data stream. The following are three ways to trigger the analyzer with **Pattern** selected.

- Trigger on any pattern (Any Trigger Mode)
- External
- Trigger on a sequence of patterns (Sequential Trigger Mode)

Snapshot Mode

To trigger immediately on any pattern, check the **Don't care (Snapshot)** button.

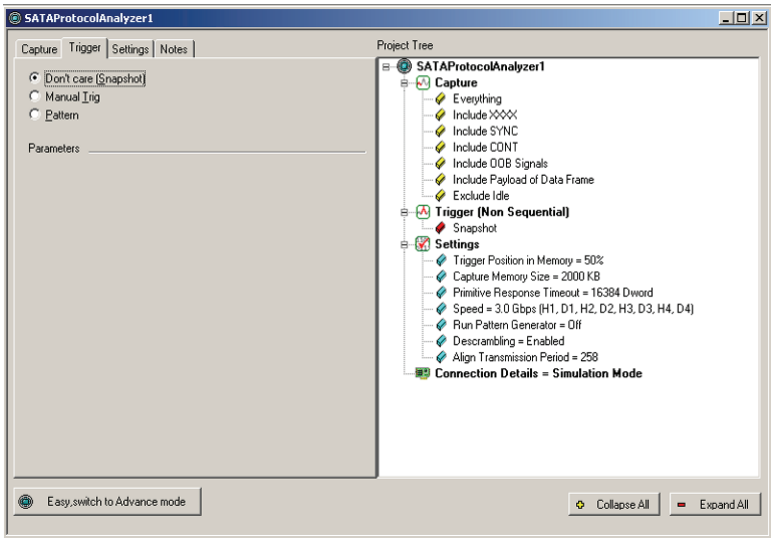


Figure 17. Default Trigger Selected

Manual Trigger Mode

In the **Manual Trigger** mode the analyzer captures bus traffic continually until you click the **Stop Hardware** button on the analyzer toolbar which triggers the analyzer. To perform a manual trigger, check the **Manual Trig** option button.

Any Trigger Mode

In the **Any Trigger** mode, triggering occurs whenever any of the patterns selected are detected. The procedure for selecting trigger parameters is identical to that for selecting capture parameters. Many of the pattern selections are also identical.

To define patterns for triggering check the **Pattern** button in the Trigger dialog.

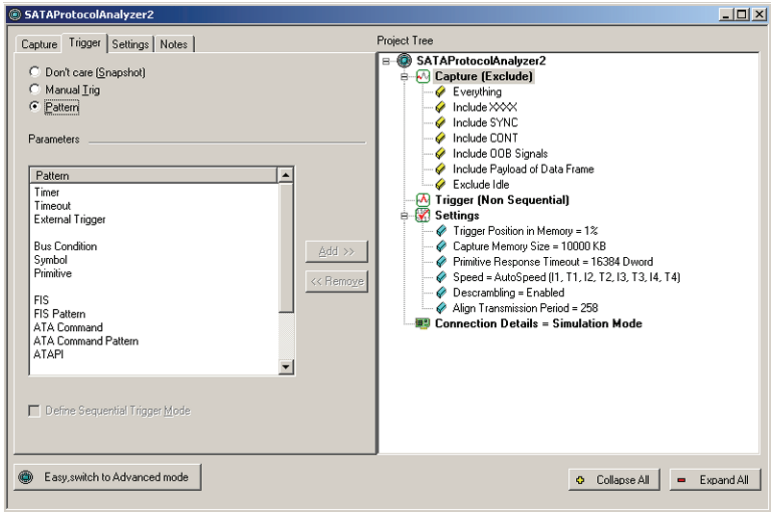


Figure 18. Select Patterns for Trigger

The **Parameters** window displays the following trigger pattern choice categories:

- Timer
- Timeout
- External Trigger
- Bus Condition
- Symbol
- Primitive
- FIS
- FIS Pattern
- ATA Command
- ATA Command Pattern
- ATAPI
- Soft Reset
- Data Pattern
- Protocol Errors

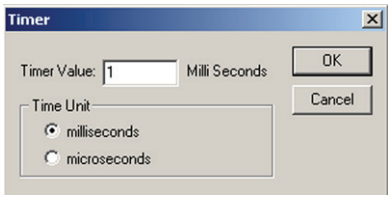
Choose a parameter

Either highlight the category and click the **Add>>** button or simply double-click the category to open a corresponding definition dialog.

To remove an item from trigger, highlight it in the Project tree and click the **<<Remove** button.

Timer

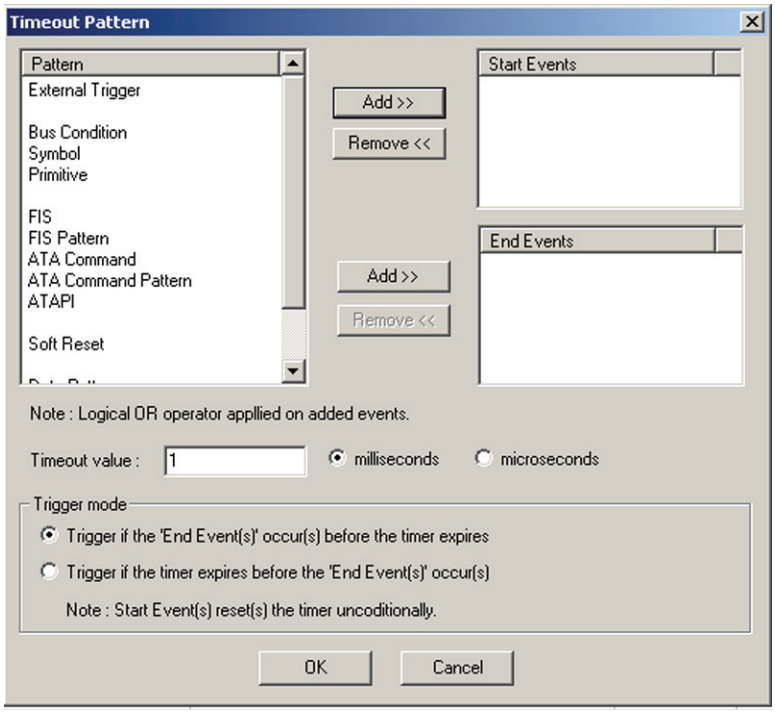
Selecting a timer for a trigger in the **Any Trigger Mode** limits the time that the analyzer looks for selected triggering conditions before triggering. The timer is activated when the Project is run. If none of the other selected triggering conditions occurs during the timers active time the Analyzer will trigger at the end of the time set for the timer.
A timer may be set independent of any other trigger condition. This unconditionally triggers the analyzer when the set time expires.
To set the timer value, double-click **Timer** in the Patterns window of the Capture Project dialog to open the Timer dialog.



Check the Time unit desired and enter a Timer Value and click **OK.**

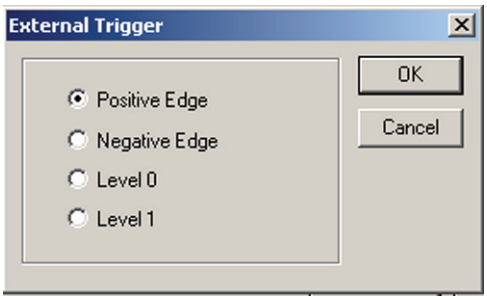
Timeout

Choosing timeout as a trigger condition allows you to define a timer with a timeout value that is reset to 0 and started by the occurrence of any events that you add to the Start Events list. You may then add one or more events to the End Events list and then choose a trigger to occur if an End event occurs before the timer expires or if the timer expires before the occurrence of an end event. The Start and End events are added and defined identically to the way the patterns to be captured are defined and added.



External Trigger

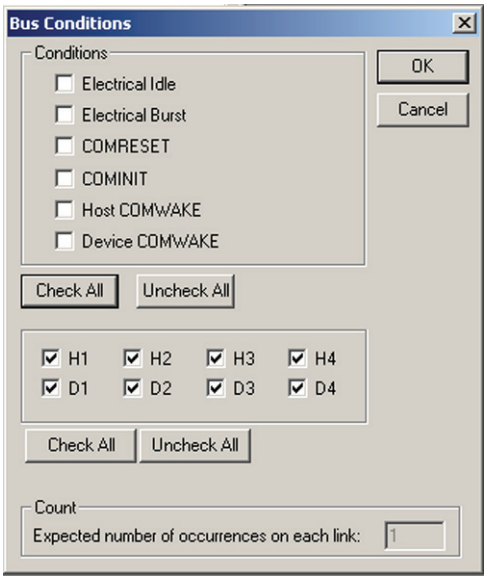
You may trigger on an external trigger which is available for both, the Combinational and Sequential triggering modes. To set up the trigger click the **External Trigger** category.



Check a desired triggering condition button and click **OK**.

Bus Condition

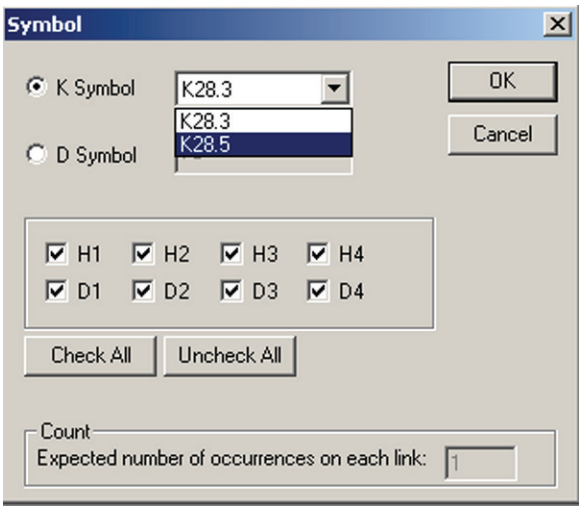
Double-click **Bus Condition** to open the Bus Conditions selection dialog.



Check the desired Conditions to trigger on and click **OK**.

Symbol

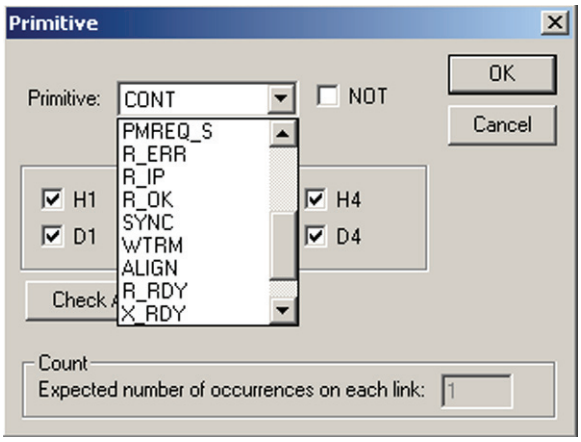
Double-click **Symbol** to open the Symbol selection dialog.



Click the down arrow next to the K Symbol dropdown list, choose a symbol to trigger on and click **OK**. To choose a D symbol click the D symbol option button and enter a Hex value for the symbol.

Primitive

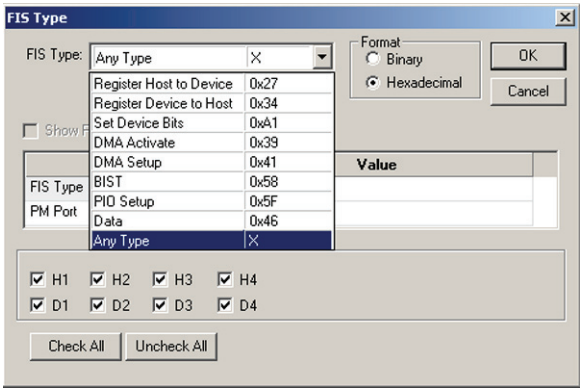
Double-click **Primitive** in the Patterns window of the Trigger dialog to open the Primitive dialog.



Click the down arrow next to the Primitive dropdown list, choose a primitive to trigger on and click **OK.** Note: Checking the box to the right of the Primitive Type allows use of the logical NOT function.

FIS (Frame Information Structure)

Double-click FIS to open the FIS Type selection dialog.



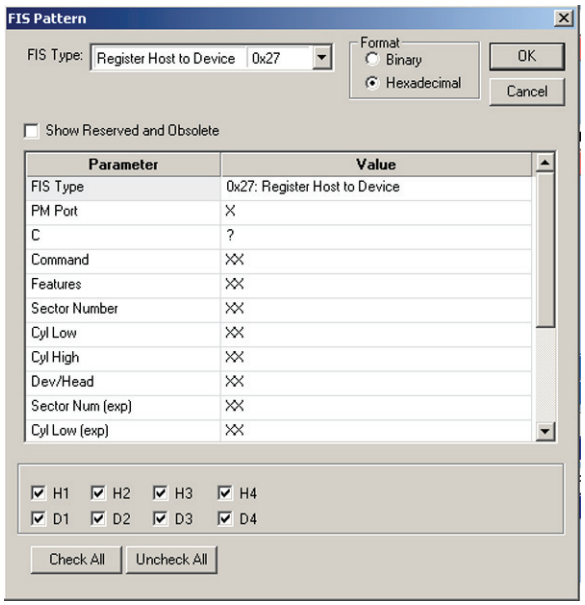
Click the down arrow next to the Type drop-down list box, choose a FIS type to trigger on and click OK. Repeat for additional types.

Available FIS Types:

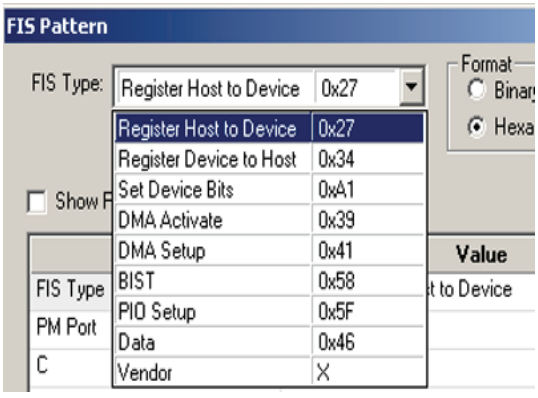
- Register Host to Device
- Register Device to Host
- Set Device Bit
- DMA Activate
- Any Type
- DMA Setup
- BIST
- PIO Setup
- Data

FIS Pattern

Double-click FIS Pattern to open the FIS Pattern selection dialog.



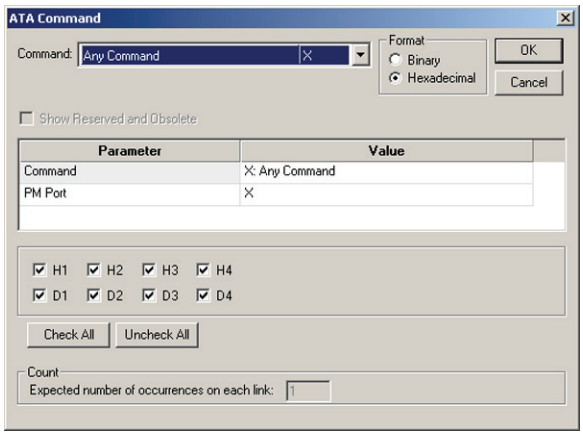
The FIS Pattern dialog opens with the default FIS Type as **Register Host to device.** To choose another available FIS Type click the down arrow next to the FIS Type list box.



Choose the desired FIS Type and complete the corresponding dialog.

ATA Command

Double-click **ATA Command** to open the ATA command selection dialog.

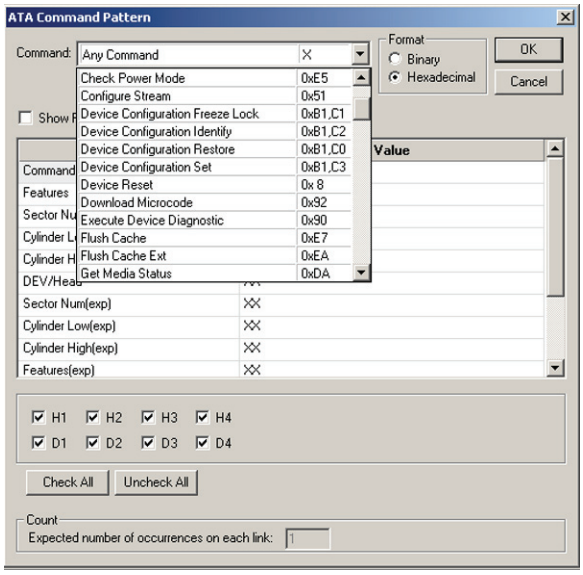


Click the down arrow next to the Command list box, choose an ATA command and click **OK**. A powerful triggering choice is **Any Command** that will cause the analyzer to trigger on any ATA command.

Note: The command code and feature set are not the only parameters that describe an ATA command. If parameters such as LBA and sector count are required, you must use the **ATA Command Pattern** dialog.

ATA Command Pattern

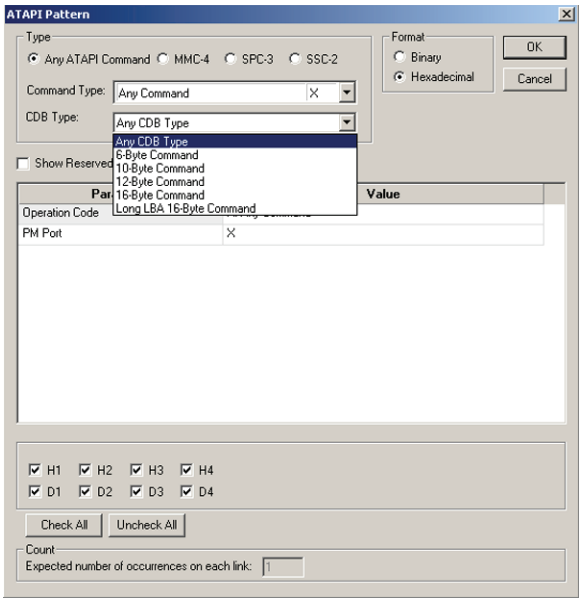
Double-click **ATA Command** Pattern to open the ATA command pattern selection dialog.



Click the down arrow next to the Command list box, choose an ATA command and click **OK**.

ATAPI Pattern

Double-click **ATAPI** to open the ATAPI Pattern dialog.

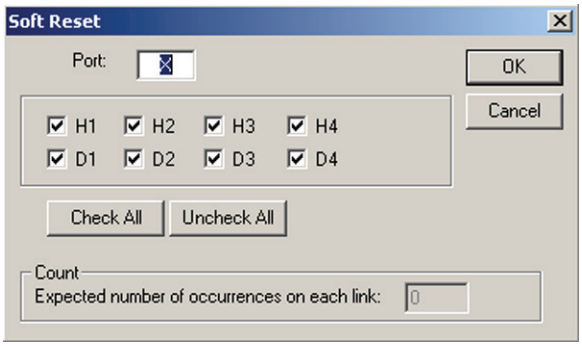


Select the pattern **Type** by checking the corresponding option button.

Soft Reset

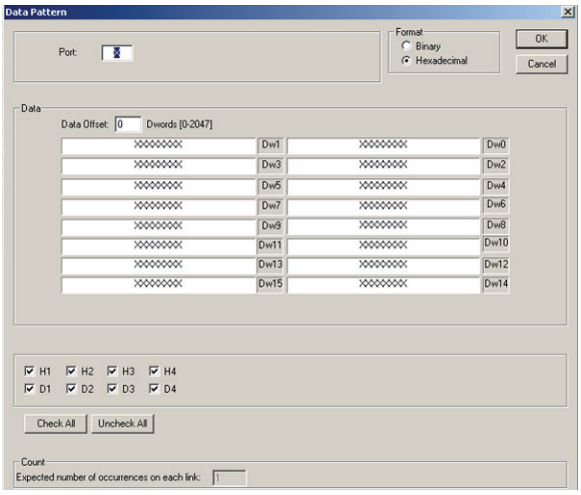
Double-click **Soft Reset** to open the Soft Reset dialog

.



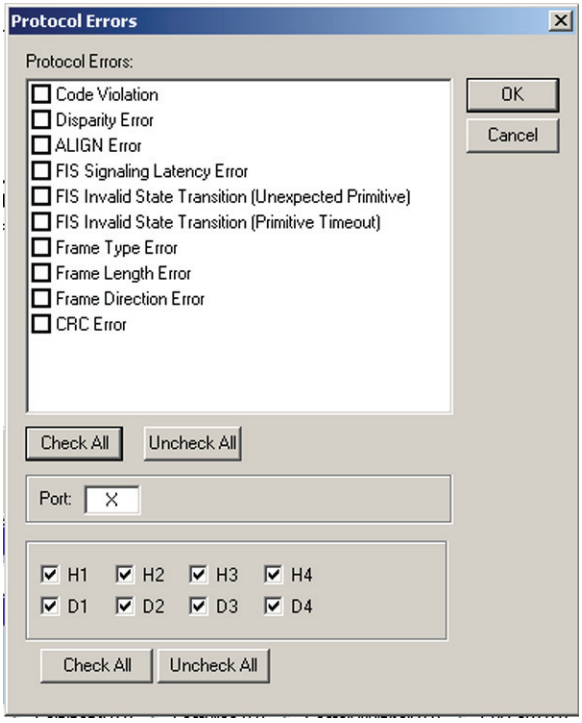
Data Pattern

Double-click **Data Pattern** to open the Data Pattern dialog.



Enter the desired patterns and click **OK.**Protocol Errors

Double-click **Protocol Errors** to open the Protocol Errors dialog.



Check the protocol error(s) that you wish to trigger on and click **OK**.

Port and Direction You may define the port and direction for each trigger by checking or unchecking the corresponding H and/or D check boxes.

Sequential Trigger Mode

In the Sequential Trigger mode, triggering occurs whenever a specific sequence of patterns are detected. The sequence is established by the order in which the triggering patterns are defined.

Note: Patterns such as Primitives and Symbols or Frames occurring very close together on different ports will cause an error in triggering.

To define a triggering sequence, check the **Define Sequential Trigger Mode** and enter triggering patterns in the sequence that you would like to trigger.

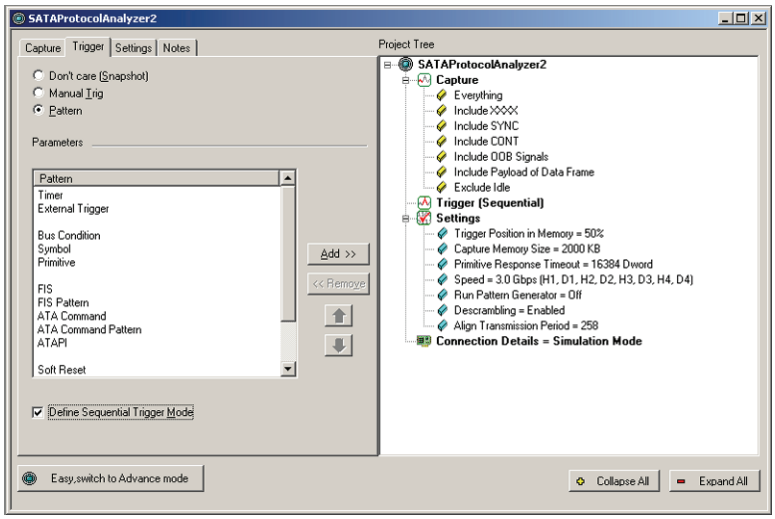
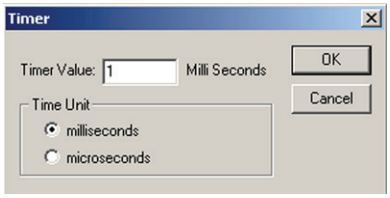


Figure 19. Select Sequential Trigger Mode

Timer

The sequential triggering mode offers the option of triggering on a timer or inserting a timer in the triggering sequence to delay detection of the next pattern in the sequence. To insert a timer in the trigger list Double-click **Timer** to open the timer definition dialog.



Enter the desired **Time Value**, choose the **Time Unit** and click **OK**.

Defining Patterns

The definition of patterns for the sequential trigger mode is identical to the Any Trigger mode with the following exception: the definition dialogs for triggering patterns have an additional setting, to count the number of occurrences. This setting allows the user to specify the number of times that the pattern must occur before triggering or proceeding in the trigger sequence.

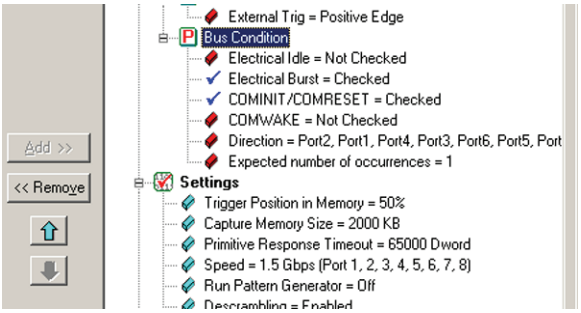
Figure 20. Number of Occurrences

Note: The events on each link are counted independently causing a trigger whenever the number of occurrences on any link equals the specified value.

Triggering order

As triggering patterns are defined and added they are displayed in the Project Tree sequentially in the order that they were entered under the Trigger On category. When the project is run, the analyzer will detect the occurrence of each pattern in order and trigger on the last one.

The sequence of triggering patterns can be re ordered if desired. To change the sequence order, highlight a trigger pattern and use the Up or Down arrow to move it to a new position.



Project Settings

To set project options click the **Settings** tab.

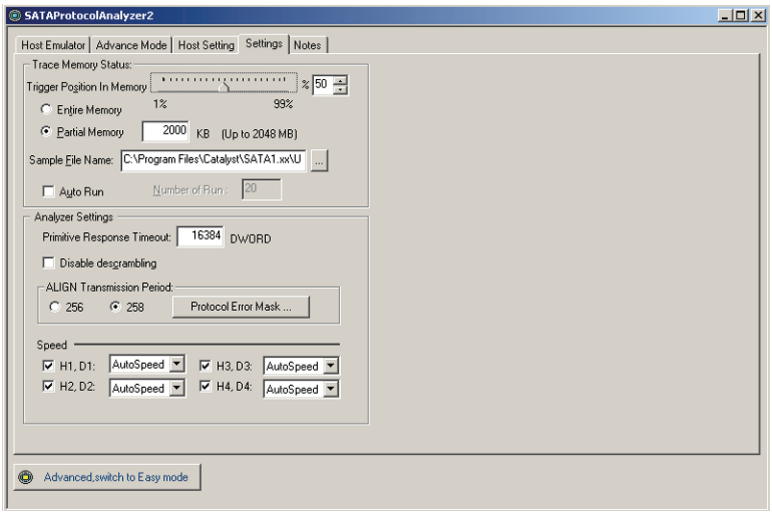


Figure 21. Setting Project Options

Memory Size To reduce the capture memory size, check Partial Memory and enter the desired buffer size if you want to partition the trace memory for multiple captures or, check entire memory to allow capture for the entire memory if you want to capture the maximum amount of trace data.

Note: In cases where the size of a data packet exceeds the set buffer memory allocation, the project will run, but no capture will result. In such cases you must increase the buffer memory size to a value greater than the packet size.

Trigger Position Pre-Trigger is set by default at 50% which defines the percentage of data to be captured before and after the triggering event. You may change this percentage by dragging the slider to the desired value.

The capture of the specified percentage of the data prior to the triggering event cannot be guaranteed and may in some cases be 0. This can occur in cases where the triggering event occurs before the required number of pre-trigger event data can be stored. In these cases the data display will show fewer than the specified data points prior to the triggering event. For more detail see “Trigger Position” on page 47.

Sample File Name Click the ellipses next to the Sample File Name text box and choose a file name and location for the results of your current project.

Align Transmission period Click the desired option button.

Auto Run To repeat the current capture and trigger setup automatically, check the **Auto Run** checkbox and enter the number of times in the **Number to Run** text box. The capture and trigger will repeat automatically for the specified number of times and the results saved in consecutively numbered **Sample.sts** files.

Disable descrambling Check this option to view scrambled data.

Primitive Response Timeout Specifies timeout value to detect a primitive response timeout error.

Choose Port Speed Click the down arrow next to the port speed box and choose a port speed. The default is Autospeed.

Note: If a Port check box is unchecked the analyzer will not capture any patterns for that Port. The trace memory for that port will be allocated to it's adjacent Port. e.g. H1, D1 <-> H2, D2 or H3, D3 <-> H4, D4.

Protocol Errors Click the **Protocol Error Mask…**button and check any or all of the protocol errors to be excluded from the capture.

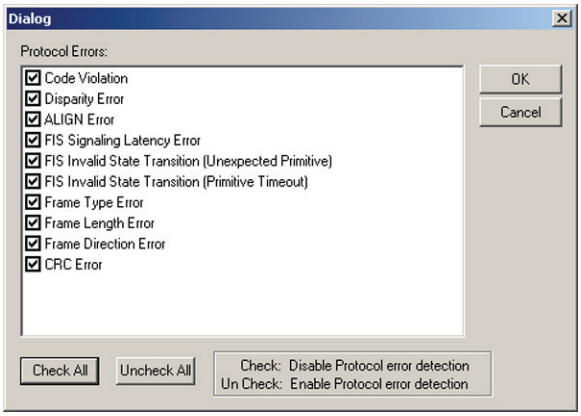
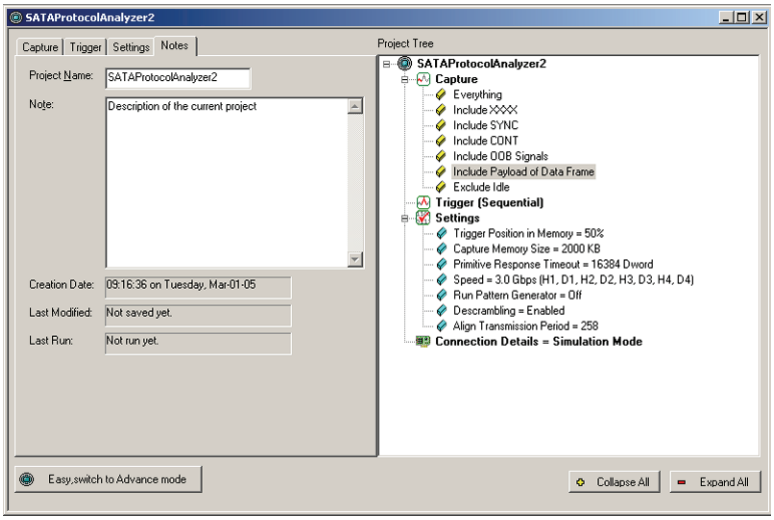


Figure 22. Protocol Error Mask

Project Note Click the **Notes** tab on the project dialog to open the Notes dialog. Enter any relevant information about the project in the Note area.



Trigger Position

The amount of data to be captured before and after the trigger point may be set as a percentage of pre-trigger in trace memory, between 1% and 99%. This may be done by positioning the pre-trigger slider to the desired percentage. This feature allows the evaluation of bus activity leading up to and after the triggering event. The operation of the pre-trigger in the trace memory is conceptually illustrated in Figure 23.

Pre-trigger Data is the capture of the specified percentage of the data prior to the triggering event. The amount of data captured cannot be guaranteed and may in some cases be 0. This can occur in cases where the triggering event occurs before the pre-trigger data can be stored. In these cases the data display will show fewer than the specified data points prior to the triggering event.

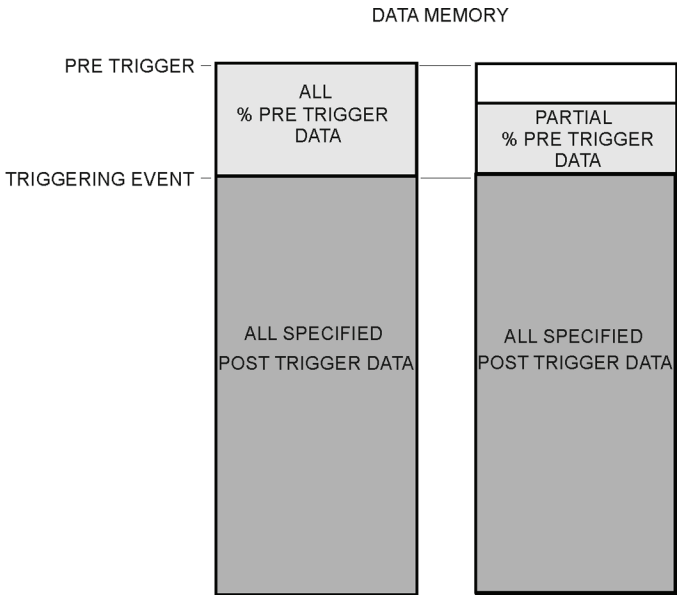


Figure 23. Pre-Trigger Example, 20% Pre-Trigger

Advanced Mode (User Defined)

This mode expands your Analysis capability by allowing you to program complex triggering and data capture projects.

The Advanced Mode is implemented as a state machine with up to 23 different states. Each state may be individually programmed to:

- Trigger on a different event or trigger unconditionally.
- Capture Everything, Nothing or a user defined pattern.
- Include up to 3 ELSE IF statements allowing a jump to any other state based on user definition.
- Use up to 3 timers that can be set to a maximum value of 42949 ms. A timer may be set in the state or continue the one set in the previous state.
- Output an external trigger High or Low.

Working in the Advanced Mode

To start working in the Advanced Mode, click the **Easy, Switch to Advanced Mode** button in an open Analyzer window.

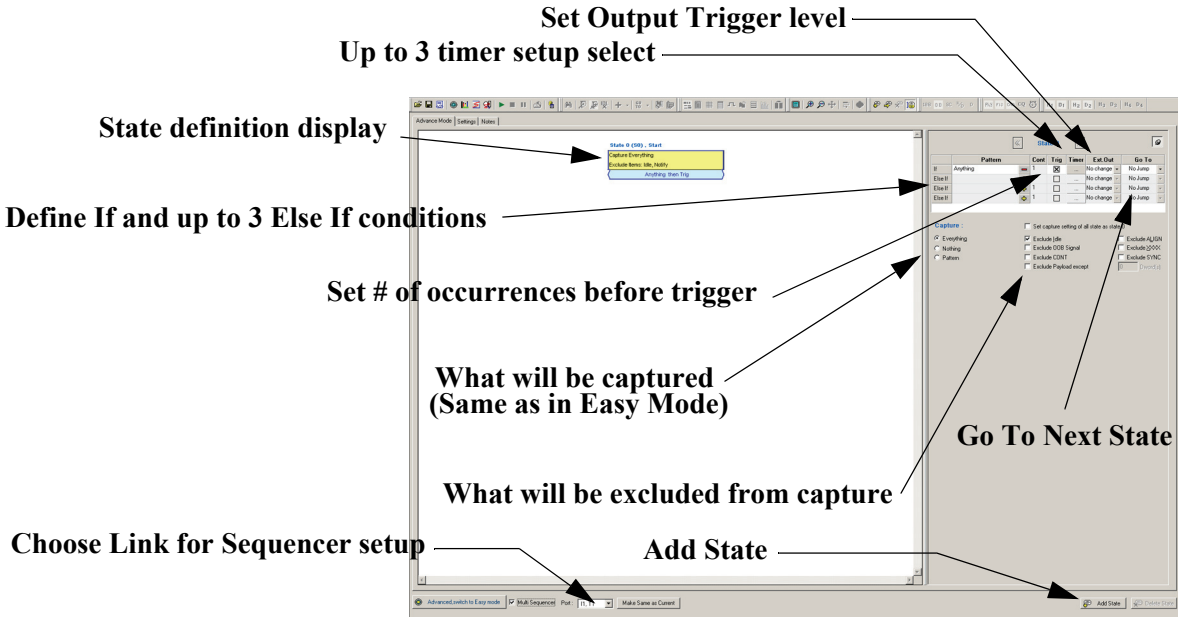


Figure 24. State Programming Dialog

Setting Trigger Conditions

To set the If and Else If trigger condition:

1. Click in the corresponding Pattern Field and choose a trigger condition from the open drop down list.

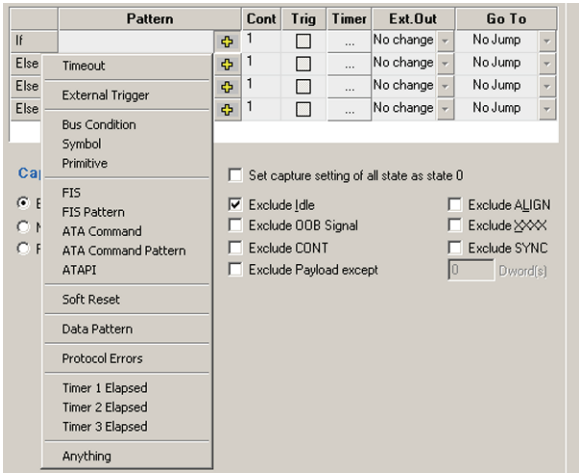
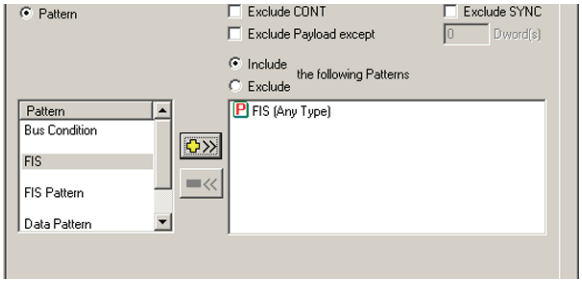


Figure 25. Choosing a Trigger Condition

2. Define each selected pattern in the same way as in the Easy Mode as described starting on page 17. You may set a timer for any if or elseif condition.
3. Choose a capture option: **Everything, Nothing** or **Pattern**.
4. If you choose Pattern, you may select patterns for inclusion or exclusion. Clicking the **Pattern** option enables a pattern definition dialog.



5. Choose a pattern(s) and click the +>> button to add it for capture or exclusion. Each pattern selected is defined in the same way as in Easy mode. (see "Defining Patterns" on page 26.)
6. If an output trigger is required, click the Combo Box arrow in the **Ext. Out** field and select the desired output trigger level.
7. To go to another state, click the down arrow in the **Go To** field and select a state to go to next. If no other state has been defined choose **New State** to add a state to goto.

Multi - Link Triggering

You may set up different triggering for each link. To set up different trigger conditions for a link, check the Multi Sequencer check box and select the link for setup from the Port drop down list.

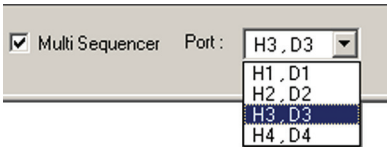


Figure 26. Multi - Link Triggering Setup

Set Timers

You may set and use up to 3 timers for triggering. Each timer may be set for each state or set to continue from one set in the previous state. You may set a timer for any IF or ELSE IF condition. To set up the timers, click the ellipses next to the IF or ELSE IF condition and define each of the timers in the Set Timers dialog.

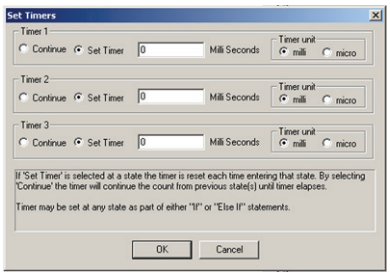


Figure 27. Set Timers Dialog

Useful Key Sequences

The following key sequences are active to assist you in navigating a defined state machine:

Ctrl+a	Add State
Insert	Insert State
DEL	Delete State
Ctrl+c/Ctrl+Ins	Copy
Ctrl+v/Shift+Ins	Paste
Up/Down arrow keys	Moves selection between states
Page Up/Page Down	Page Up and Page Down states
Home	Go to first page
End	Go to end page

Project Settings

Prior to running the Advanced mode project, click the **Settings** tab. The options in the Settings dialog are the same as for the Easy Mode and are described starting on page 45.

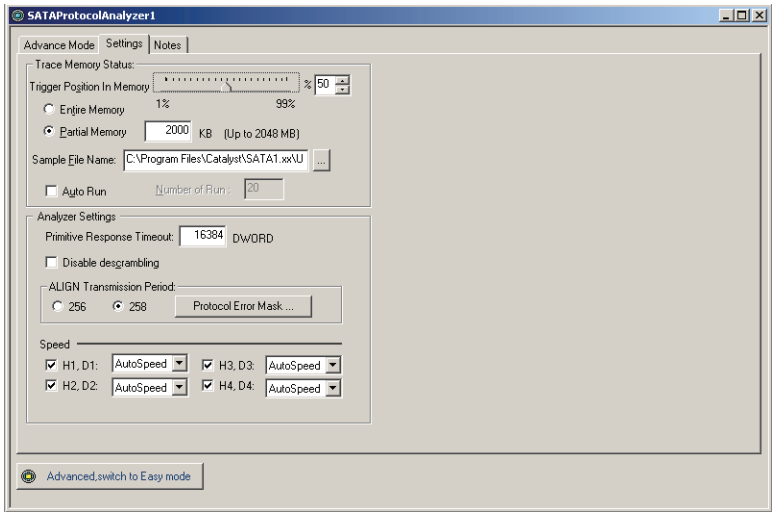


Figure 28. Project Settings Page

Memory Size To reduce the capture memory size, check Partial Memory and enter the desired buffer size if you want to partition the trace memory for multiple captures or, check entire memory to allow capture for the entire memory if you want to capture the maximum amount of trace data.

Note: In cases where the size of a data packet exceeds the set buffer memory allocation, the project will run, but no capture will result. In such cases you must increase the buffer memory size to a value greater than the packet size.

Trigger Position Pre-Trigger is set by default at 50% which defines the percentage of data to be captured before and after the triggering event. You may change this percentage by dragging the slider to the desired value.

The capture of the specified percentage of the data prior to the triggering event cannot be guaranteed and may in some cases be 0. This can occur in cases where the triggering event occurs before the required number of pre-trigger event data can be stored. In these cases the data display will show fewer than the specified data points prior to the triggering event. For more detail see “Trigger Position” on page 47.

Sample File Name Click the ellipses next to the Sample File Name text box and choose a file name and location for the results of your current project.

Auto Run To repeat the current capture and trigger setup automatically, check the **Auto Run** checkbox and enter the number of times in the **Number to Run** text box. The capture and trigger will repeat automatically for the specified number of times and the results saved in consecutively numbered **Out.sts** files.

Primitive Response Timeout Specifies timeout value to detect a primitive response timeout error.

Disable descrambling Check this option to view scrambled data.

Align Transmission period Click the desired option button.

Protocol Errors Click the **Protocol Error Mask…**button and check any or all of the protocol errors to be excluded.

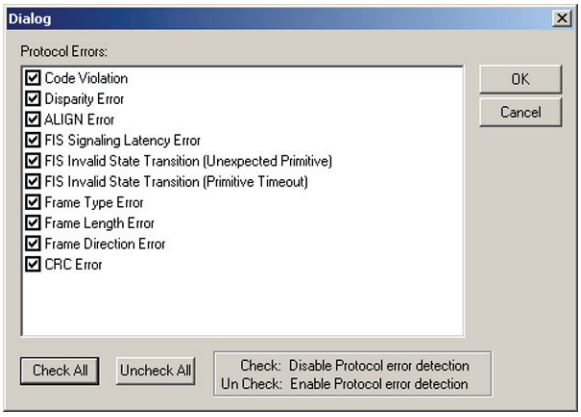


Figure 29. Protocol Error Mask

Choose Port Speed Click the down arrow next to the port speed box and choose a port speed.

Note: If a Port check box is unchecked the analyzer will not capture any patterns for that Port. The trace memory for that port will be allocated to it's adjacent Port. e.g. H1, D1 <-> H2, D2 or H3, D3 <-> H4, D4.

Project Note To include some descriptive information about the project, click on the **Notes** tab and enter a brief descriptive note about the project.

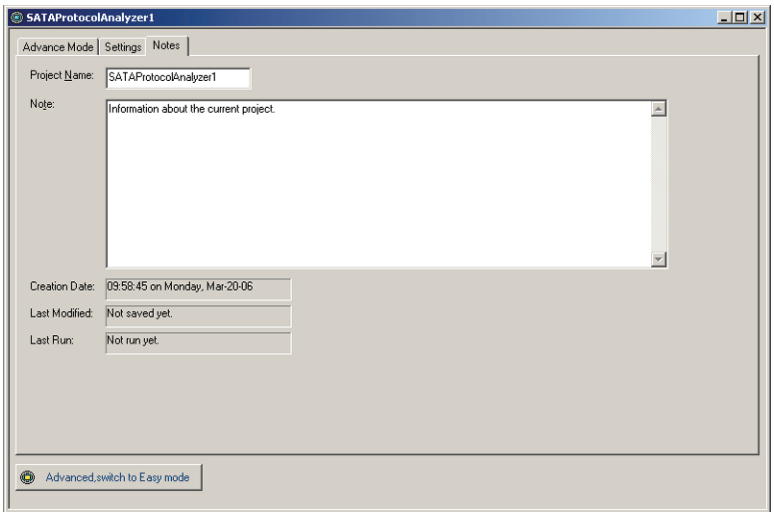
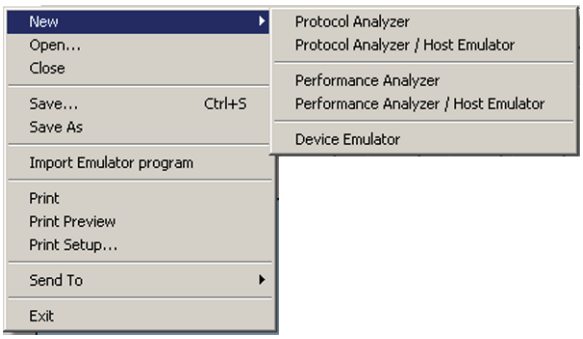


Figure 30. Project Note

Exercise and Capture

Not in current version for STX-460.

To perform a capture with Host Emulator generated bus traffic, click **File, New** and choose **Protocol Analyzer/Host Emulator.** Program the Host Emulator and then set up a capture as described in **"Protocol Analysis" on page 17**.



Programming the Host Emulator

Click the Host Emulator tab to open the Host Emulator program dialog.

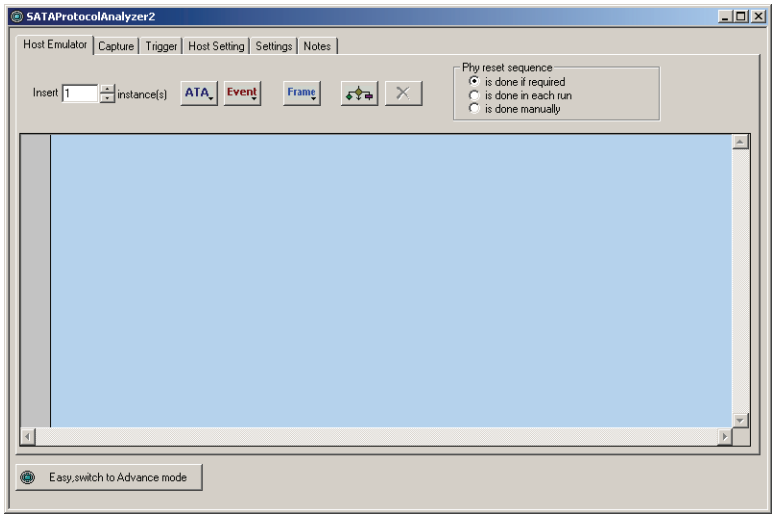
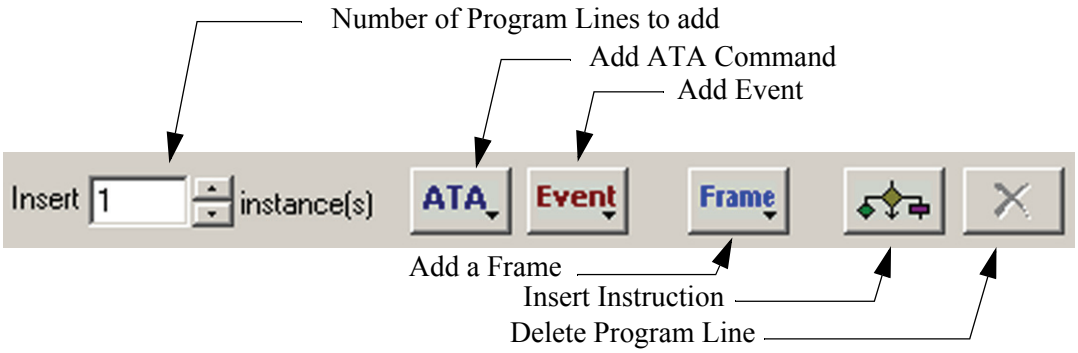


Figure 31. Host Emulator Program Dialog

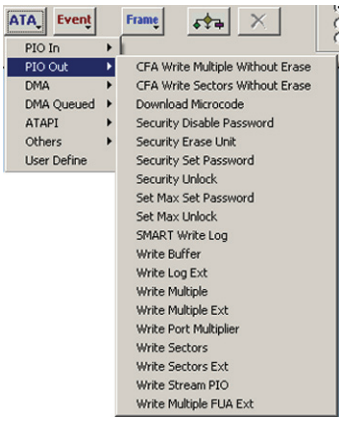
A Host Emulator program may be created using ATA commands and events. These commands may be executed in a program loop or subject to user specified conditions.



Add Program Lines To add program lines, enter the number of lines to be added and click the down arrow on the desired command button.

Adding Host Emulator Commands

Adding an ATA Command

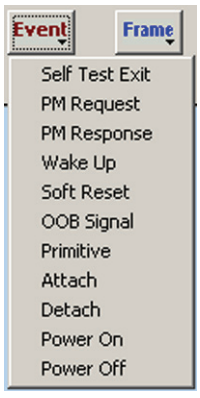


Click the down arrow on the **ATA Command button,** click on one of the 6 command categories and choose a command or select the User define command.

Note: You may send a SCSI command over the SATA interface by choosing **ATAPI** and subsequent options.

Adding an Event

Click the down arrow on the **Event** button and choose the event to be inserted.



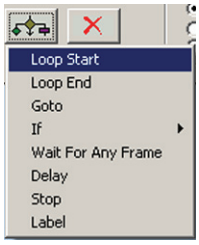
Adding a Frame

Click the down arrow on the **Frame** button and choose the frame to be inserted.

Inserting Instructions

Instructions are logical program elements that allow the definition of how the Host Emulator program is executed. Using instructions you may define program loops, make conditional jumps, insert delays and stops.

To insert an instruction, click on a command in the Host Emulator program where you want the instruction inserted and then click the **Insert Instruction** button and choose the instruction to insert.



Start Loop Click on the command where you would like to start the loop and then insert the **Loop Start** instruction.

Enter the number of times to run the loop in the number text box or click the down arrow on the Count Drop-down combo box and choose **Infinite.**

End Loop Click on the command where you would like to stop the loop and then insert the **Loop End** instruction.

Add a Label Add a label to any command such that it may be used for conditional and unconditional jumps. Click on the command you wish to label and insert the **Label** instruction.

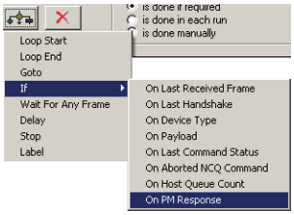
Labels are automatically labeled as Label# sequentially as they are added, however, you may assign them meaningful names such as STOP and START.

Add a Goto

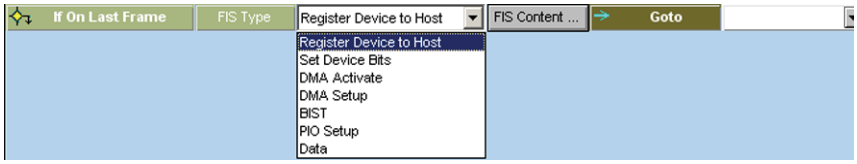
To insert an unconditional jump to a previously labeled command, insert a **Goto** instruction. Then click the down arrow on the Drop-down combo box and choose the label to designate the destination command.

Add an If

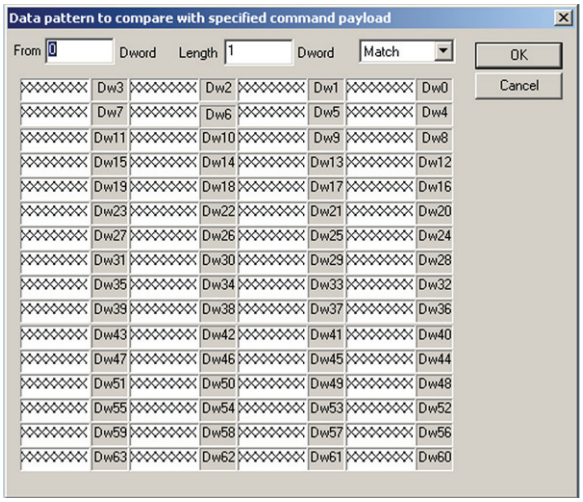
To insert a conditional jump to a previously labeled command, insert an **If** instruction.



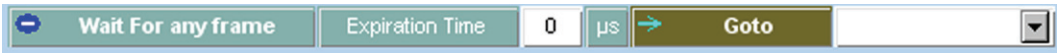
Then click the down arrow on the **If** Drop-down combo box in the inserted instruction and choose the condition for the jump and then the down arrow on the **Goto** Drop-down combo box and choose the label to designate the destination command.



For commands with payloads you may specify patterns for a condition by clicking the options button on a payload instruction and defining the required pattern(s).



Wait for any Frame To wait for the occurrence of a frame in a specific time period, insert a Wait for any Frame command and enter an expiration time in the time text box and the next program line to go to.



Insert Delay To delay program execution insert a **Delay** instruction. Enter the delay value (In milliseconds) in the number text box to define the desired delay.

Add Stop To define the end of the Host Emulator program, insert the **Stop Host Emulator** instruction.

Sample Host Emulator Program

Figure 32. shows a simple completed Host Emulator program.

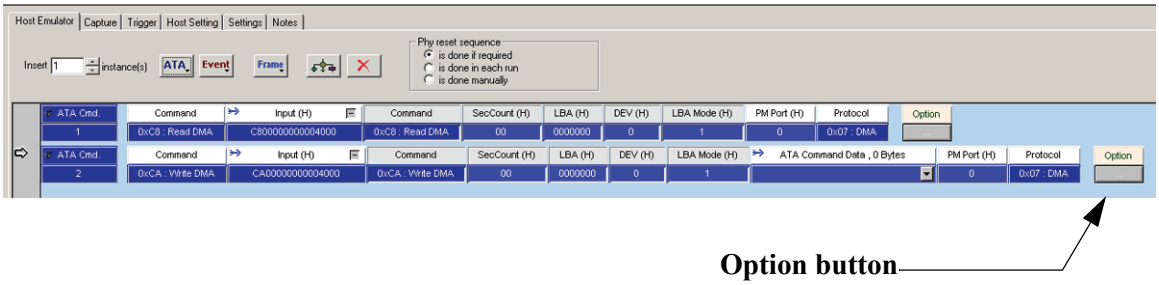


Figure 32. Sample Host Emulator Program

Data For commands requiring data blocks, click the down arrow of the **Payload Data** Drop-down combo box and choose from a set of pre-defined data blocks. If you need a new data block, click the Data Block icon on the tool bar to open a data block definition dialog. See page 68 for instructions on creating data blocks.

Phy reset sequence and initiation Will be performed when required by default, but you may choose to perform it on each run or manually.

Record and Play

This feature allows the selection of a range of commands in a trace and export them to a previously saved *.stc file for execution by the Host Emulator. You may choose to export commands from all available samples, between X, Y cursors or between designated commands.

To perform this action:

1. Run a capture project or open a previously run and saved captured trace file.
2. Set X and Y cursors if export between cursors is to be chosen.

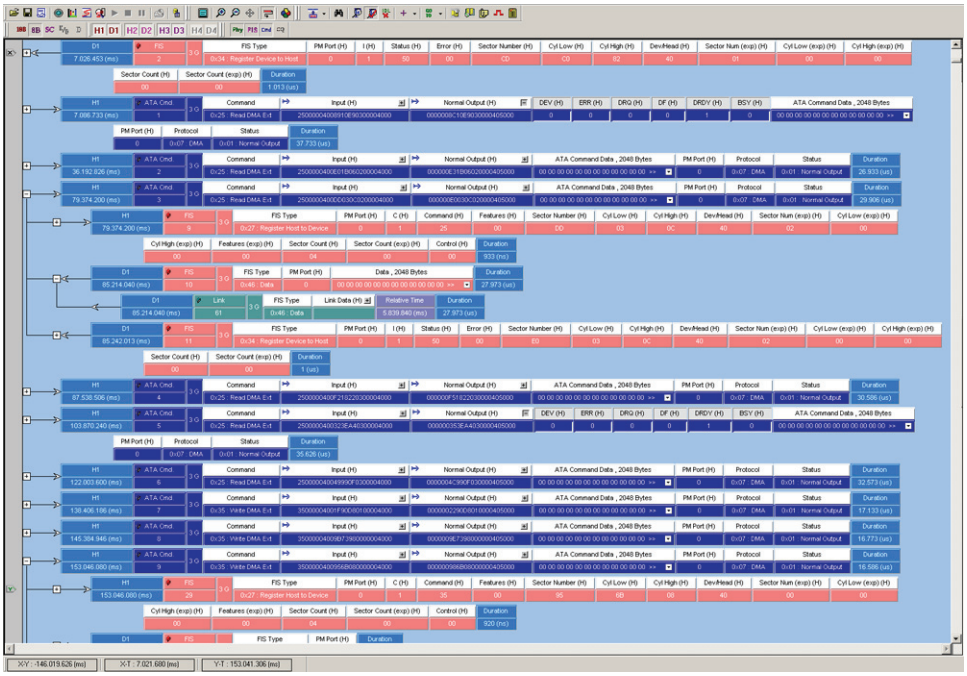
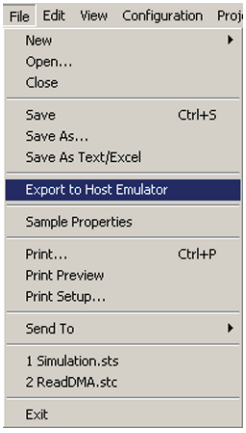


Figure 33. Captured Trace Display With Cursors Set

3. Click **File** and then choose **Export to Host Emulator.**



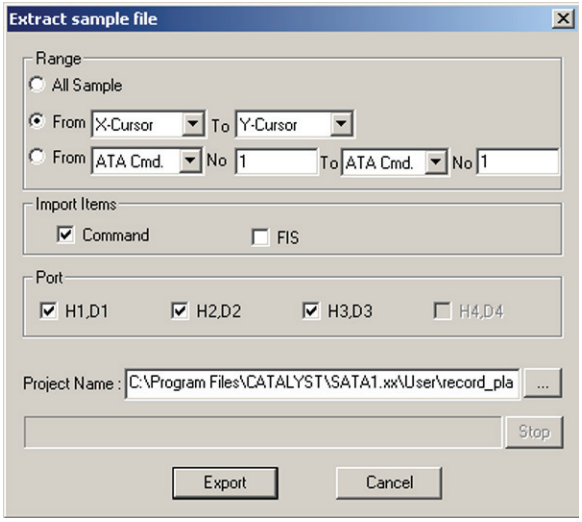


Figure 34. Export Definition Dialog

4. Click the ellipses next to the Project Name address bar and select an existing *.stc file.
5. Choose an export range and item, Command or FIS and click **Export.**
6. After the export action completes open the *.stc file that you exported to.

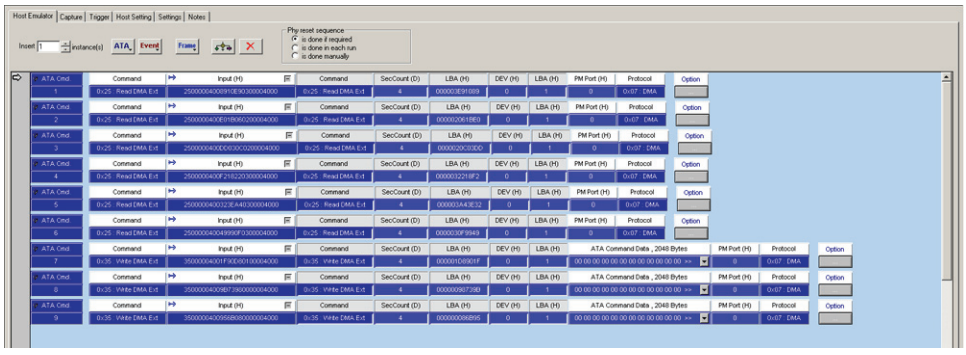


Figure 35. Host Emulator Program With the Exported Commands

7. You may run this set of commands immediately, or modify it by inserting instructions and/or additional commands.

Error and Command Settings

Each command type offers the user the ability to set a variety of command settings and to introduce errors. Click the **Option** button on a command line to display the corresponding Error and command setting dialog.

ATA Error and Command Settings

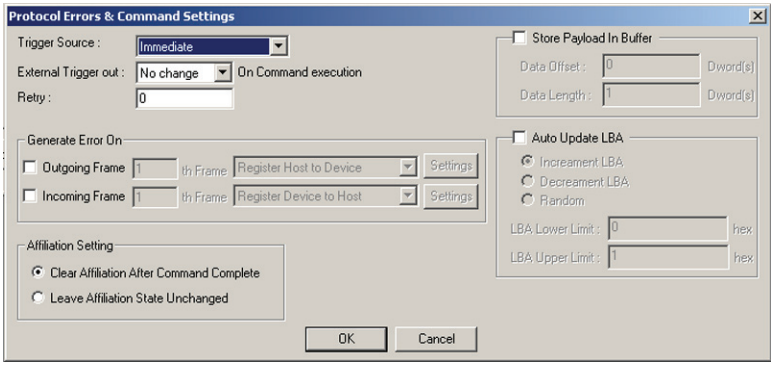


Figure 36. Protocol Error and Command Settings for ATA Dialog

Choose Trigger Source Click the down arrow on the **Trigger Source** combo box choose trigger type and click **OK**.

External Trigger Out Click the down arrow on the **External Trigger out** combo box choose the external trigger level and click **OK**.

Retry Enter a value for the number of command retries in the Retry text box.

Affiliation Setting Check **Clear Affiliation After Command Complete** to release device to commands from other sources or check **Leave Affiliation State Unchanged** to retain control of device.

Store Payload in Buffer Check the **Store Payload in Buffer** check box and enter values for **Data Offset** and **Data length** in the corresponding text box.

Auto Update LBA Check the **Auto Update LBA** check box and check the desired auto update option button.

Outgoing Frame Settings Check **Outgoing Frame** in the **Generate Error On** area and then the enabled **Settings** button to display the Outgoing Frame Settings dialog.

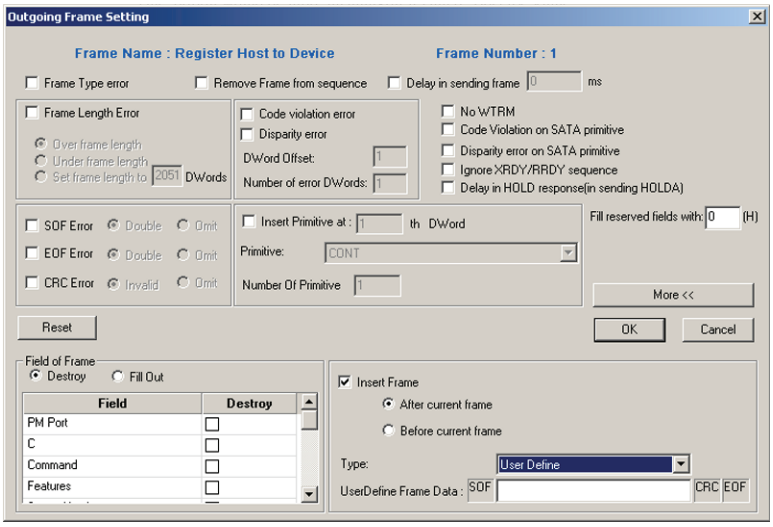


Figure 37. Outgoing Frame Settings

Frame Type Error Check to introduce a **Frame Type** error.

Delay in sending frame Check this to remove the frame from the command sequence.

Additionally check as required:

No WTRM
Code Violation on SATA primitive
Disparity error on SATA primitive
Ignore XRDY/RRDY sequence

Delay in HOLD response (in sending HOLDA)

Frame Length Error Check **Frame Length Error,** choose the type of error to introduce and click **OK**.

Code violation and Disparity error

Check these if required and specify DWord offset and the number of error DWords.

SOF, EOF and CRC errors Check these as required and specify **Double** or **Omit** by checking the corresponding option button.

Insert Frame Check **Insert Frame** and then click the down arrow on the **Type** combo box and choose the frame type. Make sure that you have clicked **More.**

Note: In addition to the pre-defined frame types you may scroll down the list to **User Defined** and enter your own frame data between SOF and EOF.

Insert Primitive Check **Insert Primitive** and click the down arrow on the **Primitive** combo box, choose a primitive and click **OK**.

Destroy Field of Frame Check **Destroy** in the **Field of Frame** area and check the fields to be destroyed.

Set Value of Field Check **Fill Out** in the **Field of Frame** area and enter desired values for fields

Incoming Frame Settings Check **Incoming Frame** in the **Generate Error On** area and then the enabled **Settings** button to display the Incoming Frame Settings dialog.

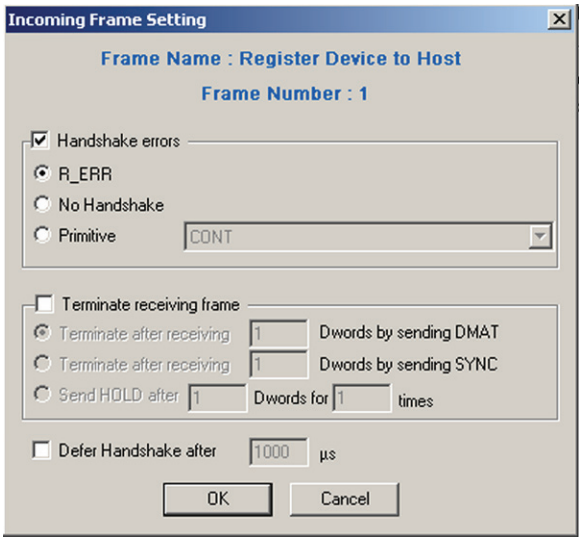


Figure 38. Incoming Frame Setting Dialog ATI

Handshake Errors Check **Handshake Errors** to enable selection of errors and primitives.

Check **Primitive** and click the down arrow on the **Primitive** combo box, choose a primitive and click **OK**.

Terminate Receiving Frame Check **Terminate Receiving Frame,** choose the termination type and click **OK**.

Defer Handshake To defer sending handshake to received commands check the **Defer Handshake after** check box and enter a value for the delay.

Host Emulator Settings

The Host Emulator Settings dialog allows you to select the Port(s) for exercising, choose to use the Host Emulator or Pattern generator and specify Host Emulator characteristics. Click the **Host Settings** tab to display the Host Emulator setting dialog.

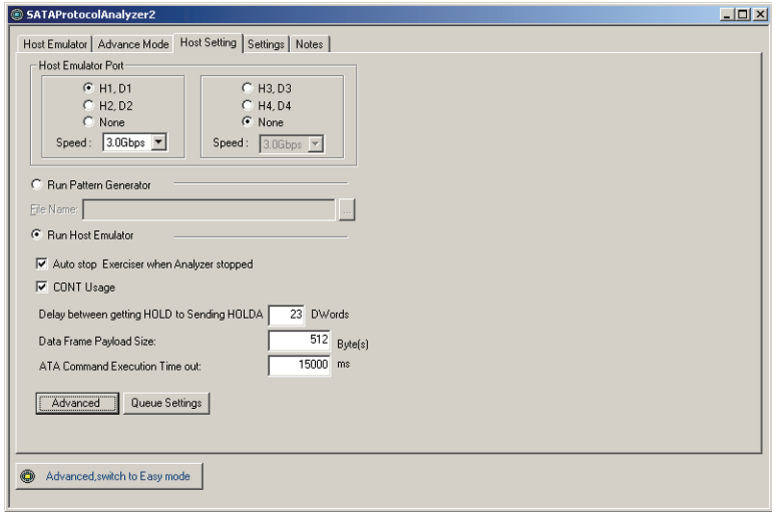


Figure 39. Host Emulator Settings Dialog

Check **Autostop exerciser when Analyzer Stopped** to halt the Host Emulator whenever the Analyzer is stopped.

Check **CONT Usage** to enable the specification of **Delay between getting HOLD to sending HOLDA**, **Data Frame Payload Size** and **ATA Command Execution Timeout**.

Click the **Advanced** button to display all settable options.

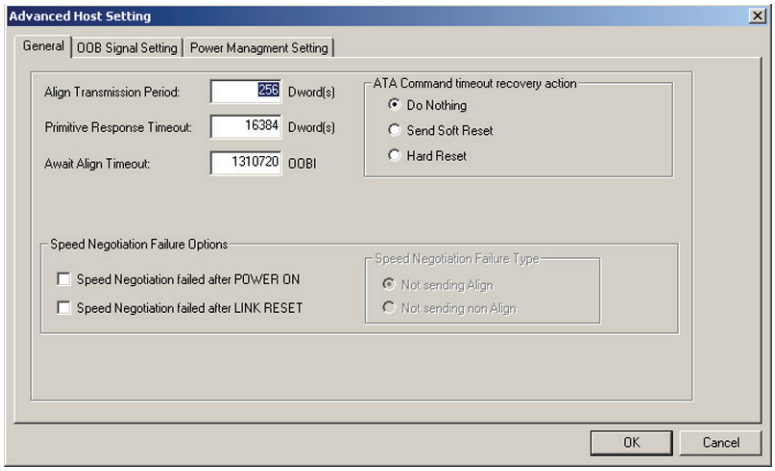


Figure 40. Advanced Settings

General

Settable options:

- Set **Align Transmission Period**, **Primitive Response Timeout** and **Await Align Timeout**.
- Choose **ATA Command timeout recovery action** by checking the corresponding option button.
- Specify **Speed Negotiating Failure Option(s)** and a **Speed Negotiation Failure Type**.

OOB Signal Settings

Click the **OOB Signal Setting** tab in the Advanced Host Setting dialog to specify OOB parameters in the enabled editable fields.

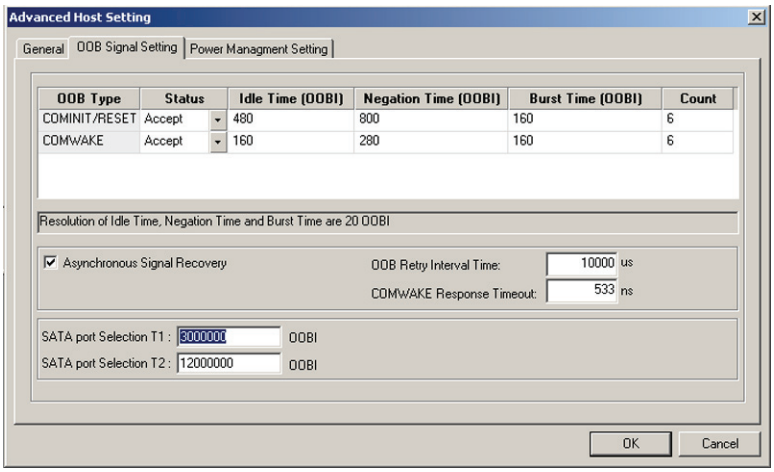


Figure 41. OOB Signal Settings

Asynchronous signal recovery

Set the OOB Retry Interval Time, check the Asynchronous signal recovery option check box and enter a value for the desired time.

Inter-reset Delay Settings

Enter a value for **SATA port Selection T1** to specify the inter-reset-assertion delay for the first event of the selection sequence and **SATA port Selection T2** to specify the inter-reset- assertion delay for the second event of the selection sequence.

Power Management Settings

Click the **Power Management Setting** tab in the Advanced Host Setting dialog to specify the settings in the editable fields.

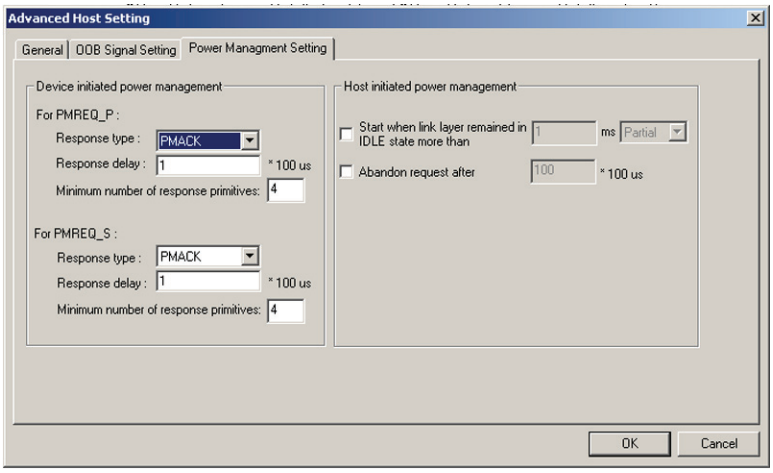


Figure 42. Power Management Settings

Set the required parameters for Host and/or Device initiated power management.

Queue Setting

Return to the **Host Emulator Setting** dialog (Figure 39.) and click the **Queue Settings** button to display the Queue Settings dialog and specify:

- Queue Depth
- NCQ Error Recovery
- Queued/Non-Queued Items Status
- NCQ/Queue Command Timeout

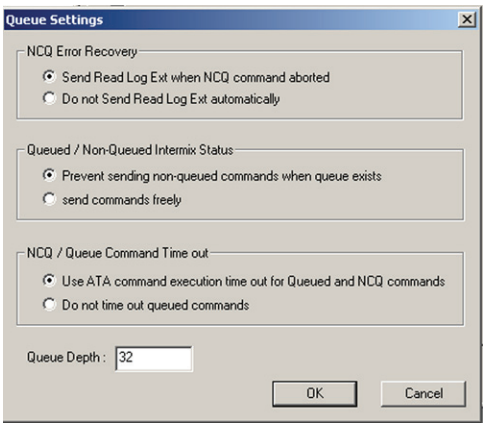


Figure 43. Queue Settings Dialog

Run Pattern Generator

To perform a capture with a Pattern Generator, check the Run Pattern Generator check box and enter a path to a pattern generator *.spg file in the File Name dialog.

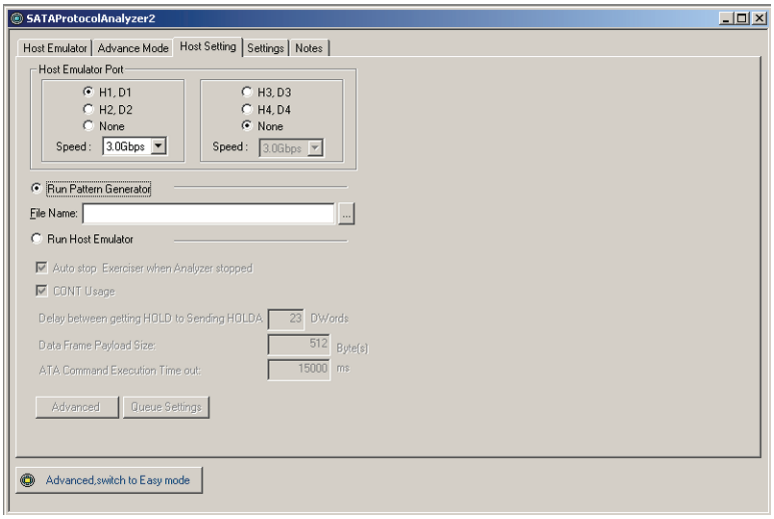


Figure 44. Run Pattern Generator Enabled

See Appendix A for instructions how to create a Pattern Generator file.

Project Settings

Click the **Settings** tab to display the Project settings dialog.

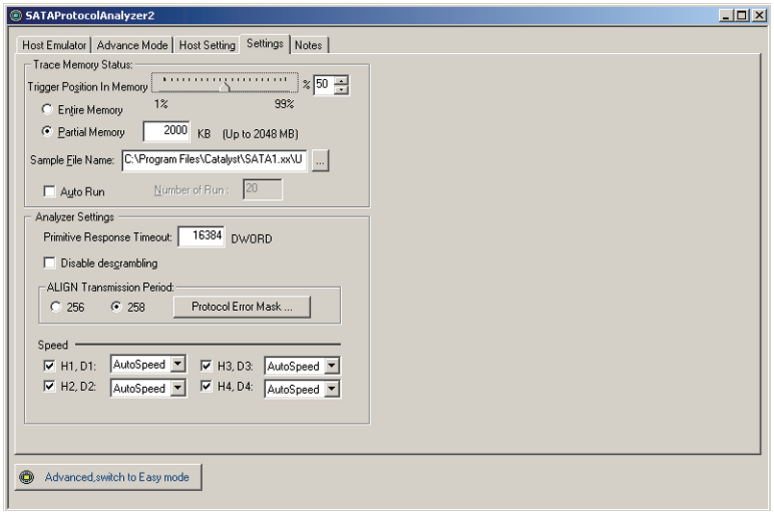


Figure 45. Project Settings Dialog

Memory Size

To reduce the capture memory size, check Partial Memory and enter the desired buffer size if you want to partition the trace memory for multiple captures or, check entire memory to allow capture for the entire memory if you want to capture the maximum amount of trace data.

Trigger Position Pre-Trigger is set by default at 50% which defines the percentage of data to be captured before and after the triggering event. You may change this percentage by dragging the slider to the desired value.

The capture of the specified percentage of the data prior to the triggering event cannot be guaranteed and may in some cases be 0. This can occur in cases where the triggering event occurs before the required number of pre-trigger event data can be stored. In these cases the data display will show fewer than the specified data points prior to the triggering event. For more detail see "Trigger Position" on page 47.

Sample File Name Click the ellipses next to the Sample File Name text box and choose a file name and location for the results of your current project.

Align Transmission period Click the desired option button.

Auto Run To repeat the current capture and trigger setup automatically, check the **Auto Run** checkbox and enter the number of times in the **Number to Run** text box. The capture and trigger will repeat automatically for the specified number of times and the results saved in consecutively numbered **Out.sts** files.

Disable descrambling Check this option to view scrambled data.

Primitive Response Timeout Specifies timeout value to detect a primitive response timeout error.

Choose Port Speed Click the down arrow next to the port speed box and choose a port speed.

Protocol Errors Click the **Protocol Error Mask** button and check any or all of the protocol errors to be excluded.

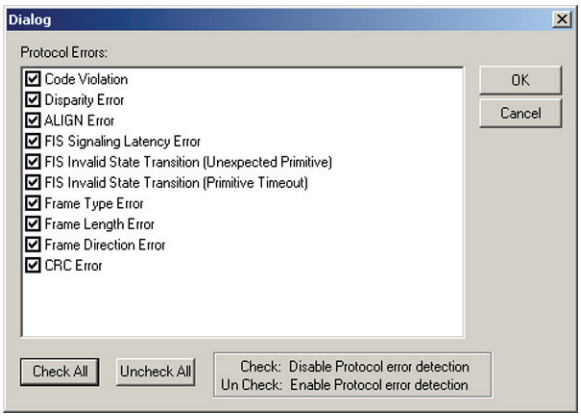


Figure 46. Protocol Error Mask

Project Note To include some descriptive information about the project, click on the **Notes** tab and enter a brief descriptive note.

Creating a Data Block

You may create the following four types of data blocks for use wherever data fields are used:

1. Random data pattern
2. Custom data pattern specifically for your application
3. Counter
4. Walking “1” or “0” pattern

To create a data block, click the **Default Data Block** Button on the Main toolbar to open the Data Block dialog box as shown in Figure 47.

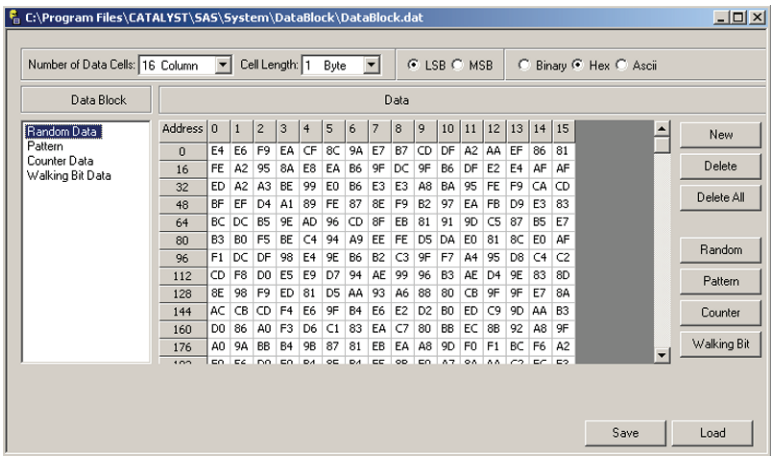


Figure 47. Default Data Block Dialog Box

1. To add another data block, click the **New** button in the Data Block dialog box.

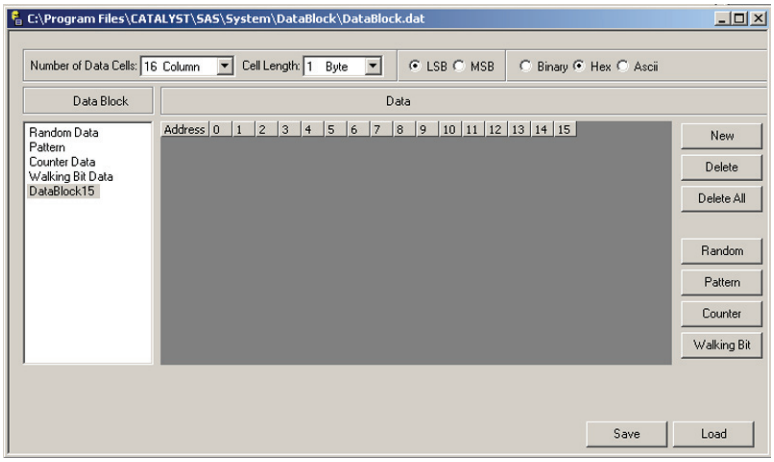


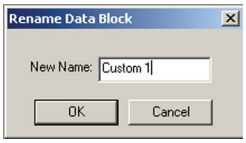
Figure 48. New Data Block Dialog Box

2. Choose the number of data columns (Up to 16 Data Cells/Row) and the Cell length (Up to 16 Bytes/Cell). This is a display function only.
3. Click either the Bin, Hex or Ascii option button to choose a desired number format.
4. Click either the LSB or MSB option button to choose a desired bit order.

Naming a Data Block

Each new data block is automatically assigned a sequential data block number as it is created. To assign a unique descriptive name to a data block, right click the data block name to open the data block edit menu.

Choose **Rename.**



Enter a descriptive name in the **New Name** edit box and click **OK.**

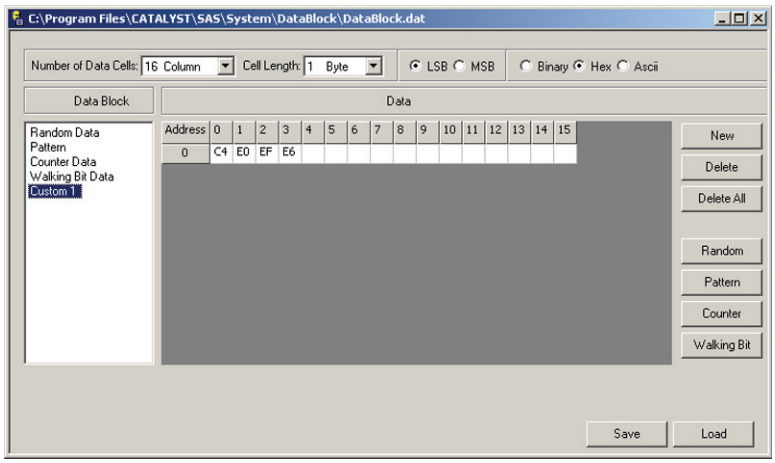


Figure 49. Sample Active Data Block Custom 1

You may enter data in the defined cell structure by choosing one of the four available methods. Define your own pattern, set a counter, choose a Random Pattern or choose a Walking Bit Pattern.

Define Your Own Pattern

1. Click **Pattern** to open the Define Pattern dialog box as shown in Figure 50.
2. Enter the desired data pattern in the Data Pattern edit box.
3. Choose the number of times that you would like that pattern to be repeated and click **OK.**

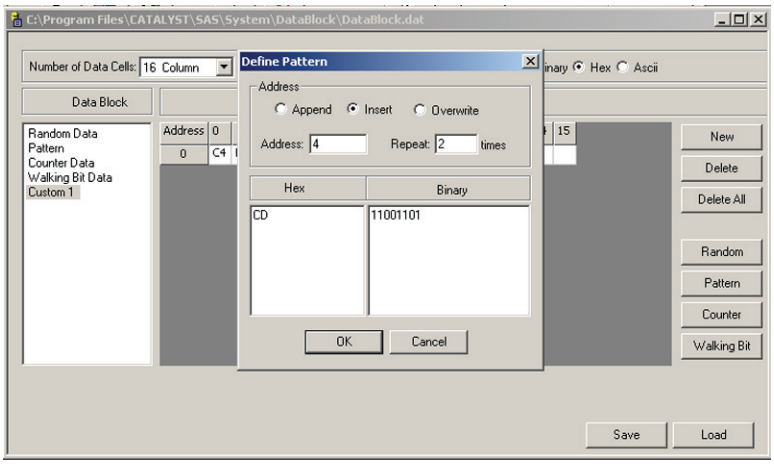


Figure 50. Define Your Own Data Pattern

Address The cell address starts at 0 for the first data entry and automatically increments to the next available address as data is written. You may set it back to a previously defined address to modify its content or insert additional data at that point.

Insert/Overwrite Data To define if the data in a previously defined cell will be overwritten or new data inserted after that cell click the **Insert/Overwrite** button to toggle to the desired operation.

Counter To use a counter as data click the **Counter** button, enter a Starting Number for the counter and the data address that you wish to count to and click **OK.**

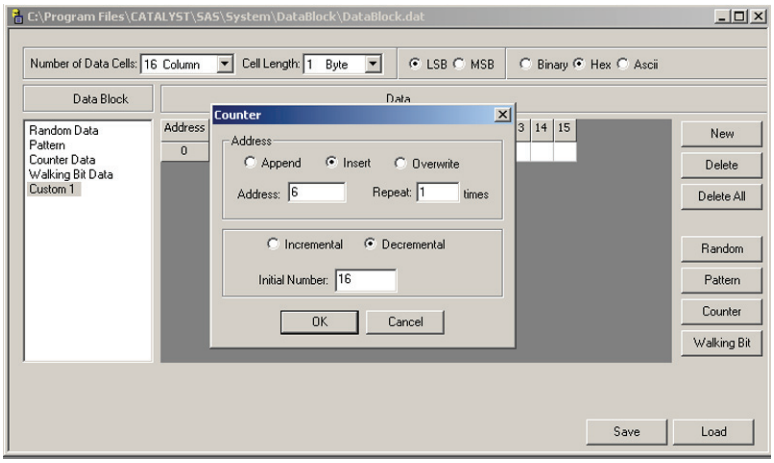


Figure 51. Set Counter as Data

Random Data Pattern

To use a random data pattern, click the **Random** button and enter the number of times that you wish the pattern repeated and click **OK.**

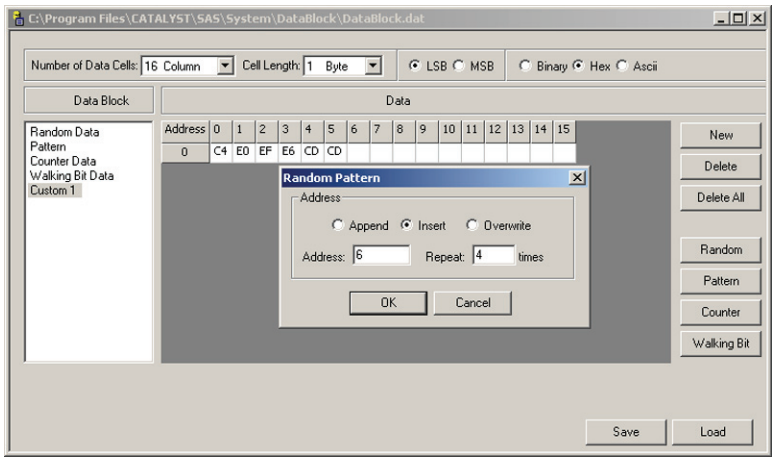


Figure 52. Choose a Random Pattern

Walking Bit Pattern

To use a walking bit pattern, click the **Walking Bit** button and choose either a walking bit of "0" or "1", the walk direction, the start position and the number of times the pattern is to be repeated.

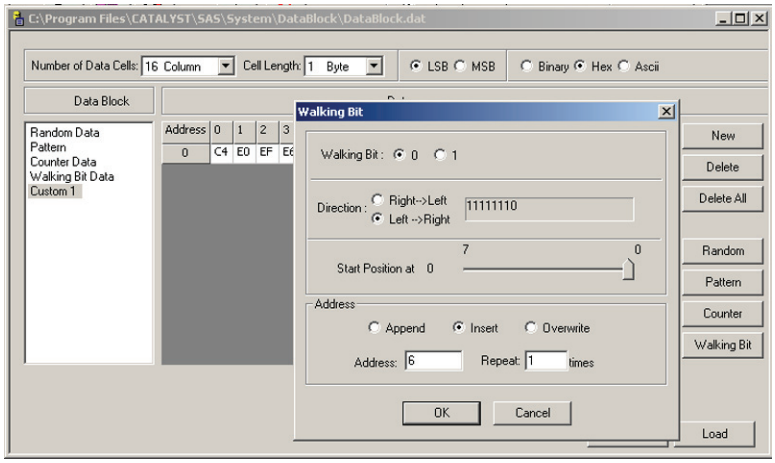


Figure 53. Define a Walking Bit Pattern

Save

When you have completed a data block definition click the **Save** button to save the newly created data block.

Creating and Editing Data Blocks as Text

You may create and edit data blocks using a text editor such as Windows Notepad. To create a data block in Notepad. Launch notepad. Enter a header consisting of [Item1, Item2, Item3, Item4, Item5] where:

Item1 is the name of the Data Block'

Item2 is Size of the Data Block or the number of bytes in the format

Item3 is Format of the data (HEX, BIN, ASCII)

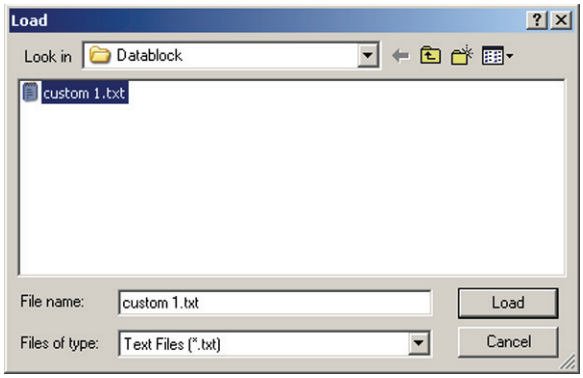
Item4 is the group of bytes defined (1, 2, 4, 8 or 16)'

Item5 is the direction (LSB or MSB)

Then enter the data in space delimited Hex format and save as a *.txt text file.

Load data

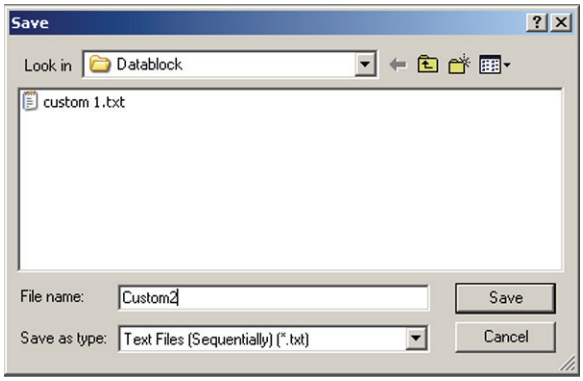
To import Text Editor created data click the **Load** button in the data block definition dialog to open the **Load** dialog.



Choose the desired file and click **Load**.

Modify existing data

To create a new data block from an existing data block using a text editor: Select the data block to be edited from the Data Block Name list and click **Save As** to open the **Save As** dialog**.**



Assign a name to the new data block text file and click **Save**.

You may now edit the newly created text file using Notepad or any other text editor and then import it into the data block definition as described above.

Performance Analysis

Your SATA analyzer incorporates a powerful real time Performance Analysis capability that allows you to quickly measure performance parameters such as:

Link Usage

- Bus Utilization (Ratio of FIS/Update Interval[1])
- Transmission Efficiency (Ratio of Payload Time/FIS Time)
- Throughput (Amount of data transferred during the update interval)
- Average Payload Size (Ratio of Overall Payload Size/Number of Data FIS)
- Completed Command Rate
- Data Efficiency (Ratio of Data Payload Time/Data FIS Time)

Event Counts

- Number of Frame Events
- Number of Data Frame Events
- Number of Protocol Errors
- ATA Command Number
- Number of Completed Commands

Event Times

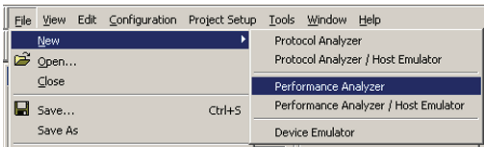
- Total Frame Event time
- Total Data Frame time
- Total Data Payload time
- Total Idle time

The results are displayed in any of user selectable formats such as Area, Column and Bar, Line or Pie chart with 3D and Grid enhancement options for ease of interpretation.

1. The time during which parameters are measured.

Performance Analysis with Analyzer Only

New Performance Analysis Project To start a **New** project, click **File** on the main menu bar, choose **New**, and select "Performance Analyzer".



Click the **Open Last Saved Performance Analyzer** button on the main menu bar to open the last saved performance analysis.

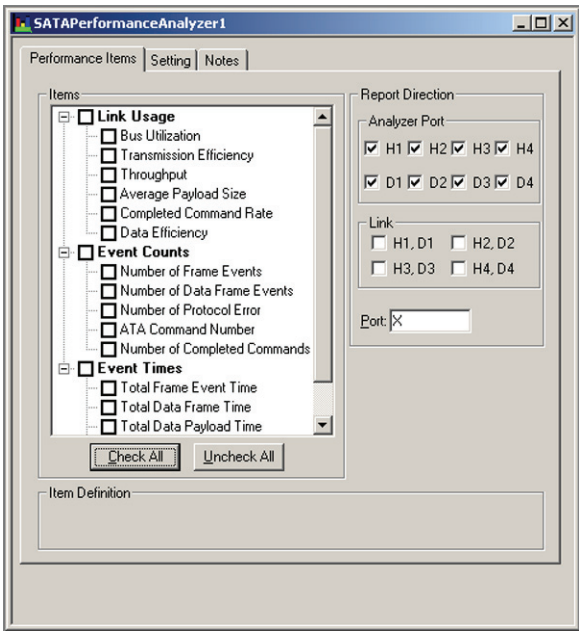


Figure 54. Performance Analysis Definition

Define Performance Click the corresponding Performance Item check box in the Items area. Performance may be measured in the following categories:

1. Link Usage including:
   - Bus Utilization (Ratio of payload time to update interval)
   - Transmission Efficiency (Ratio of payload time to frame time)
   - Throughput (Quantity of payload or useful data transferred during update interval)
   - Average Payload Size (Ratio of overall payload size to size of number of data frame)

- Completed Command Rate (Ratio of completed commands to total commands)
- Data Efficiency (Ratio of data payload time to data frame time)

2. Event Counts including:
   - Number of Frame Events
   - Number of Data Frame Events
   - Number of Protocol Errors
   - Number of Completed Commands
   - Number of Command Events
3. Event Times including:
   - Total Frame Event Time
   - Total Data Frame Time
   - Total Data Payload Time
   - Total Idle Time
4. Click the Setting tab to open the Setting dialog.

The settings dialog allows you to set the Analysis update interval and define the counter mode.

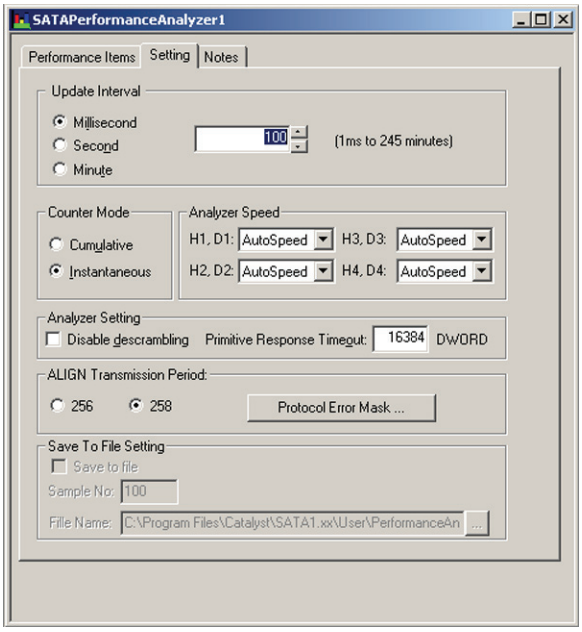
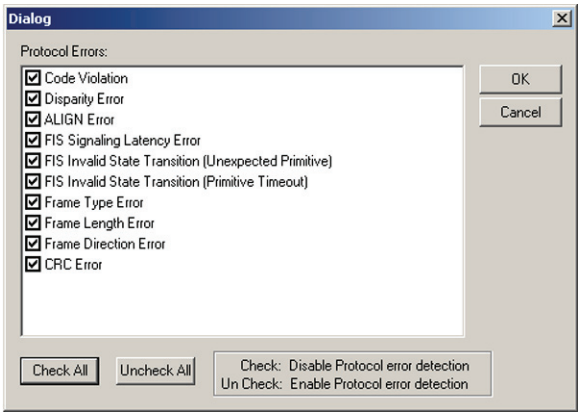


Figure 55. Performance Analysis Settings

1. Choose the **Update Interval** in the range of 1ms to 245 minutes.

2. Set the **Counter Mode** to either Cumulative or Instantaneous (Cumulative causes the measurement counters to continually increment and Instantaneous causes the measurement counters to be reset for each Update interval).
3. Set the **Analyzer Speed** for each port used.
4. Check **Disable Scrambling** for the analysis as required and specify **Primitive Response Timeout**.
5. Choose **ALIGN Transmission Period**.
6. To select **Protocol Errors** for exclusion from the analysis, click the **Protocol Error Mask** button and check and or all of the protocol errors to be excluded.



Save the Analysis result To save the analysis result for later review. check **Save to file** and enter the path where you would like to save the result.

Add a note To assist in interpreting the results at a later time you may include a descriptive note about the project by clicking the Notes tab to open the **Notes** dialog.

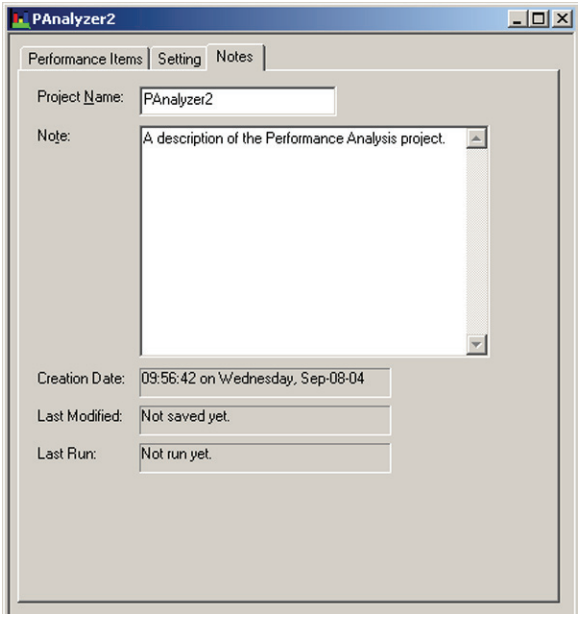


Figure 56. Performance Analysis Descriptive Note

7. To Perform the defined Analysis click the  **Run Hardware** button and wait for the result to display**.**

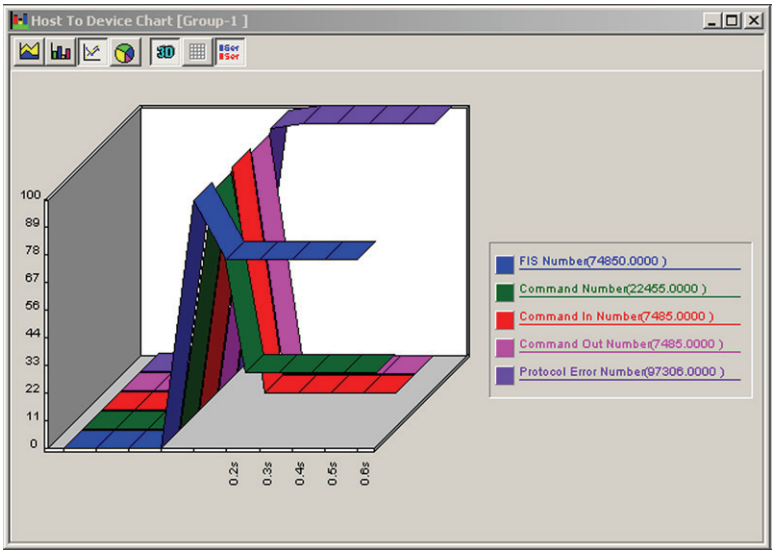
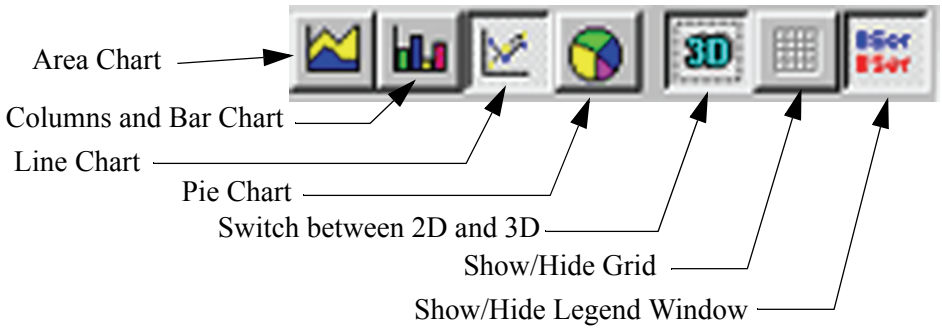


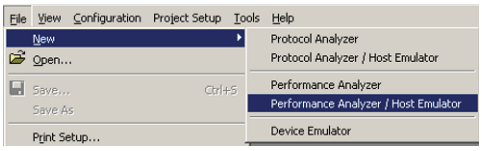
Figure 57. Typical Performance Analysis Results Display

Alternate Display Format You may choose to display the result as 2D, 3D, etc. by clicking the corresponding "Graphics Setting" on the Performance Analysis display toolbar.



Performance Analysis with Host Emulator

To perform a Performance Analysis with Host Emulator generated bus traffic, click **File, New** and choose **Performance Analyzer/Host Emulator**.



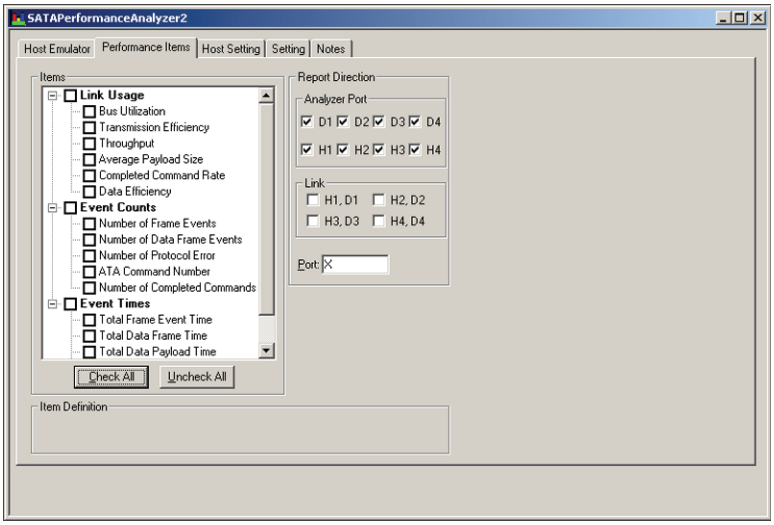


Figure 58. Performance Analysis Definition with Host Emulator

1. Click the Host Emulator tab and program the Host Emulator as described in "Programming the Host Emulator" on page 53.
2. Click the Performance Items tab and set up the Performance analysis as described starting on page 74.

Host Settings

1. Click the Host Setting tab and check the **Run Host Emulator** option button.
    - Chose the **Emulator Port(s)**
    - Enable **CONT** Usage and **Accepting Power Management** as required.
    - Specify Delay between getting **HOLD** and sending **HOLDA**, **Data Frame Payload Size**, and **ATA Command Execution Time-out**.

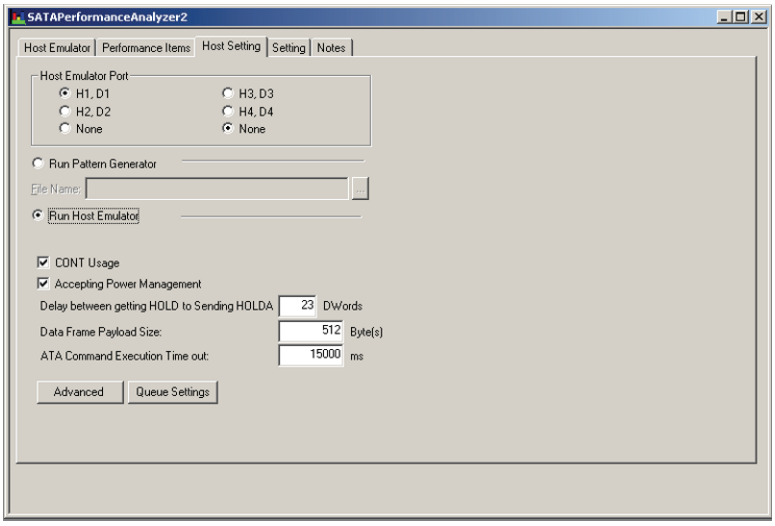


Figure 59. Host Setting Dialog

2. Click the **Advanced** button in the Setting dialog to:

   - Set **Align Transmission Period**, **Primitive Response Timeout** and **Await Align Timeout**.
   - Choose **ATA Command timeout recovery action** by checking the corresponding option button.
   - Specify **Speed Negotiating Failure Option(s)** and a **Speed Negotiation Failure Type**.

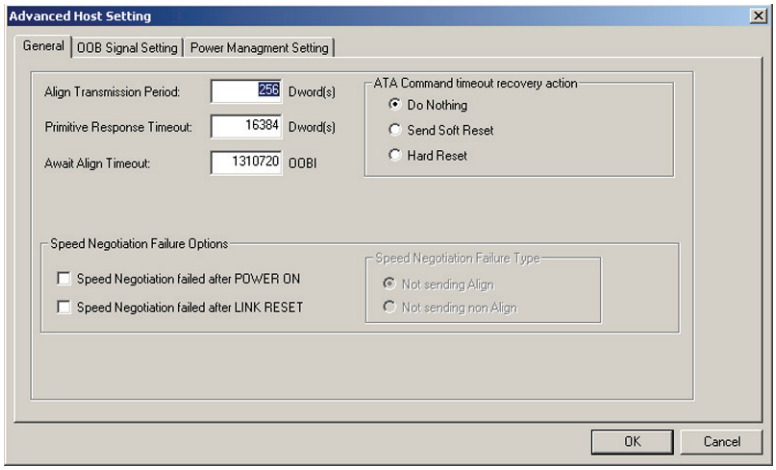


Figure 60. General Settings Dialog

3. Click the **OOB Signal Setting** tab in the Advanced Host Setting dialog to specify OOB parameters in the enabled editable fields.

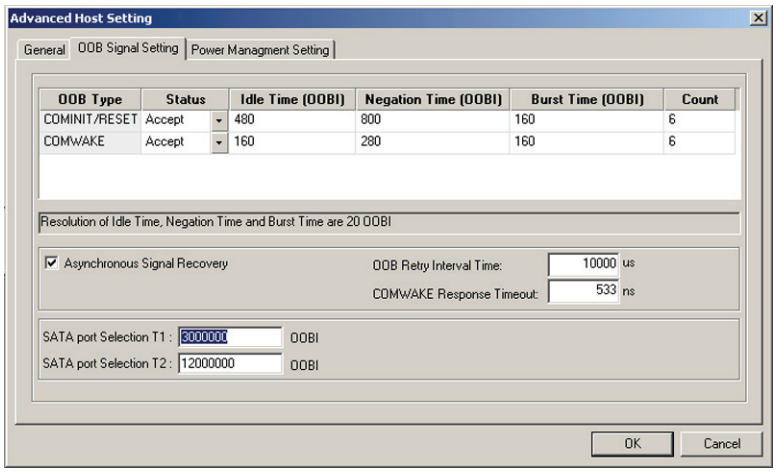


Figure 61. OOB Signal Setting Dialog

Asynchronous signal recovery To set the OOB Retry Interval Time, check the Asynchronous signal recovery option check box and enter a value for the desired time.

Inter-reset Delay Settings Enter a value for **SATA port Selection T1** to specify the inter-reset-assertion delay for the first event of the selection sequence and **SATA port Selection T2** to specify the inter-reset- assertion delay for the second event of the selection sequence.

Power Management Settings Click the **Power Management Setting** tab in the Advanced Host Setting dialog to specify the settings in the editable fields.

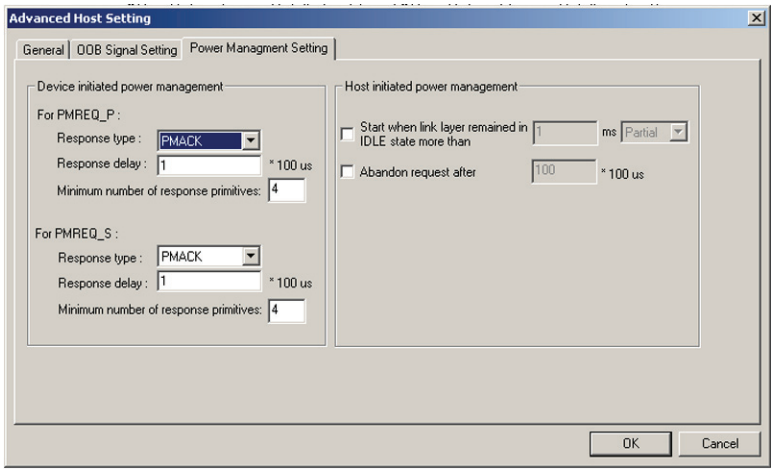


Figure 62. Power Management Settings

Set the required parameters for Host and/or Device initiated power management.

4. Return to the **Host Setting** dialog (Figure 59.) and click the **Queue Settings** button to display the Settings dialog and specify:
    - Depth
    - NCQ Error Recovery
    - Queued/Non-Queued Items Status
    - NCQ Command Timeout.

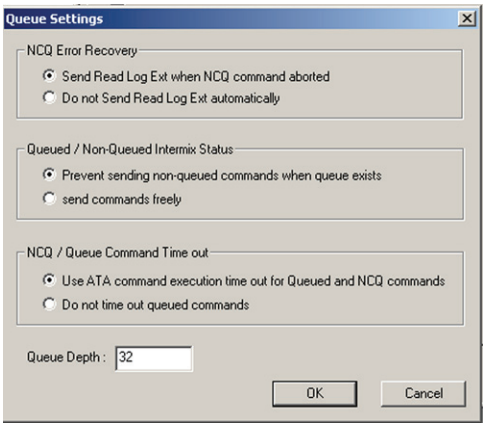


Figure 63. Settings Dialog

To perform an analysis with a Pattern Generator check the **Run Pattern Generator** option button and choose a pattern generator *.spg file. See Appendix A for instructions for creating a Pattern Generator file.

Performance Analysis Project Settings

Click the **Setting** tab in the Performance Analysis definition dialog (Figure 58.) to display the Performance Analyzer Project Setting dialog.

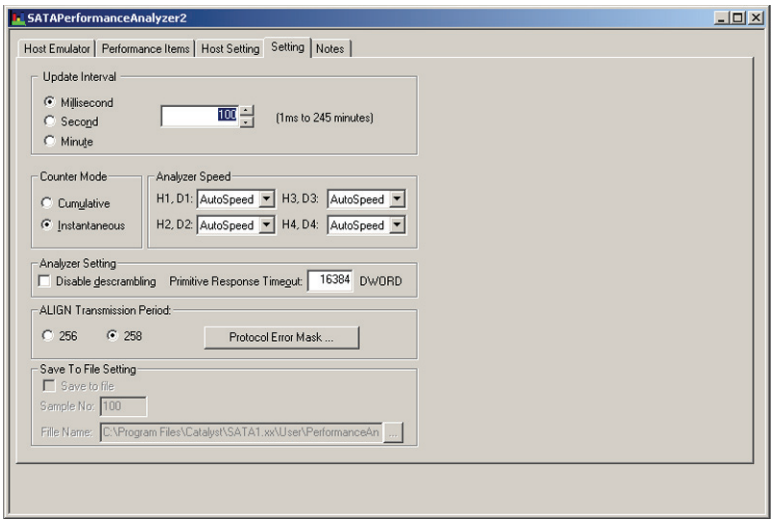


Figure 64. Performance Analysis Project Setting Dialog

1. Choose the **Update Interval** in the range of 1ms to 245 minutes.
2. Set the **Counter Mode** to either Cumulative or Instantaneous (Cumulative causes the measurement counters to continually increment and Instantaneous causes the measurement counters to be reset for each Update interval).
3. Set the **Analyzer Speed** for each port used.
4. Check Disable Scrambling for the analysis as required and specify Primitive Response Timeout.
5. Choose ALIGN Transmission Period.
6. To select **Protocol Errors** for exclusion from the analysis, click the **Protocol Error Mask** button and check any or all of the protocol errors to be excluded.

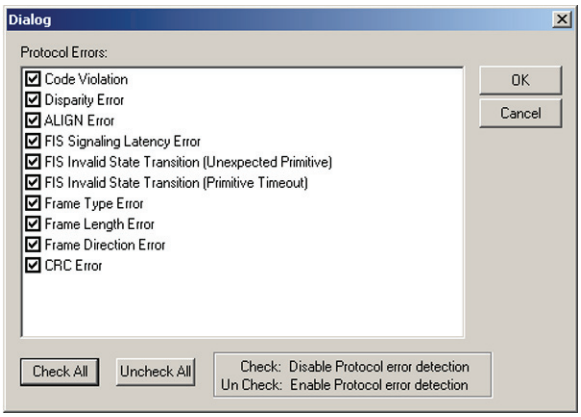


Figure 65. Protocol Error Mask

Save the Analysis result To save the analysis result for later review. check **Save to file** and enter the path where you would like to save the result.

Add a note To assist in interpreting the results at a later time you may include a descriptive note about the project by clicking the Notes tab to open the **Notes** dialog.

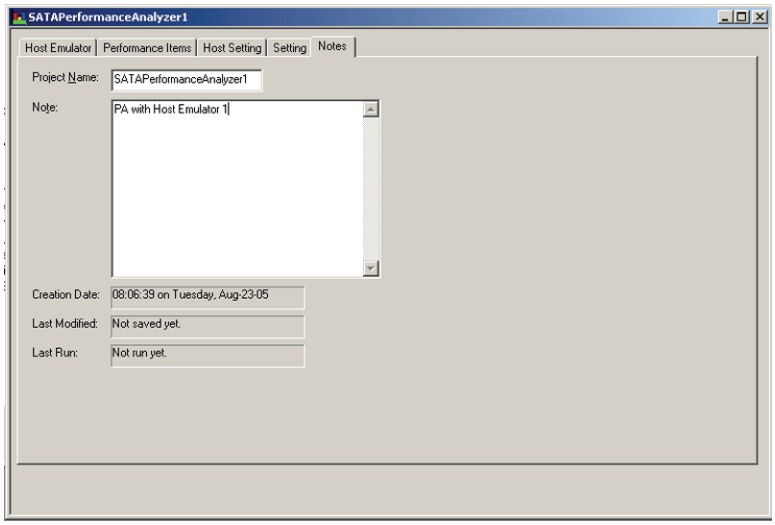


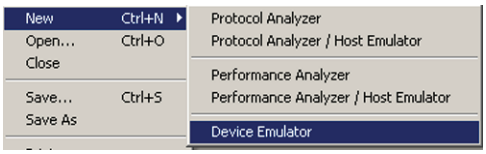
Figure 66. Performance Analysis Descriptive Note

7. To Perform the defined Analysis click the  **Run Hardware** button and wait for the result to display**.**

Device Emulation

Not in current version for STX-460.

To set up a Device Emulation, click **File, New** and choose **Device Emulator**.



The Device Emulation project opens with the Identify Page tab selected

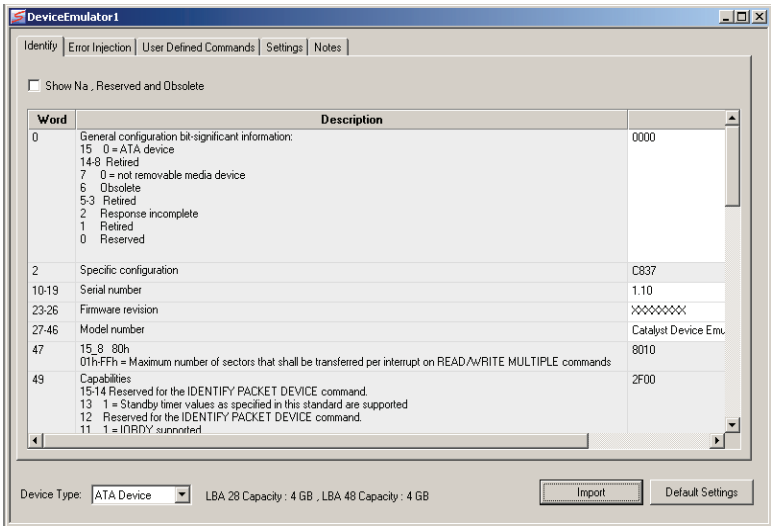


Figure 67. Device Emulation Project Pages Tab

The Device Emulator pages open with default settings for each page. To change settings for your application:

1. Set the Device Type of either ATA or ATAPI
2. Set required values for each of the enabled (White) Value fields on the Identify page.
3. To reset the pages to the default settings, click the **Default Settings** button.
4. To use a previously defined emulation, click the **Import** button and select the desired emulation.

Error Injection

Clicking the Error Injection tab opens the **General Errors** dialog. In addition to specifying general errors you may also set errors for **ATA Commands** by clicking the corresponding Icon in the Errors window.

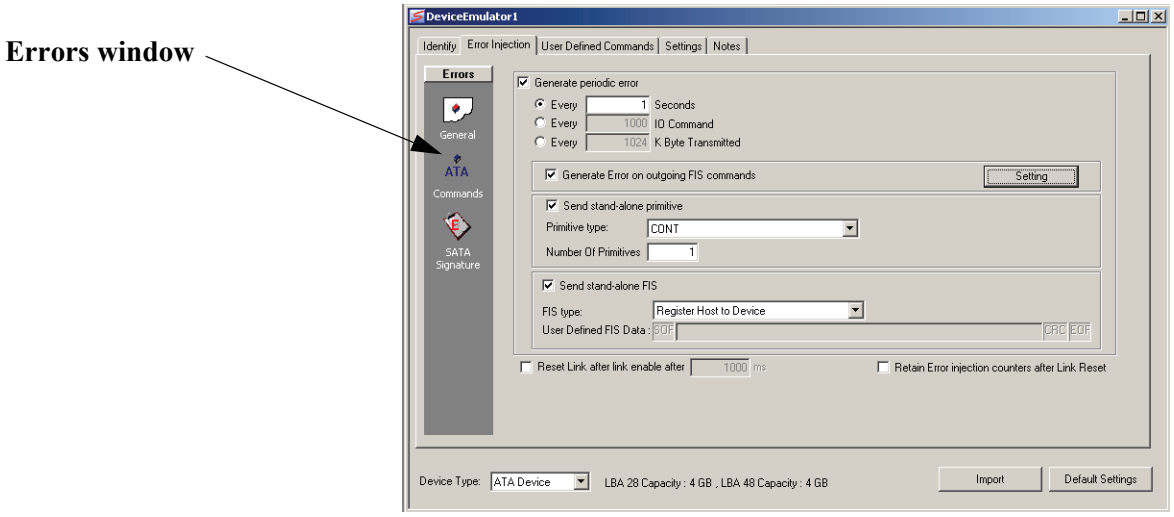


Figure 68. Setting General Errors

Setting General Errors

Device Type Click the down arrow on the Device Type drop down combo box and choose a device type.

Generate Periodic Error Check **Generate Periodic error** and choose an error rate.

Note: Checking **Generate Periodic Error** enables specific error selection in this category with the exception of **Reset Link** which is independently selectable.

Stand-alone Primitive Check **Send stand-alone primitive**, click the down arrow on the **Primitive** list box, choose a primitive type and enter a value for the number to be sent.

Stand-alone FIS Check **Send stand-alone FIS,** click the down arrow **FIS type** list box and choose an FIS type.

Note: In addition to the pre-defined FIS categories you may define your own by scrolling down the list and choosing User Defined and enter the desired FIS data.

Reset Link Check **Reset Enable after link enable after** and specify time.

Generate Error on Outgoing FIS Check **Generate Error on Outgoing FIS of commands** and then click the **Settings** button.

Retain Error injection counters after link reset Check this to retain error count.

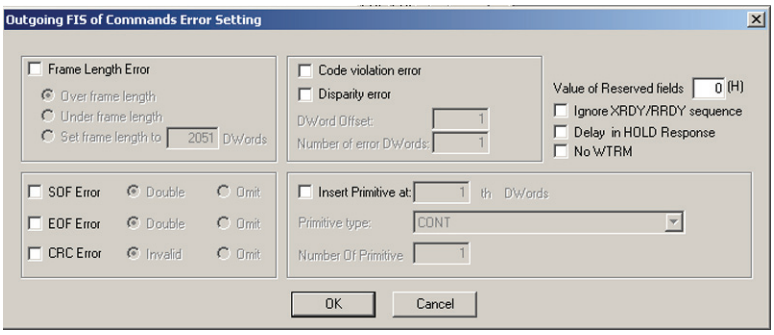


Figure 69. Outgoing FIS Error Setting

SOF, EOF and CRC Errors Check any or all as required and check the criteria for introduction.

Code Violation and Disparity errors Check as required and specify **DWord** offset and Number of DWord errors for Disparity error.

Additional Settings Set and check as required:

- Value for Reserved fields
- Ignore XRDY/RRDY Sequence
- Delay in HOLD Response
- No WTRM

Insert Primitive Check **Insert Primitive**, click the down arrow on the **Primitive** list box, choose a primitive type and enter a value for the Number of Primitive.

ATA Commands Errors

Click the **ATA Commands** icon in the Errors window to display the ATA Commands Error Setting Dialog.

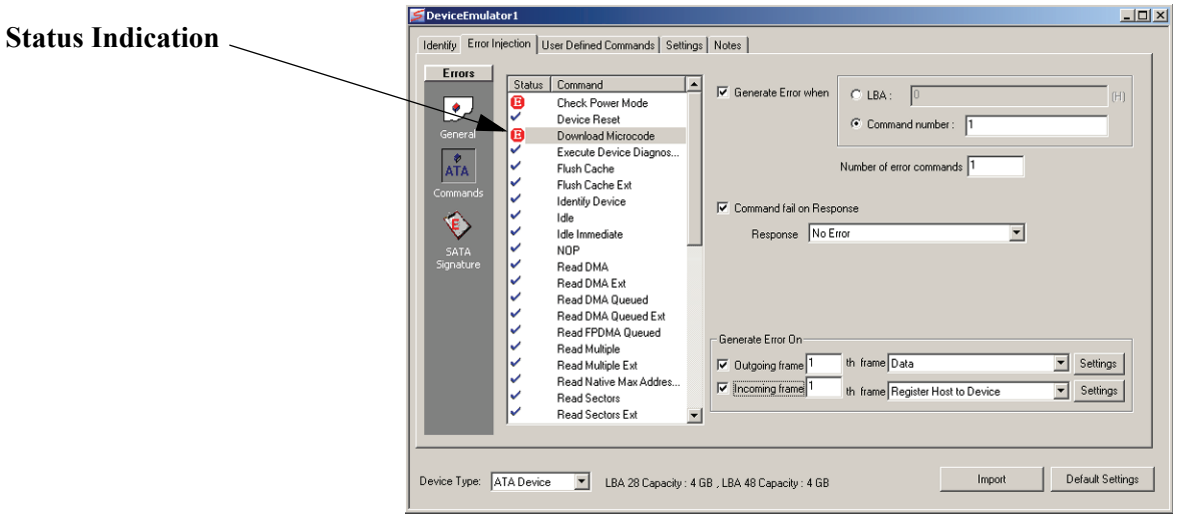


Figure 70. SAS Commands Error Setting Dialog

To set errors for ATA commands:

1. Click on a command for which you wish to set an error and then check the **Generate Error when** check box and chose **LBA or Command number** with a value in the corresponding text box.
2. Set the number of times the error should be repeated by entering a value in the **Number of Error Commands** text box.

Note: Once you check the **Generate Error when:** check box for a command, a red status indication appears next to the command selected indicating an error condition.

3. You may force a defined response frame to be sent to the host by checking the **Command fail on Response** check box and then choose a pre-defined response.
4. Repeat for every command for which you wish to set an error.

Outgoing Frame Settings

Check **Outgoing Frame** in the **Generate Error On** area and then the enabled **Settings** button to display the Outgoing Frame Setting dialog.

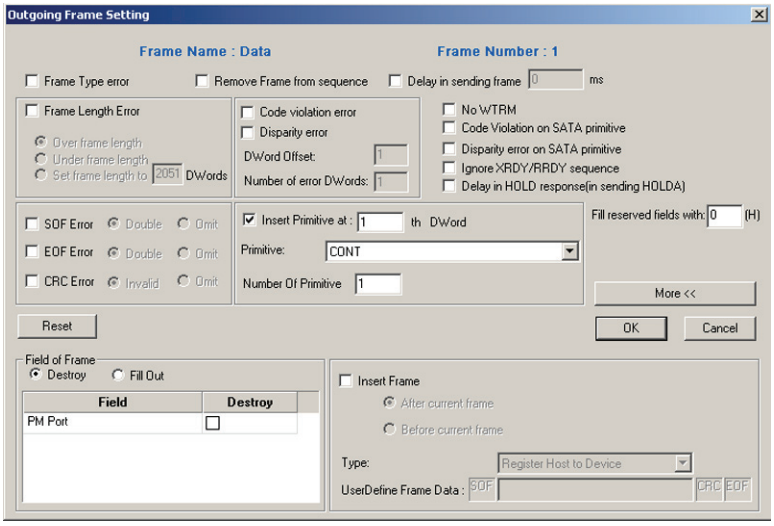


Figure 71. Outgoing Frame Setting Dialog

Frame Type Error Check this to introduce a frame type error.

Remove Frame from sequence Check this to remove frame from sequence.

Delay in sending frame Check this to delay sending frame and enter a value for the desired delay.

Additionally check as required:

- No WTRM
- Code Violation on SATA primitive
- Disparity error on SATA primitive
- Ignore XRDY/RRDY sequence
- Delay in HOLD response (in sending HOLDA)

Frame Length Error Check **Frame Length Error,** choose the type of error to introduce and click **OK**.

Code violation and Disparity error:

Check these if required and specify DWord offset and the number of error DWords.

SOF, EOF and CRC errors Check these as required and specify **Double** or **Omit** by checking the corresponding option button.

Insert Primitive Check this and then click the down arrow on the **Primitive** drop down combo box and choose a primitive type to insert.

Destroy Field of Frame Check **Destroy** in the **Field of Frame** area and check the fields to be destroyed.

Set Value of Field Check **Fill Out** in the **Field of Frame** area and enter desired values for fields

Insert Frame Check **Insert Frame** and then click the down arrow on the **Type** combo box and choose the frame type. Make sure that you have clicked **More.**

Note: In addition to the pre-defined frame types you may scroll down the list to User Defined and enter your own frame data between SOF and EOF.

Incoming Frame Settings

Check **Incoming Frame** in the **Generate Error On** area and then the enabled **Settings** button to display the Incoming Frame Settings dialog.

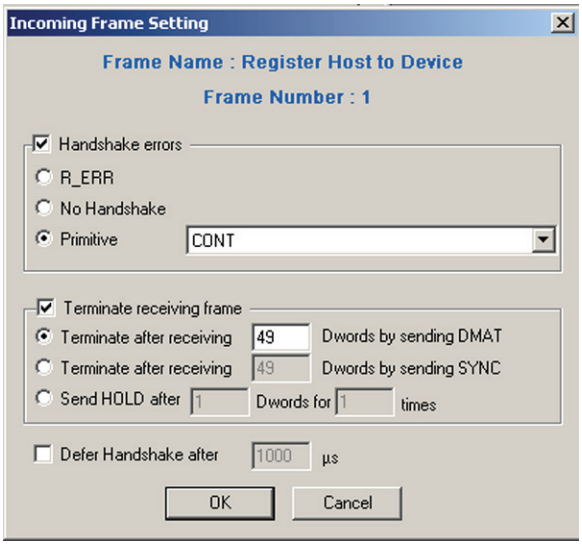


Figure 72. Incoming Frame Setting Dialog

Insert Error Check **Handshake Errors** to enable selection of errors and primitives.

Insert Primitive Check **Insert Primitive** then click the down arrow on the **Primitive** drop down combo box and choose a primitive.

Terminate receiving frame Check **Terminate receiving frame** and choose the termination condition.

SATA Signature

Click the **SATA Signature** icon in the Errors window to display the SATA Signature Dialog.

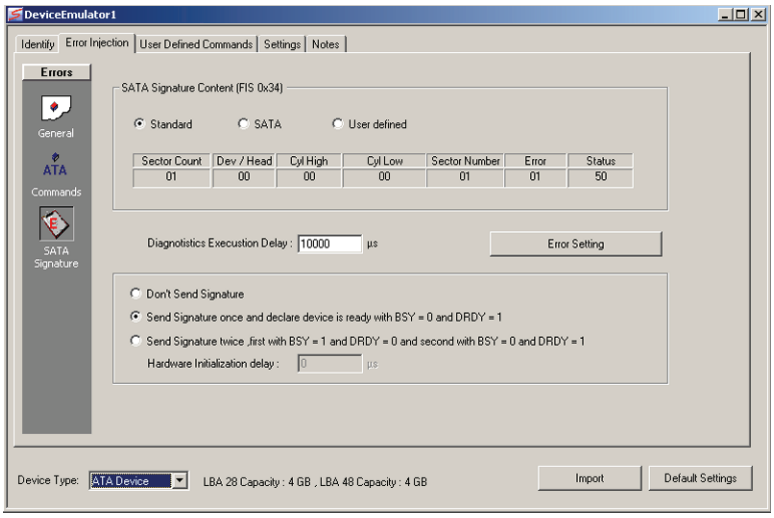


Figure 73. SATA Signature Dialog

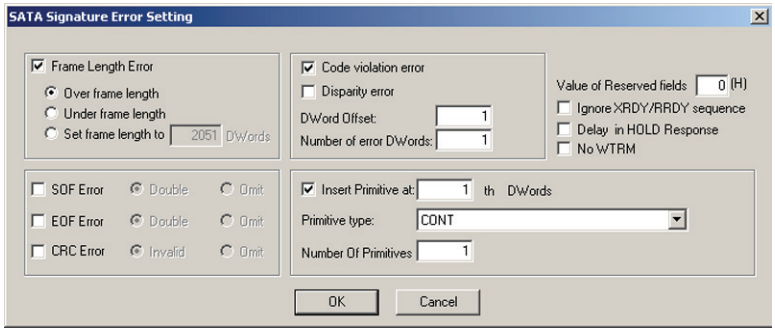
Choose SATA Signature Content Choose either the pre-defined **Standard** or **SATA** or you may define your own by checking the **User Defined** option button and entering values in the enabled fields.

Specify when to send Signature You may choose from **Don't Send Signature**, **Send Signature once and declare device is ready with BSY=0 and DRDY=1** or **Send Signature twice, first with BSY=1 and DRDY=0 and second with BSY=0 and DRDY=1** by checking the corresponding option button.

Note Choosing **Send Signature twice, first with BSY=1 and DRDY=0 and second with BSY=0 and DRDY=1** the Hardware initialization delay text box is enabled allowing you to set the hardware initialization delay.User Defined Commands

SATA Signature Errors

You may define errors to be introduced when sending a SATA Signature. To define the errors click the **Error Setting** button to open the SATA Signature Error Setting dialog.



Frame Length Error Choose the Frame Length Error type to introduce.

SOF, EOF and CRC Errors Check any or all as required and check the criteria for introduction.

Code Violation and Disparity errors Check as required and specify **DWord** offset and Number of DWord errors for Disparity error.

Additional Settings Set and check as required:

- Value for Reserved fields
- Ignore XRDY/RRDY Sequence
- Delay in HOLD Response
- No WTRM

Insert Primitive Check **Insert Primitive**, click the down arrow on the **Primitive** list box, choose a primitive type and enter a value for the Number of Primitive.

User Defined Commands

To create a command(s) specifically for your application Click the **User Defined Commands** tab to display the command definition dialog.

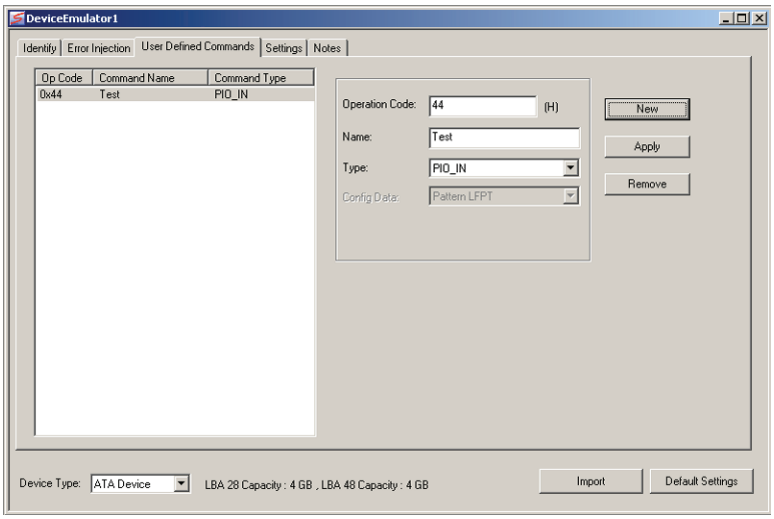


Figure 74. Command Definition Dialog

To define a command:

1. Click the down arrow on the Device Type combo box and choose the device type for which you wish to create a command.
2. Enter an **OP Code** and a **Name** in the corresponding text box.
3. Click the down arrow on the **Type combo box** and choose a command type.
4. For command types requiring configuration data click the down arrow on the enabled **Config data:** combo box and choose appropriate configuration data.

Note: When the device type is set to ATAPI the command types are totally different from ATA and enable you to set CDB length.

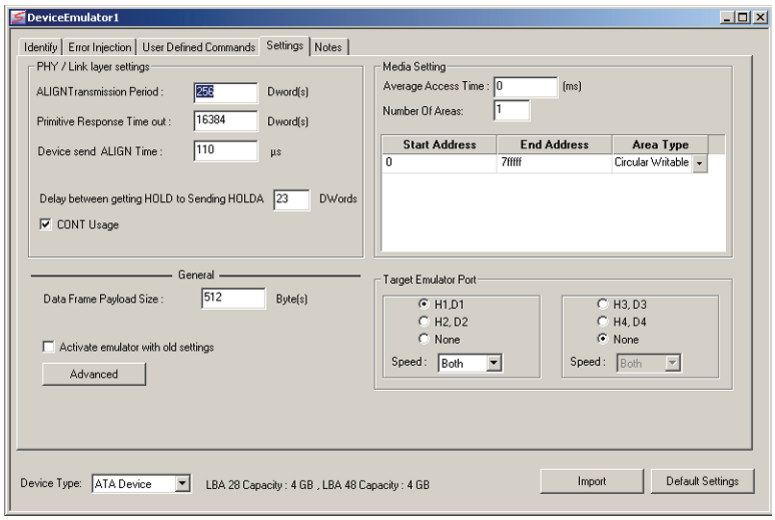
5. When done click the **New** button.
6. The defined command appears in the command name window.
7. To make changes to a previously defined command, highlight the command, make necessary changes and click **Apply.**

Device Emulator Settings

Click the **Settings** tab to display the Device emulator settings dialog.

The Settings page opens with a default set of values which you may modify for your specific application as required. To return to these values at any time click the **Default Settings** button.

.



PHY/Link layer settings Enter values for **Align Transmission Period**, **Primitive Response Timeout**, and **Standby Timeout**, **Device send Align Time** and **Delay between getting HOLD to Sending HOLDA** and specify desired delay. Check **CONT** usage if required.

Media Settings Enter a value for **Average Access time** and **Number of Area.** Define a **Start** and an **End** address, click the down arrow under the **Area Type** and choose **Normal Writable**, **Circular Writable** or **Non Writable.**

Payload size Enter a value for **Data Frame Payload Size** in the General area.

Choose Target Emulator Port Click a desired port option button and click the down arrow next to the **Speed** list box and choose a port speed.

Note: If you have made some changes to a defined emulation and would like to return to the original definition check the **Activate the emulator with old settings**.

Advanced Options Click the **Advanced** button to display the OOB Signal Setting, Power Management setting, Speed Negotiation, NCQ Command setting and Miscellaneous additional setting dialogs.

OOB Signal Setting

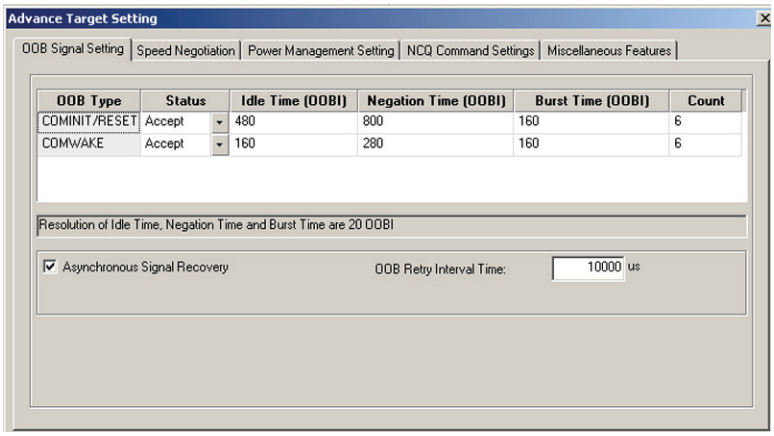


Figure 75. OOB Signal Setting Dialog

Make the necessary edits to the default values displayed in the white editable fields.

Speed Negotiation

Click the **Speed Negotiation** tab to display the Speed Negotiation dialog.

Specify **Speed Negotiating Failure Option(s)** and a **Speed Negotiation Failure Type**

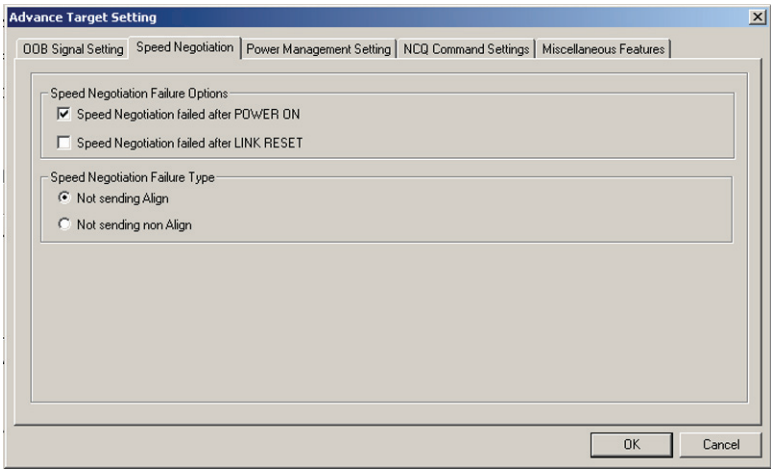


Figure 76. Speed Negotiation Dialog

Power Management Settings Tab

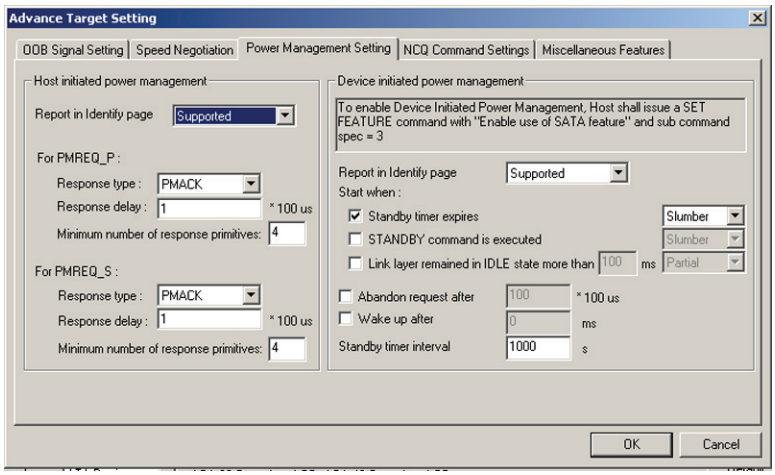


Figure 77. Power Management

1. In the **Host Initialized Power Management** area choose if the Report in the Identify page is supported or not.
2. Specify the response type for PMREQ_P and PMREQ_S, the delay and Minimum Number of Response Primitives.
3. In the **Device Initiated Power Management** area choose if the Report in the Identify page is supported or not.
4. Define start event by choosing Standby timer expires, STANDBY command is executed or Link layer remained in IDLE state more than [XXX] ms.
5. Specify time for Abandon request and Wake up after times as required and the Standby timer interval.

NCQ Command Settings Tab

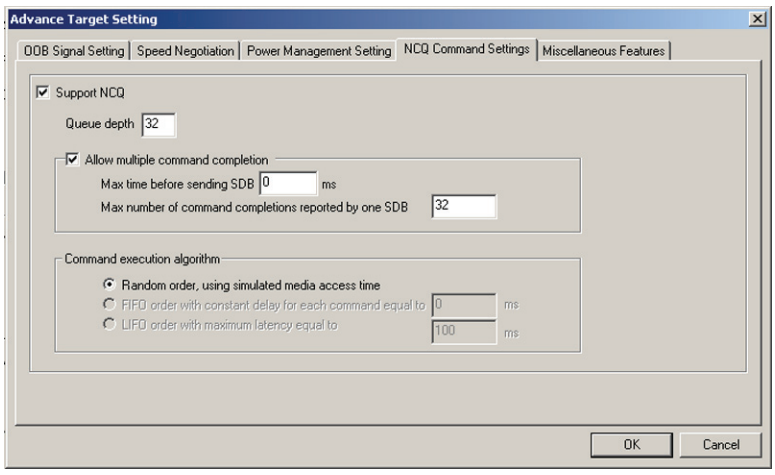


Figure 78. NCQ Commands

1. To enable **NCQ** commands, check **Support NCQ** and specify **Queue depth**.

2. If required, check **Allow multiple command completion** and enter values for **Max time before sending SDP** and **Max number of command completions reported by one SDB**.

Miscellaneous Features Tab

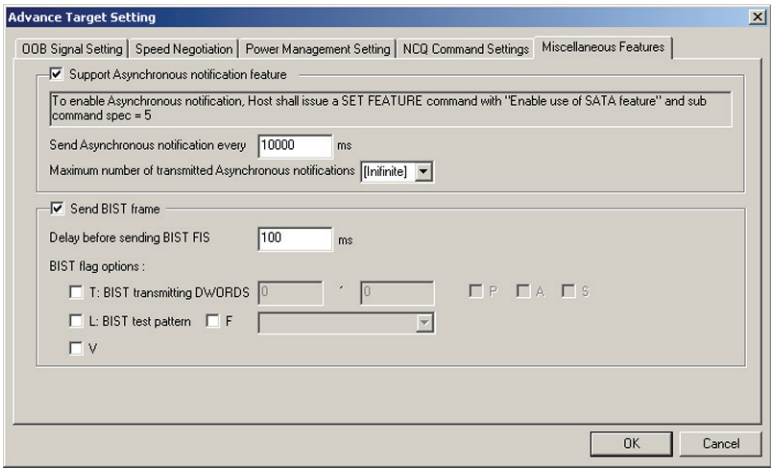


Figure 79. Miscellaneous Features

1. To enable Asynchronous notification, check the **Support Asynchronous notification feature** and specify the Asynchronous notification interval and the Maximum number of transmitted Asynchronous notifications.
2. To send a BIST frame, check **Send BIST frame** and enter a value for the required **Delay before sending BIST FIS**.
3. Choose **BIST flag options**.

Project Note

Click the **Notes** tab and enter a **Project Name** and a brief description of the Device Emulation project.

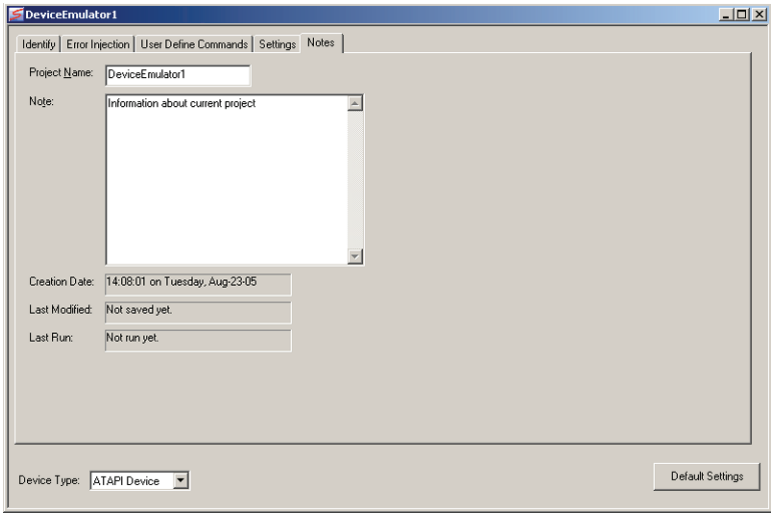


Figure 80. Project Note

Run Device Emulation Click the  **Activate Device** button to start emulation.

Note: You may run the analyzer while the device emulation is active to monitor bus traffic.

Display Manipulation

Viewer Display

The data viewer display may be configured to meet your individual test and viewing preference needs. Toolbars are available for quick access to data viewer display features. The following sections present a quick reference about the toolbars and details about the data viewer features.

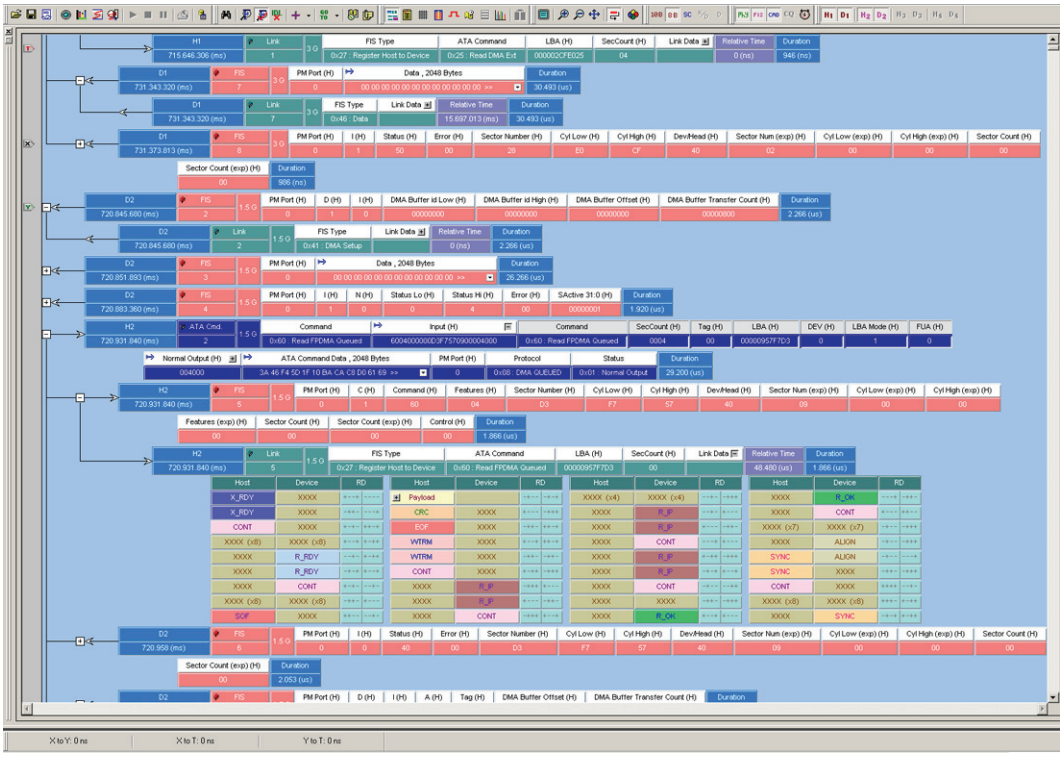


Figure 81 Viewer Data Display

Whenever a data capture is made it is displayed, by default, as a Packet View. You may, however display the same data in a Column View with transactions grouped for each active Port, a Text View similarly grouped for each active Port, a Spreadsheet view and a Histogram view.

Switching Views

To display the capture in any of the other available views you can make the selection on the View Type toolbar.

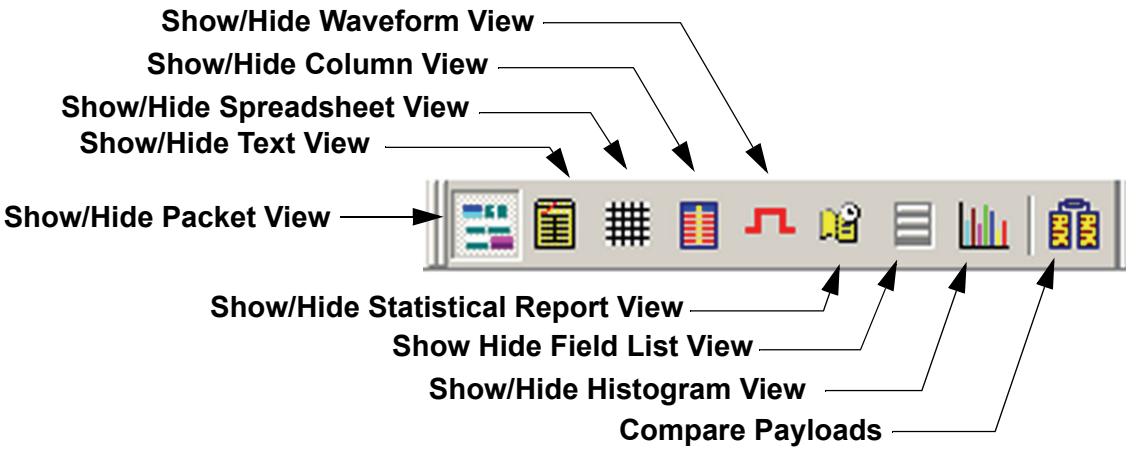


Figure 82. View Type Toolbar

Whenever you make a View selection, the selected view appears in a split window view with the packet view. To maximize the selected view display area click the **Show/Hide Packet View** button.

When scrolling through either display using the scroll bar, the corresponding display in the other view scrolls with it.

You may rearrange the tiling by clicking Window and choosing the tiling as Vertical or Horizontal according to your preference.

Text View

Text View displays the captured data interpreted as transaction frames grouped in columns by Port.

To display Text View of the current capture click **View** on the main tool bar and choose **Text View** or click the  button.

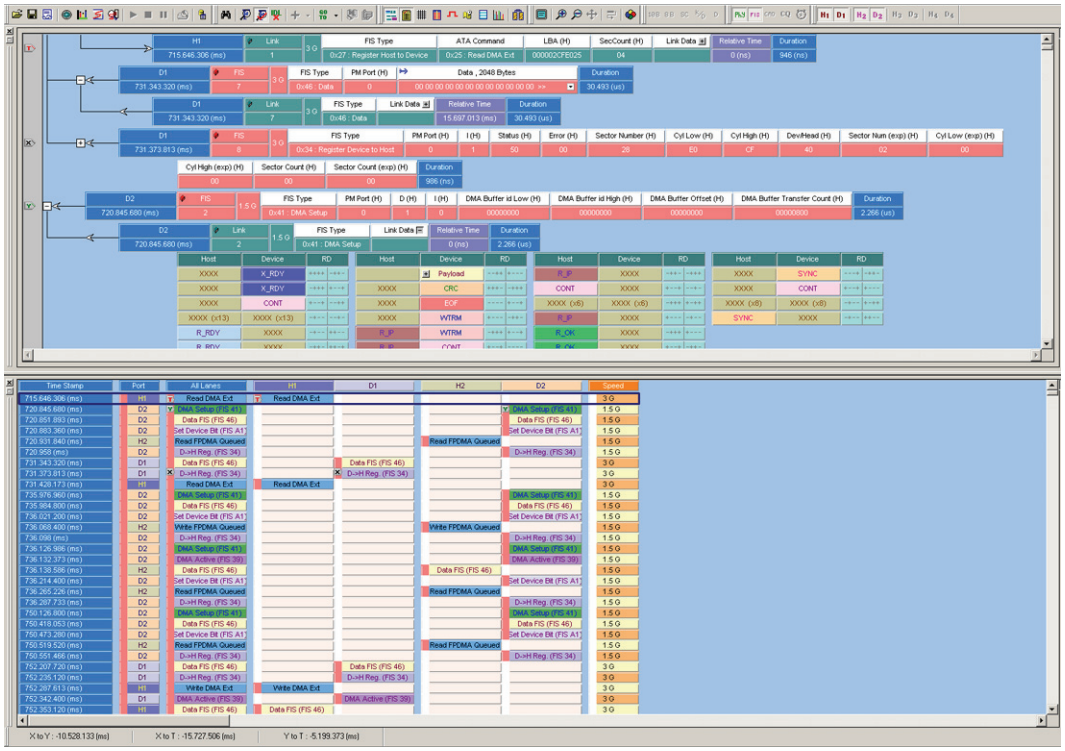


Figure 83. Simultaneous Packet View and Text View

Field List View

Whenever an additional view in addition to the packet view such as Text View is displayed you may also display a Field List View that displays field information in a list format.

To open a Field List View of the current capture click **View** on the main tool bar and choose **Field List View** or click the  button.

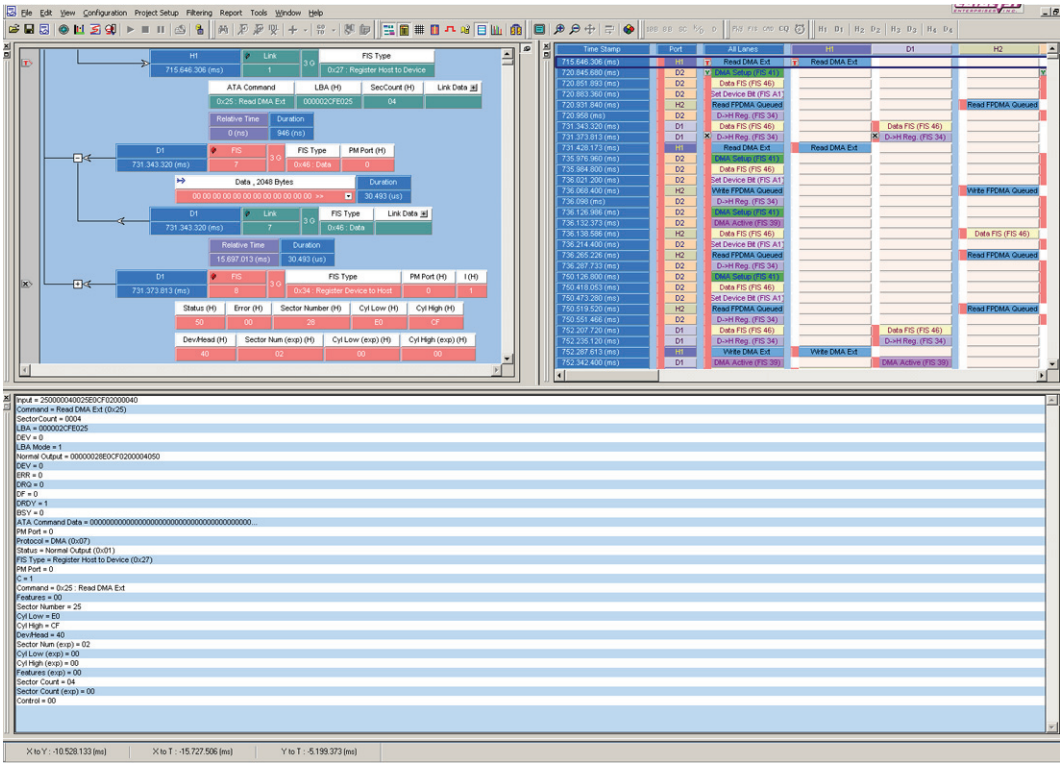


Figure 84 Field List View

Column View

Column View displays the captured data interpreted as **TBD** grouped in columns by Port.

To display Column View of the current capture click **View** on the main tool bar and choose **Column View** or click the  button.

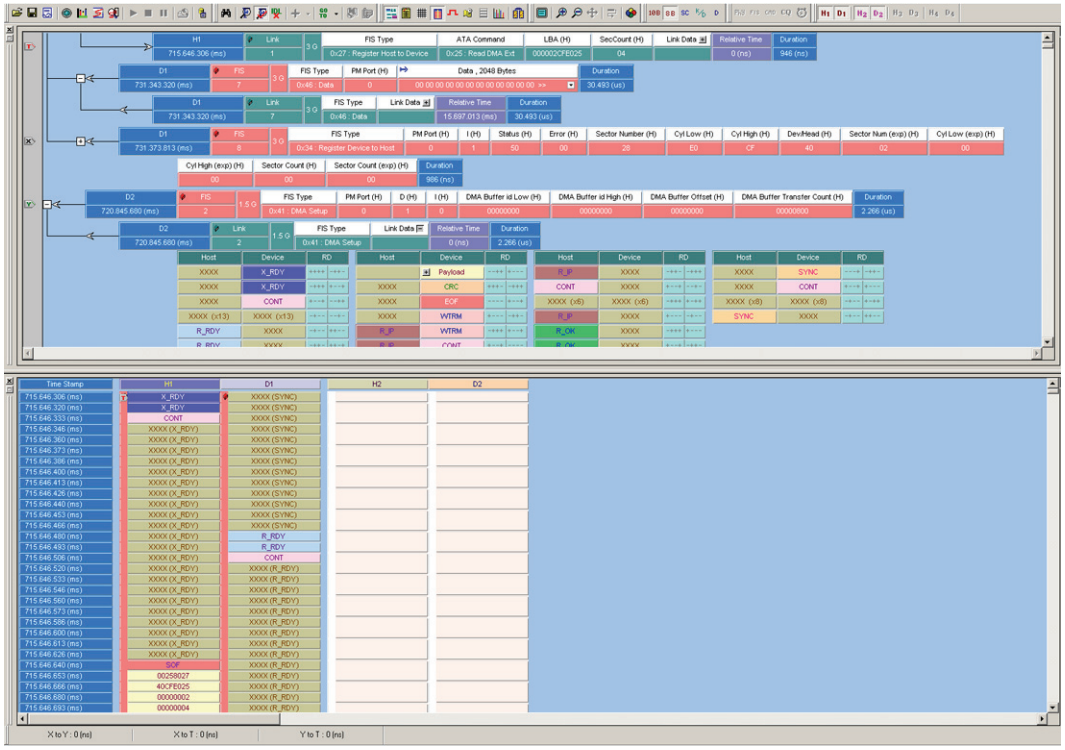


Figure 85. Simultaneous Packet View and Column View

Spreadsheet View

Spreadsheet View displays all of the Packet View fields in a time sequential spreadsheet format.

To display the Spreadsheet View of the current capture click **View** on the main tool bar and choose **Spreadsheet View** or click the  button.

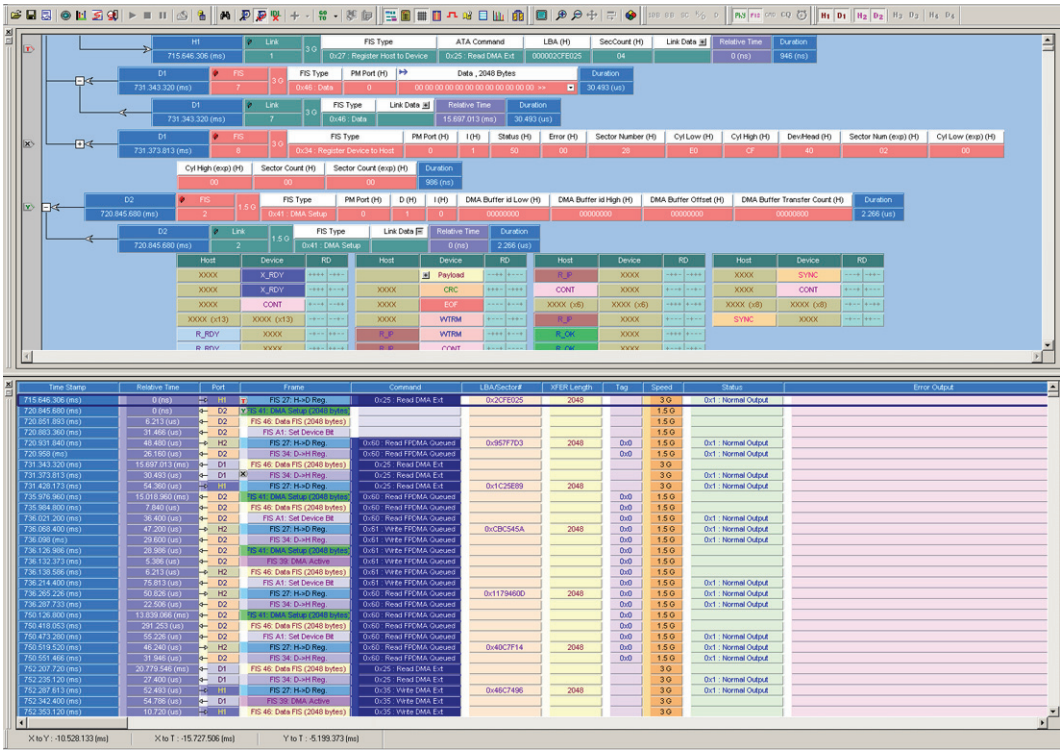


Figure 86. Simultaneous Packet View and Spreadsheet View

Histogram View

The Histogram View displays a histogram of Frame type transfers.

To display the Histogram View of the current capture click **View** on the main tool bar and choose **Histogram View** or click the  button.

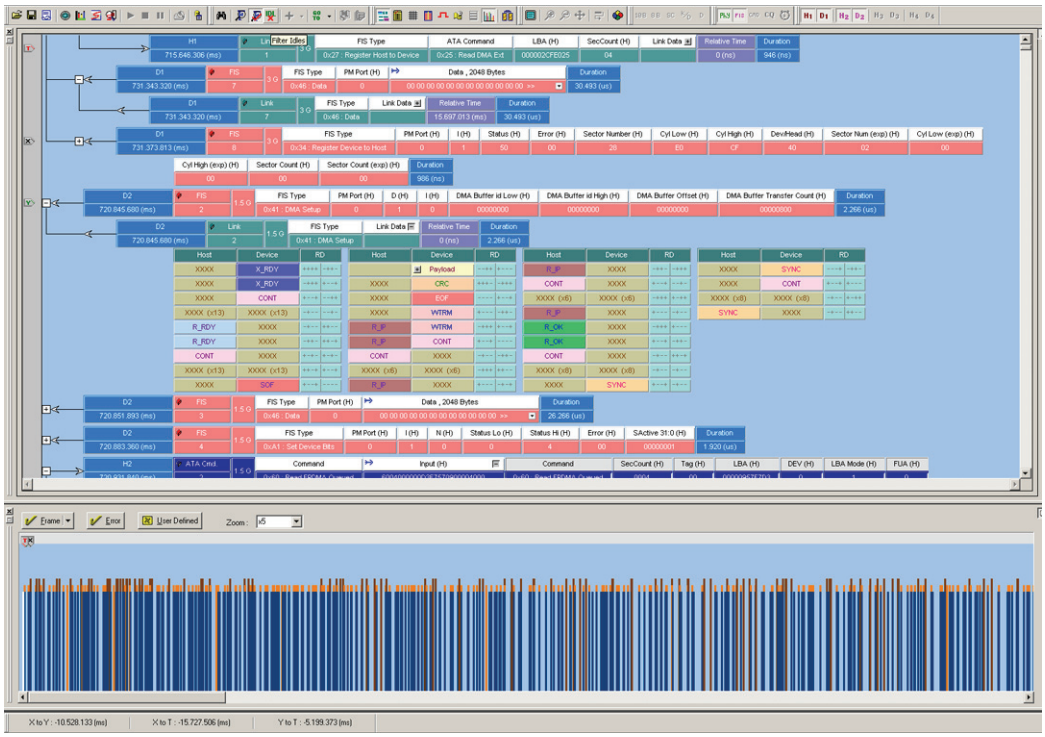
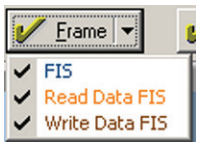


Figure 87. Simultaneous Packet View and Histogram View

Hide Frames

You may customize the histogram by including only the frame types of interest.

To choose the frame types to be included in the display, click the down arrow on the Frame button on the Histogram toolbar and check the frame types to be included in the histogram.



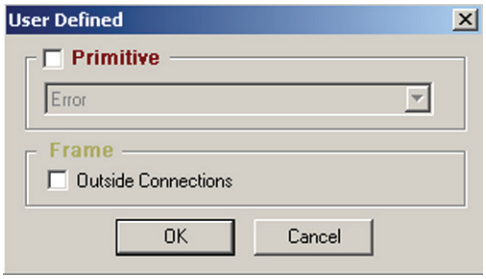
Hide Error Frames

Frames with errors are displayed as red. To hide error frames from the histogram click the  button.

User Defined

You may define additional items for inclusion in the Histogram by clicking the 

button to open the User Defined dialog.



You may choose to include Primitives and/or outside connections Frames.

To include Primitives check the **Primitive check** box click the down arrow on the Primitive list box and choose a Primitive.

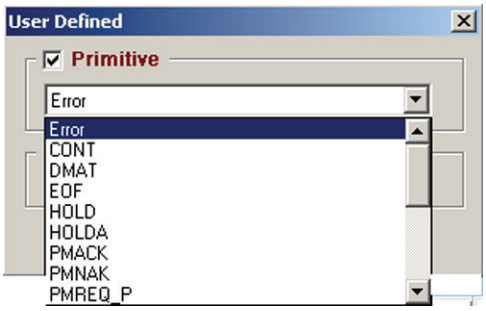


Figure 88. Choosing a Primitive

Then check the desired Connection Type option button and click **OK**.

Customize Display

Rename Port

You may rename each port for easy identification. To rename a port, right click the port ID in Text View or Column View and choose **Rename title of port** to open the **Rename Port** dialog.

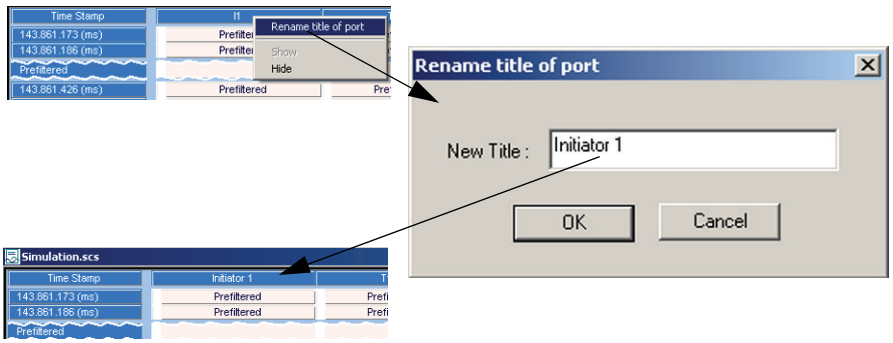


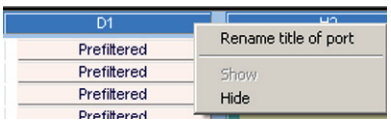
Figure 89. Rename Port

Show/Hide Port

You may simplify the Viewer display by hiding the captures of Ports that are not of current interest. All active ports are highlighted on the Show/ Hide Ports toolbar. Click the desired port button to hide the capture for that port.

Figure 90 Show/Hide Ports Toolbar

You may also show/hide a port by right clicking the Port name in column view and choosing **Hide**.



To show the port, right click in the column view port title area and choose a port to unhide.

Resize Columns

You may resize the columns in column view by clicking in the column boundary and dragging the boundary to a new position.

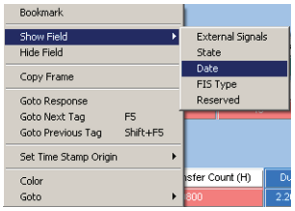
Rearrange Columns

You may rearrange columns by left clicking in the column title and then dragging the drag and drop icon  to a new position.

Show/Hide Field

You may simplify the Viewer display by hiding some of the fields that are of no current interest. You may hide the **Duration**, **Relative Time**, **External Signals** and **Packet number** fields by right clicking on the corresponding field title and choosing **Hide Field**.

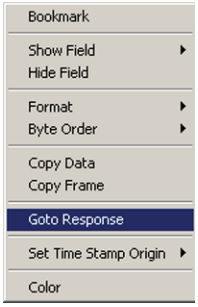
To restore a field to the display, right click in the Port number title field and choose the hidden field to be restored.



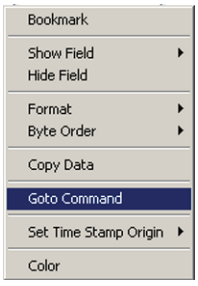
Note: Only the fields previously hidden will appear in the restore list.

Related Frames

Right click on a Command frame for SSP Frames or Register Device to Host for STP frames to open a short-cut menu and choose **Goto Response** to jump to the corresponding Response frame in the viewer.

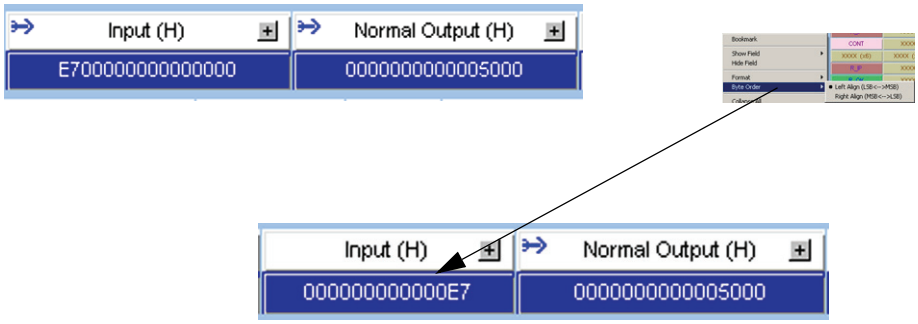


Similarly right click on a Response frame SSP Frames or Register Device to Host for STP frames to open a short-cut menu and choose **Goto Command** to jump to the corresponding Command frame in the viewer.



Byte Order

For reviewing enhancement you may change the byte order in fields marked by an arrow. Right click in the field Select **Byte Order** and choose the ordering.



Note: A blue arrow in the byte order field indicates that it has been changed.

Data View

To display data transactions in a data view, double click in the data area or right click in the data area and choose **Open as Data View**.

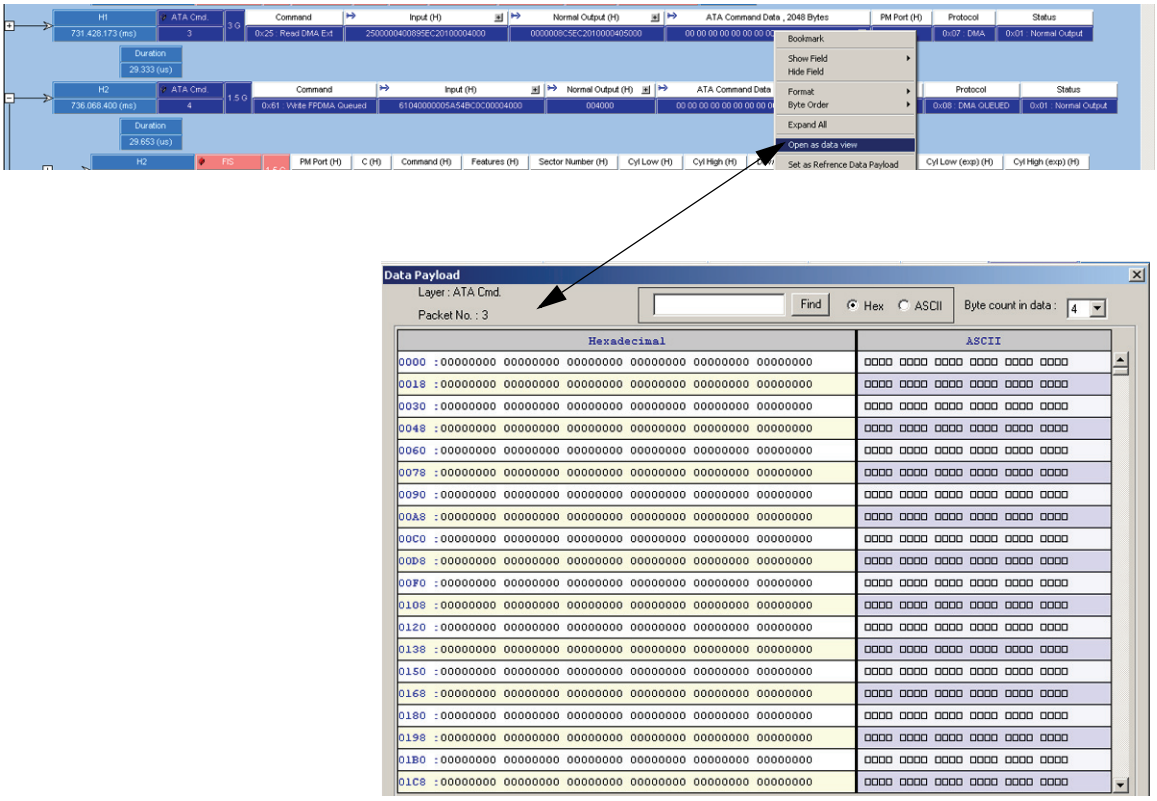


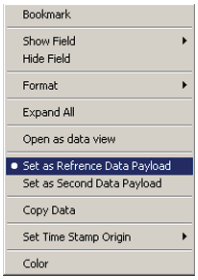
Figure 91. Selecting Open as Data View

Find Data Pattern To quickly locate a data pattern in the current frame enter the pattern in the Text Box and click the **Find** button.

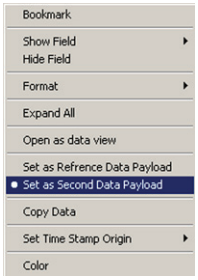
Figure 92.

Compare Payloads

To compare two payloads, locate the first transaction with a payload and right click in the data field and choose **Set as Reference Data Payload**.



Then scroll to the transaction with a payload that you wish to compare and right click in the data field and choose **Set as Second Data Payload**.



Click the **Compare Payloads** button on the Viewer Toolbar to perform the comparison.

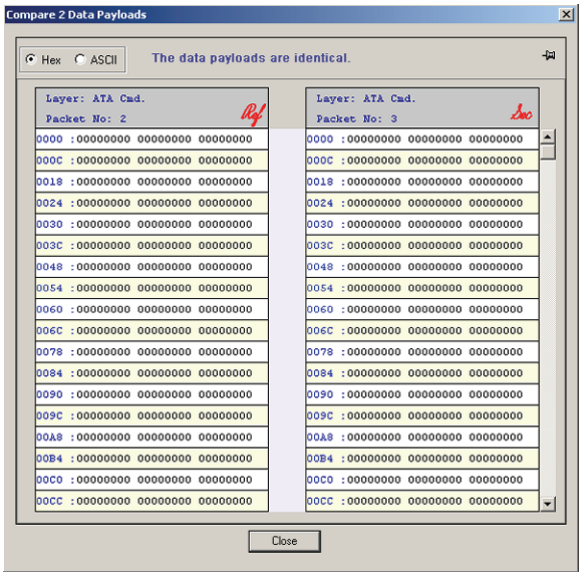


Figure 93 Payloads Compared

Waveform Display

You may enable a waveform display for all active ports and perform timing measurements by positioning timing cursors within the waveform display.

Click the **Show/Hide Waveform** button to enable the waveform display.

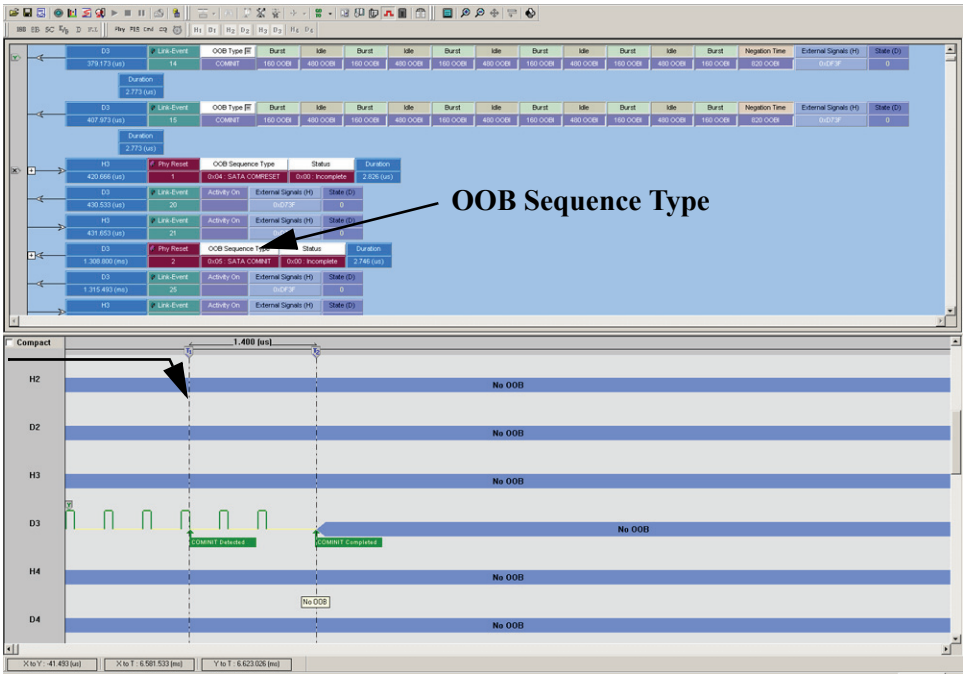


Figure 94 Waveform Display with Timing Cursors

Making a timing measurement Timing measurements are made with two timing cursors T1 and T2. Click the left mouse button in the gray bar on the top of the waveform display at a point where you wish to put the T1 cursor and the right mouse button where you wish to place the T2 cursor. The time difference between the cursors is displayed on a line connecting the two cursors.

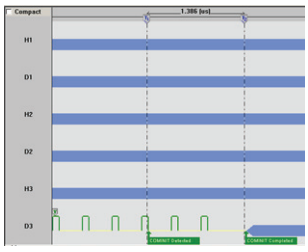


Figure 95 Timing Cursors Enabled

Compact Waveform View To see the OOB Sequence with speed negotiation (Hardware version 4 or later) and to see a 10x time scale expansion of the Waveform, check the **Compact View** checkbox in the Waveform View window.

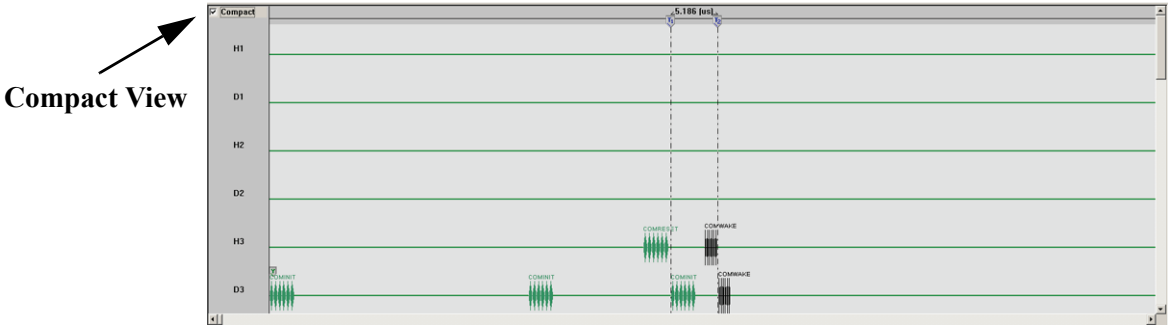


Figure 96 Compact Waveform View

Port Status

You may get an overview of the active Ports by clicking the Port Status button at the bottom of the screen.

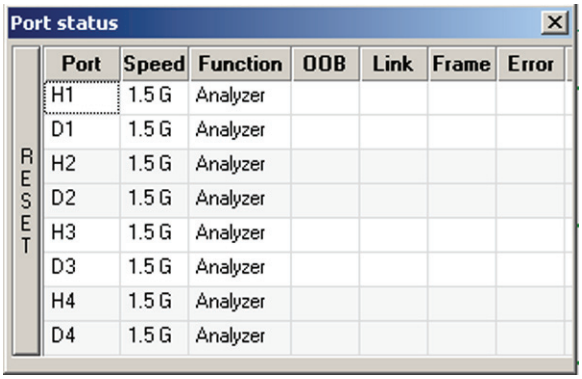


Figure 97 Port Status Display

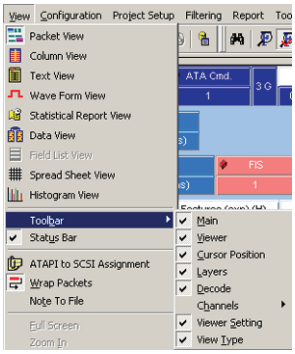
In addition to displaying OOB, Link Frame or Error a display showing the % buffer full will open whenever a trigger occurs.

Note: If samples are taken with more than one unit active, additional Port status windows will display.

Toolbars

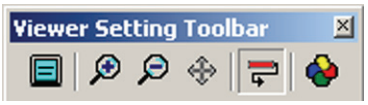
Enabling Tool Bars

To customize your Viewer Display workspace, you can enable and reposition the available toolbars. To display or hide toolbars from the **View** menu, select **Toolbars** and check or uncheck the appropriate toolbar**.**



Once enabled, the toolbars can be docked to the Viewer Display window or allowed to float on the windows desktop.

Viewer Setting Toolbar



The **Full Screen** button on the Viewer Setting Toolbar increases the data display area to full on the screen.

The **Zoom In** button on the Viewer Setting Toolbar magnifies the data display area of interest on the screen. Clicking this button in column or text frame view increases the column width only.

The **Zoom Out** button on the Viewer Setting Toolbar scales the data display area to display more data lines on the screen. Clicking this button in column or text frame view decreases the column width only.

The **Normal Zoom** button on the Viewer Setting Toolbar resets the zoom to default normal on the screen. Clicking this button in column or text view resets the column width only.

The **Wrap Packets** button on the Viewer Toolbar wraps the packet data in the display to eliminate the need for horizontal scrolling.

The **View Setting** button on the Viewer Setting Toolbar opens the Sample Viewer Configuration dialog.

Viewer Toolbar

The **Search** button opens the search dialog.

The **Filtering Setup** button opens the Filter dialog that allows you to specify the criteria for filtering the result.

The **Enable Disable Filtering** button toggles the result between a filtered and unfiltered view.

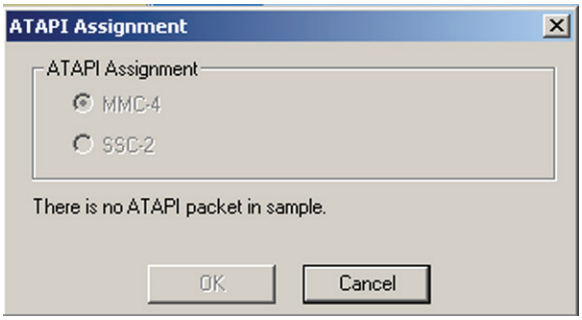
The **Filter Idle** button toggles the display to show/hide idle packets.

The **Expand/Collapse all Layers** button expands or collapses layers to simplify results display.

The down arrow on the **Go To** button allows the location of the cursors or specific packets.

The **Data Report** button displays the data report.

The **SCSI Spec Assignment** button displays the ATAPI to SCSI assignment dialog.


ATAPI Assignment
ATAPI Assignment
MMC-4
SSC-2
There is no ATAPI packet in sample.
OK
Cancel

Layers Toolbar

The **Show/Hide Waveform View** button to display or hide the waveform display.

The **Show/Hide Physical Packet** button toggles the display of physical layer packets.

The **Show/Hide CMD packet** button on toggles the display of the CMD packets.When “ON”, only the command layer is displayed.

The **Show/Hide FIS packet** button toggles the display of FIS layer packets. When “OFF” the FIS layer and its links are hidden.

The **Show/Hide Command Queue** button displays queued commands.

The **Order/Reorder** toggles the time order of packets.

Decode Toolbar

The **Decode Toolbar** allows the control of the following encoding and scrambling features.

Clicking the **10B** button displays the payload data as 10 bit encoded data.

Clicking the **8B** button displays the payload as 8 bit scrambled or unscrambled data depending on the Scrambled setting.

Clicking the **SC** button selects scramble/unscramble for the 8 bit payload data.

To view corresponding Unscrambled and Scrambled payload data values instantaneously, position the mouse pointer over a data field.

Running Disparity indication

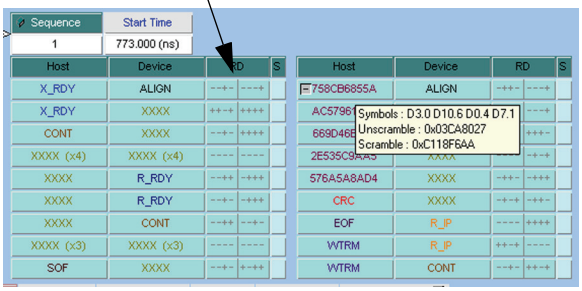


Figure 98 Payload Data Display

The **Symbol Notation** button decodes the 8 bit or 10 bit encoded data in List View.

The **Show Data** button displays the payload data values in Column View.

Filter

The Filtering patterns option allows you to modify data in the Viewer display to exclude Packets with a set of user defined patterns and save the result in a file.

*.sts sample file

In order to set up filtering, you must have a viewer display open.

Filter Setup

To display the Filter setup dialog, click the **Filter** button on the Viewer toolbar or, choose Filtering from the Filtering menu.

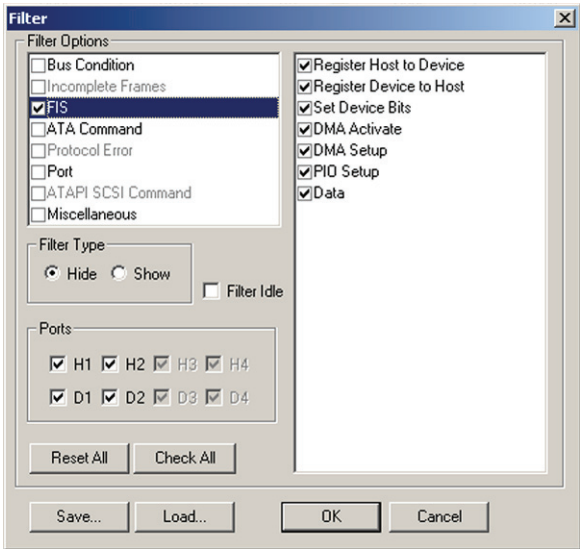


Figure 99 Filter Setup Dialog

Each of the items shown in the Filter Options window can be selected or deselected for filtering by checking or unchecking a corresponding check box. Items not in the current Sample are shaded.

Note 1. When a group is selected, all of its child items are also selected.

Note 2. Only packets captured at run time are available to be selected for filtering.

Filter Type

You may choose to show or hide the items selected for filtering by checking the **Show** or **Hide** option buttons as appropriate.

Filtering Direction

Items may be selected for filtering in a single direction or both directions by checking the corresponding Port. By default, all of the ports are enabled. Uncheck the port checkboxes for ports that you do not wish to include in the filter.

Filter Idle When selected on the Filter Type, depending on Hide/Show selection Idle packets in the Sample Viewer will be shown or hidden.

You may quickly filter idles by clicking the **Filter Idle** button. Note that this button toggles between Show and Hide Items.

Click the **Filter Enable** button on the display menu bar to toggle between a Filtered and unfiltered display.

Selectable Filter Options

- Bus Condition
- Incomplete Frames
- FIS
- ATA Command
- Protocol Error
- Port
- ATAPI SCSI Command
- Miscellaneous
- Filter Idle

Bus Condition When selected, depending on the Filter Type, the Hide/Show selection will show or hide captured Bus Conditions in the Sample Viewer.

Incomplete Frames When selected, depending on the Filter Type, the Hide/Show selection will show or hide Incomplete Frames in the Sample Viewer.

FIS When selected, depending on the Filter Type, the Hide/Show selection will show or hide captured FIS items in the Sample Viewer.

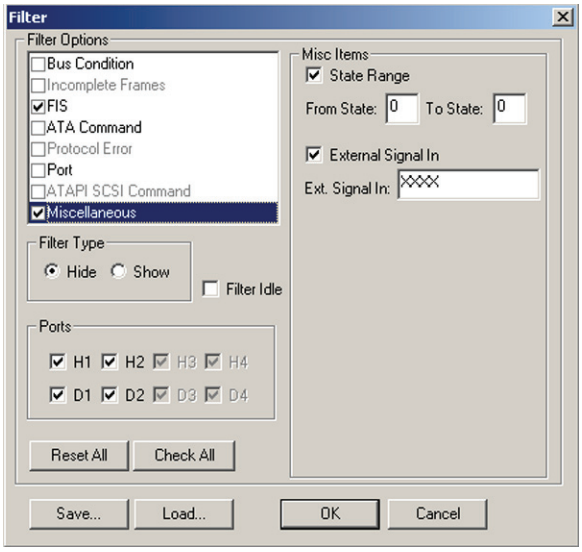
ATA Command When selected, depending on the Filter Type, the Hide/Show selection will show or hide captured ATA commands in the Sample Viewer.

Protocol Error When selected, depending on the Filter Type, the Hide/Show selection will show or hide captured packets with the specified Protocol Errors in the Sample Viewer.

Port When selected, depending on the Filter Type, the Show/Hide selection, will show or hide packet traffic for the selected port.

ATAPI SCSI Command When selected, depending on the Filter Type, the Show/Hide selection will show or hide ATAPI SCSI commands.

Miscellaneous When you choose Miscellaneous an additional dialog is displayed allowing you to specify the filtering of State Range and/or External Signal In.



Save Filter Setup Once you have set up a Filter configuration you may save it as a Filter file (*tfl) by clicking **Save**. You may then use it on a different capture by clicking **Load** in the Filter dialog.

Using the Cursors and Bookmarks

Cursors The data viewer display incorporates three cursors labeled **X, Y** and **T .** All of the cursors are initially overlaid and positioned at location 0, which is the trigger position of the display. The Trigger, or **T**, cursor is the measurement reference and is always locked at location 0 in the display.

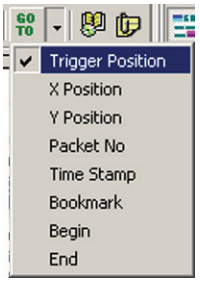
Positioning the X Cursor To position the X-Cursor within the viewer data display, click the left mouse button in the gray bar on the left side of the sample viewer next to the line where you wish to see the cursor.

Positioning the Y Cursor To position the Y-cursor within the viewer data display, click the right mouse button in the gray bar on the left side of the sample viewer next to the line where you wish to see the cursor.

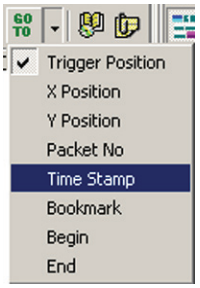
Note: You may also left click to set the X-cursor and right click to set the Y cursor in the frame and the column view by clicking in the narrow strip on the very left side of a cell. Similarly you may set the cursors in the Waveform view by left and right clicking at the beginning of a waveform.

Time differences between the cursors are displayed in the cursor position toolbar. To display the cursor position toolbar, select Toolbar from the view menu and choose Cursor Position.

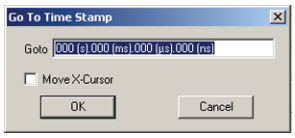
Locate Cursors To quickly locate any of the cursors within the data viewer display, click the **Go To** button and choose the cursor to locate. You may also locate the cursors by selecting Go To from the Edit menu and choosing the cursor to locate.



Go to time stamp To locate a timestamp click the **Go To** button and choose Timestamp.



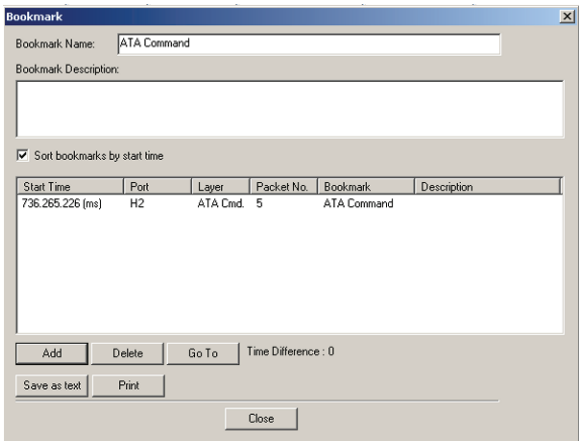
Enter a time stamp value in the Go To Timestamp dialog and click **OK.**



Bookmarks

Bookmarks is a convenient way to mark a point in the data viewer display by name, such that you can rapidly return to that point. To create a bookmark, right click the mouse in the data viewer area on a packet where you wish to place the bookmark.

Click **Bookmark** from the fly out menu to open the Bookmark Comment Dialog.



Enter a description for the bookmark and click the **Add** button. Repeat for additional bookmarks.

Finding a Bookmark

To find a bookmark in the data viewer display, right click the mouse in the sample viewer and select **Bookmark.**

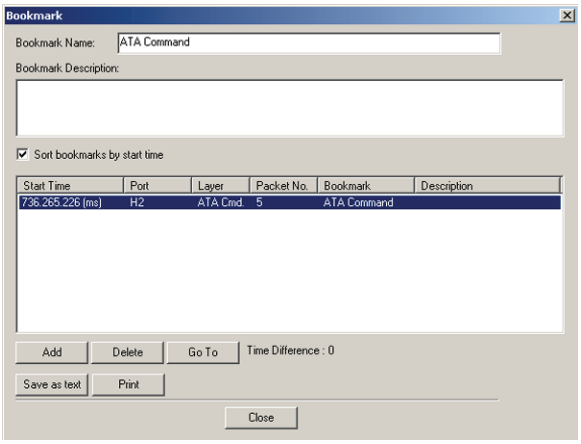


Figure 100 Go To Bookmark Dialog Box

Highlight the bookmark that you wish to go to and click the **Go To** button, or double-click on the selection**.**

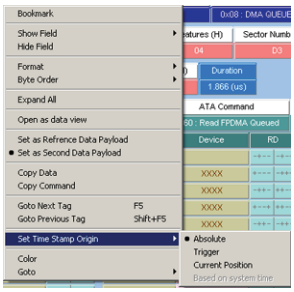
Figure 101 Bookmark Found Example in Data Viewer Display

Bookmark description

To get a quick description of a displayed bookmark, position the tool tip over a bookmark. The name and description of the bookmark will display.

Set Time Stamp Origin

Right click in the sample viewer to open the fly out menu:



Highlight **Set Time Stamp Origin** and choose either Absolute, Trigger, Current Position or Based on system time.

Search

The search option permits you to examine any data capture file to quickly locate the desired packet/data pattern.

To perform an initial search, click the **Search** button to open the search setup dialog as shown in Figure 102. You may also perform the search by selecting Search from the Edit menu.

Note: Only items captured in the sample file are enabled for search.

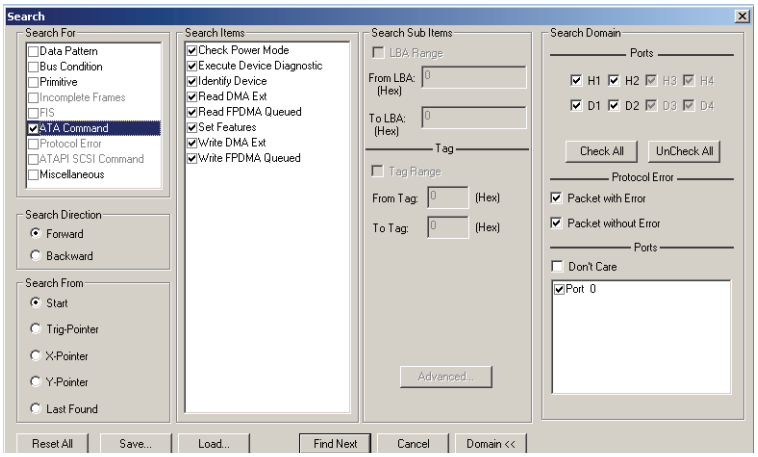
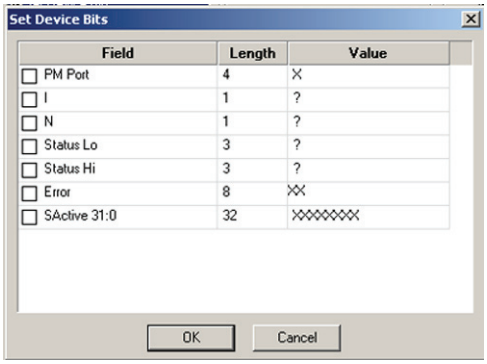


Figure 102 Search Parameter Definition Dialog Box

Search For Choose a category to search in the **Search For** window. Each of the search categories offers additional choices in the **Search Items window** to refine the search. Check the desired items for the selected category.

Advanced options Some of the Search Categories offer Advanced options for search. To set these options, highlight the search item in a category and click the **Advanced** button to open the Advanced options dialog.



Set the desired options and click **OK.**

Search direction Choose either Forward or Backward direction in which to perform the search.

Search From Choose a starting point to begin or continue a search.

- Start of the sample file
- Trigger Pointer
- X Pointer
- Y Pointer
- Last Found

Packets With or Without Error You may refine the search to locate packets with an error or without an error.

Data Pattern Search for Data Pattern allows you to search for a specific Data Type, Pattern and Length.

- Data Pattern Only
- Data Payload Length Only
- Data Pattern and Data Payload Length

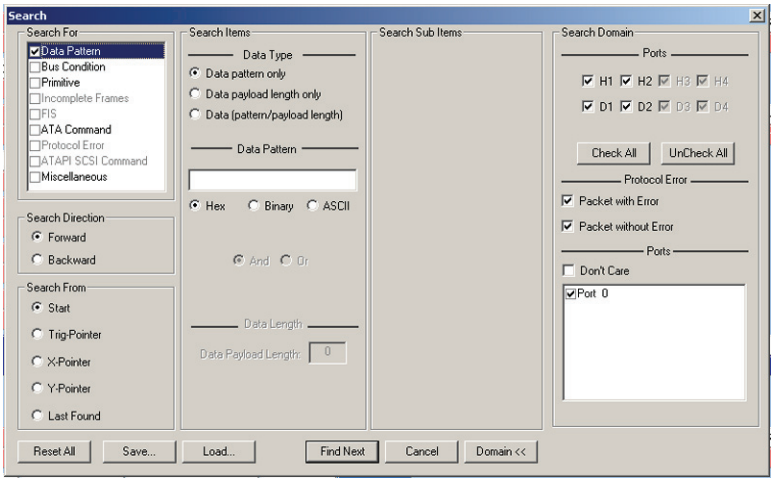


Figure 103 Search for Data Pattern

Search domain Click the Domain button and choose a search domain from All ports or a specific port.

Protocol Error Choose Packet with error or Packet without error

Click the **Find Next** button to perform the specified search.

Note: When searching for Protocol Errors in column view, you cannot search for a specific Protocol Error type. Search will return any protocol error.

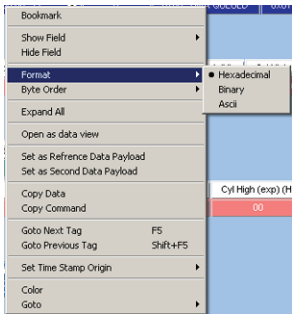
You may continue to search the output file using **Next Search** or **Previous Search** for the same pattern until you redefine the data capture search parameters.

Save Search Setup Once you have set up a Search configuration you may save it as a Search file (*tsh) by clicking **Save**. You may then use it on a different capture by clicking **Load** in the Search dialog.

Interpretation Aids

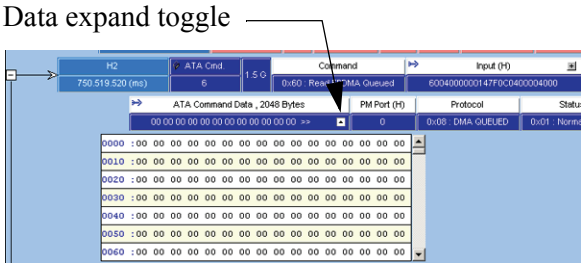
Choose Data Format

You may display data values either in hexadecimal (default) or binary. To choose data format right click the mouse over a data field, choose **Format** and then the format.

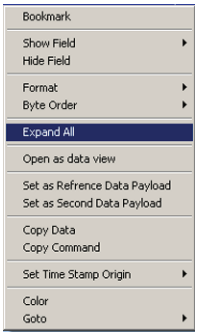


Show all Data

To display all captured data, click the data expand toggle arrow in a data field where you wish to examine the data in detail.



You may expand or collapse all data fields globally. To expand all data fields, right click the mouse in a data field and choose **Expand All.**



To collapse all fields right click the mouse in a data field and choose **Collapse All.**

Set Port Alias

Port Alias allows you to assign a meaningful name to each port to assist in interpreting the results displayed in the sample view.

To assign port names in an open sample view, click **Configuration** on the tool bar and choose **Set Port alias**.

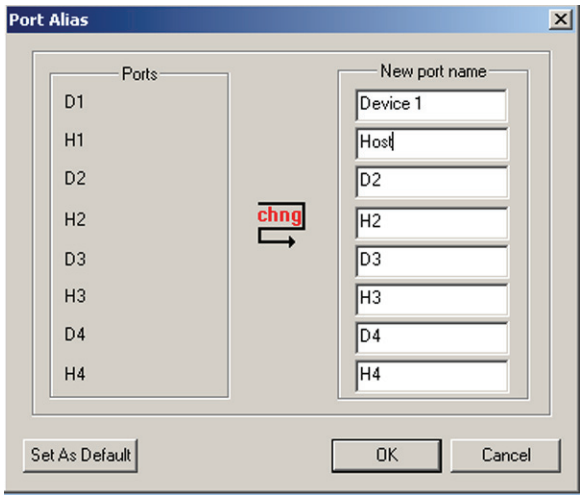


Figure 104 Assign Port Name

Assign a meaningful name to each port in use and click **OK**. The assigned names replace the port numbers in the sample view.

If you elect to save the capture sample file, the assigned port names will be saved together with the result so that when you open the sample file later, the assigned names will be retained.

Software Settings

Software settings allows you to define template files for new Analyzer projects, to specify how sample files will appear when opened and to set ATAPI Spec Assignment.

To perform software settings in an open sample view, click **Configuration** on the tool bar and choose **Software Settings**.

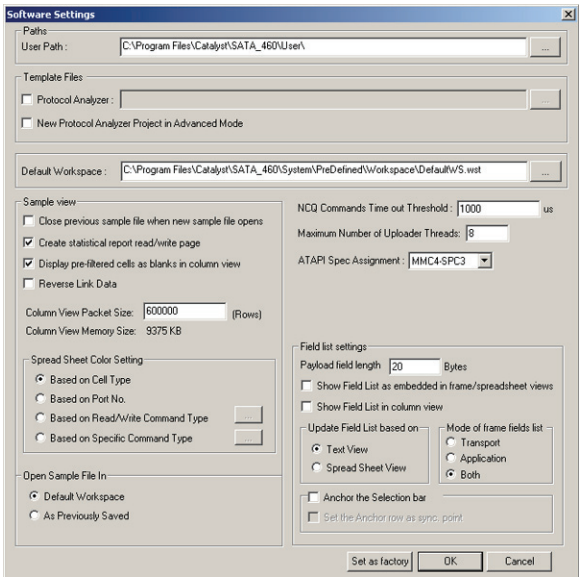


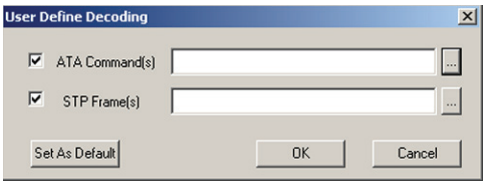
Figure 105. Software Settings Dialog

Set the desired options and click **OK**.

User Defined Decoding

User defined decoding allows you to create a definition file to interpret commands and frames that are not in the standard set recognized by the software.

Click **Configuration** and choose **User Defined Decoding** to open the User Defined Decoding dialog.



Check **ATA Commands** and/or **STP frames.** Click the ellipses next to a command type text box to display the **Open** dialog. Choose an appropriate script file and click **Open**.

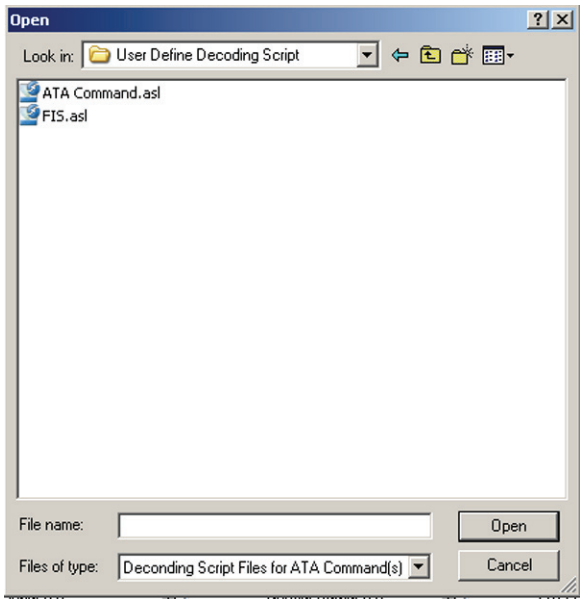


Figure 106 Choosing a Script File

Display Configuration

The Analyzer ships with a default display configuration of field and viewer settings. You may, however, define your own field and viewer settings for a particular testing scenario.

The Sample Viewer Configuration dialog allows the user to change the following display settings:

- Field settings
  - Data format
  - Field header text color
  - Hide/Show field
- Viewer Settings
  - Change fonts
  - Wrap packets
  - Time Stamp Origin
  - Enable/Disable tool tip
- Save Display Configurations in a file
- Load Display Configuration settings from a file
- Factory Setting (Restores Default Settings)

To customize your display, click the **Configuration** button on the Viewer toolbar, or select **Sample Viewer Configuration** from the Configuration menu to open the Sample Viewer Configuration dialog.

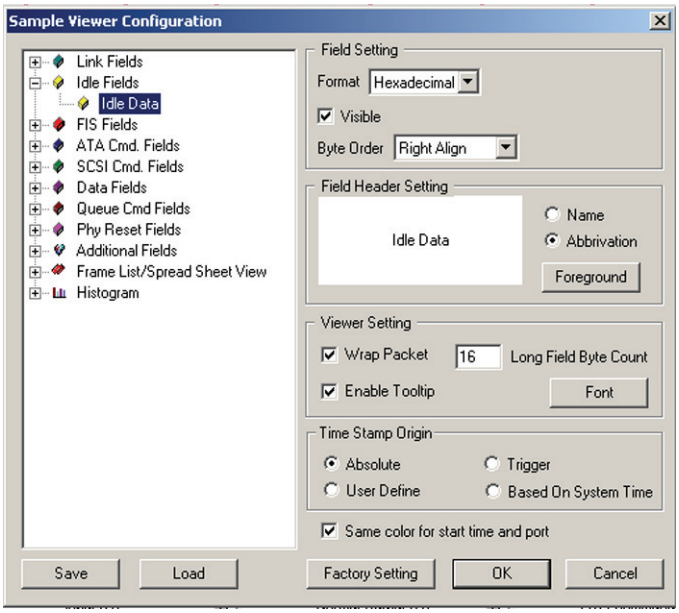
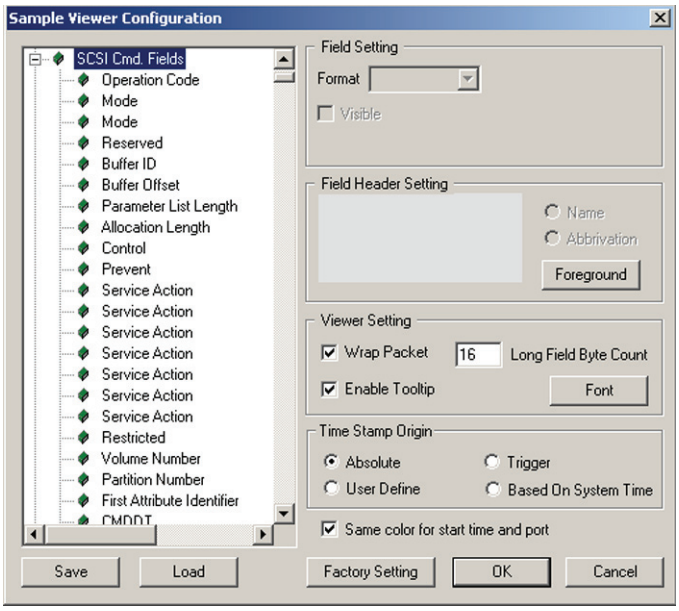


Figure 107 Display Configuration Dialog Box

Field Settings

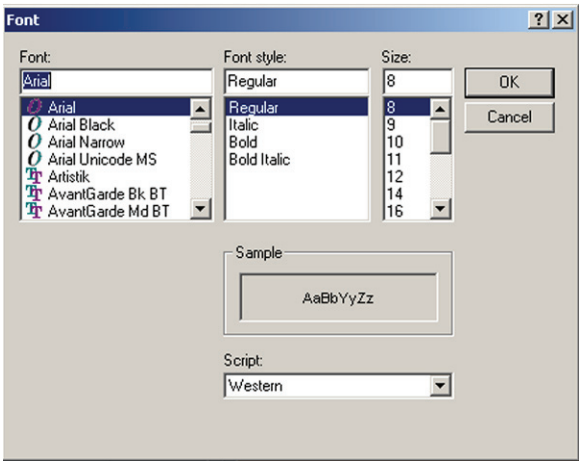
To view a packet field, select a field from the packet field tree and check the Visible box. Uncheck it to hide the field. To change the data format of a packet field, select the field and choose a data format from the Format drop-down list.



To change the color of the text in a packet field header, select a field from the packet field tree and click the **Foreground** button.

Choose an appropriate color and click **OK**

To change display fonts, click the **Font** button to open the Font dialog box.



Choose the desired font, font style, size and click **OK.**

Viewer Settings

Check the Wrap Packet box to enable the wrapping of packets in the display.

Check the Enable Tooltip box to enable tool tips for packet fields.

To change the length of long byte fields displayed, enter a number of bytes to display in the Long Field Byte Count size box.

Check **Absolute Trig** to display trigger in real time. If left unchecked the trigger position is t=0 with samples before trigger shown as a (-) number and after trigger as a (+) number.

When finished, click **OK** to save changes and close the Display Configuration dialog.

Save/Load Settings

You may save the customized configuration settings in a *.cfg file by clicking the **Save** button and completing the Save As procedure. To load a previously saved configuration file click **Load** and choose an appropriate file.

Statistical Report

Whenever a captured sample is displayed in the sample viewer, a **Statistical Repor**t selection in the **Report** menu and a **Statistical Report Button** on the viewer toolbar are enabled. You may create a Statistical Report for the entire capture or a select portion of it as desired.

To display a Statistical Report, click the **Statistical Report** button on the viewer toolbar, or select **Statistical Report** from the **Report** menu.

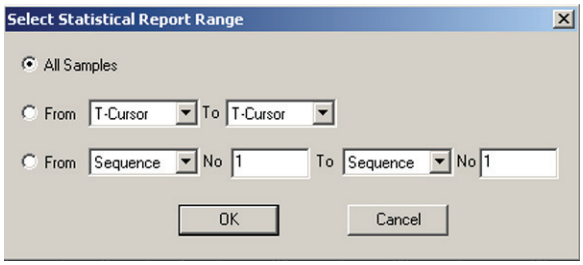
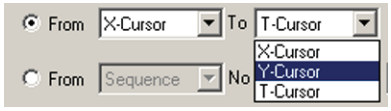


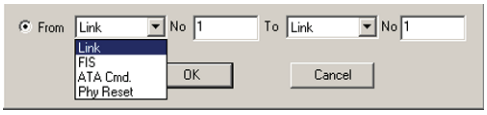
Figure 108 Statistical Report Range Dialog

The default statistical report is generated using all samples. You may, however, set a specific Statistic report range between defined cursor positions or events.

Report Between Cursors Click the option button next to the **From** cursor selection drop down list. Then click the **From** down arrow and choose the 1st. cursor, click the **To** down arrow to choose the 2nd cursor and click **OK.** The resulting report is limited to the capture between the cursors.



Report Between Events Click the option button next to the **From** the event selection dropdown list. Then click the **From** down arrow to choose the 1st event and enter the number of its occurrence. Next click the **To** down arrow to choose the 2nd event, and enter the number of its occurrence.



Click **OK.** The resulting report will be limited to the capture between the defined events.

Statistical Report Content

A complete statistical report consists of the following reports a that are accessed by clicking on the corresponding tab in the dialog:

- General
- Primitive
- Bus Condition
- FIS
- ATA Command
- Read/Write Command
- Protocol Error
- Performance
- Others
- Queued Commands
- ATAPI

Note: Results are displayed only for items that have been captured in the sample.

Report Options

Some report categories offer options to display only items of interest.These report categories incorporate drop-down list boxes offering pre-defined and custom options. For details see “Formatting the Statistical Report View” on page 146

General Report

To display the General report, click the **General** tab.

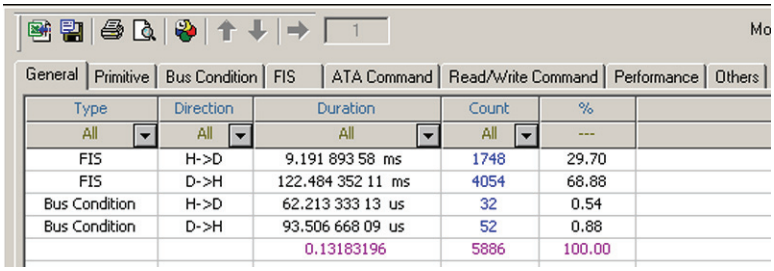


Figure 109 Sample Statistical Report

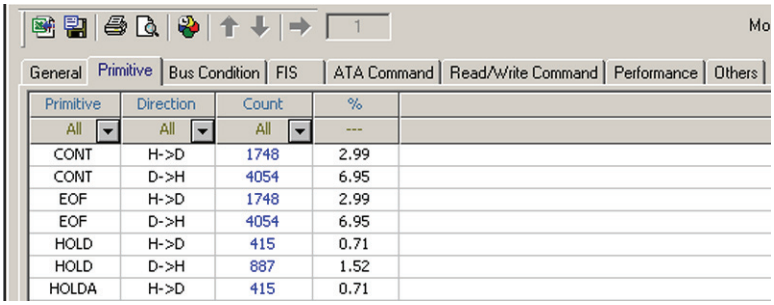
The General report displays the report data in columns with the following information:

- Type
- Direction
- Duration
- Count (Number of occurrences)
- % of total count

Primitive Report View

To display the Primitive Report, click the **Primitive** tab. The Primitive report displays information in the following columns:

- Primitive
- Direction
- Count (Number of occurrences)
- % of total count

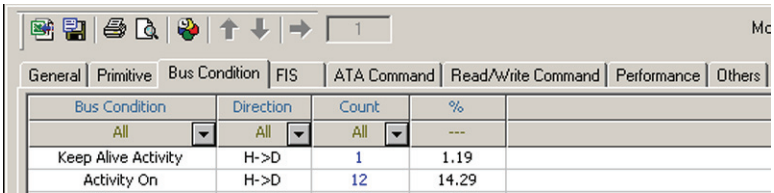


Primitive	Direction	Count	%
All	All	All	---
CONT	H->D	1748	2.99
CONT	D->H	4054	6.95
EOF	H->D	1748	2.99
EOF	D->H	4054	6.95
HOLD	H->D	415	0.71
HOLD	D->H	887	1.52
HOLDA	H->D	415	0.71

Bus Condition Report View

To display the Bus Condition Report, click the **Bus Condition** tab. The Primitive report displays information in the following columns.

- Bus Condition
- Direction
- Count (Number of occurrences)
- % of total count

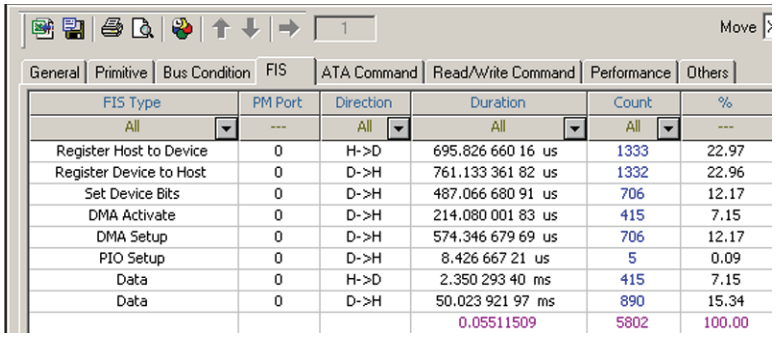


Bus Condition	Direction	Count	%
All	All	All	---
Keep Alive Activity	H->D	1	1.19
Activity On	H->D	12	14.29
Activity On	D->H	12	14.29
COMINIT/COMRESET	H->D	1	1.19
COMINIT/COMRESET	D->H	28	33.33
COMWAKE	H->D	12	14.29
COMWAKE	D->H	12	14.29
COMSAS	H->D	6	7.14
		84	100.00

FIS Report View

To display the FIS Report, click the **FIS** tab. The FIS Report view displays information in the following columns:

- FIS Type
- PM Port
- Direction
- Duration time (Accumulative)
- Count (Number of occurrences)
- % of total count

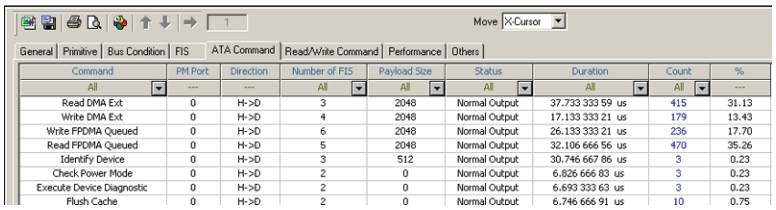


FIS Type	PM Port	Direction	Duration	Count	%
All	---	All	All	All	---
Register Host to Device	0	H->D	695.826 660 16 us	1333	22.97
Register Device to Host	0	D->H	761.133 361 82 us	1332	22.96
Set Device Bits	0	D->H	487.066 680 91 us	706	12.17
DMA Activate	0	D->H	214.080 001 83 us	415	7.15
DMA Setup	0	D->H	574.346 679 69 us	706	12.17
PIO Setup	0	D->H	8.426 667 21 us	5	0.09
Data	0	H->D	2.350 293 40 ms	415	7.15
Data	0	D->H	50.023 921 97 ms	890	15.34
			0.05511509	5802	100.00

ATA Command Report View

To display the Command Report view, click the **Command** tab. The Command Report view displays information in the following columns:

- Command
- PM Port
- Direction
- Number of FIS
- Payload Size (Dwords)
- Status
- Duration
- Count (Number of occurrences)
- % of total count
- Timeout

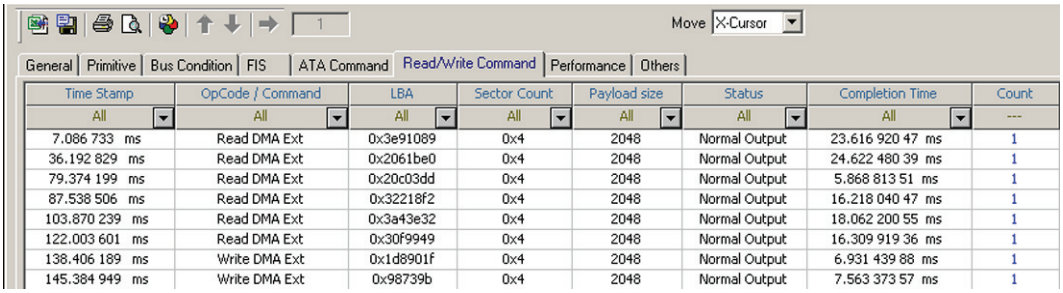


Command	PM Port	Direction	Number of FIS	Payload Size	Status	Duration	Count	%
All	---	---	All	All	All	All	All	---
Read DMA Ext	0	H->D	3	2048	Normal Output	37.733 333 59 us	415	31.13
Write DMA Ext	0	H->D	4	2048	Normal Output	17.133 333 21 us	179	13.43
Write FPDMA Queued	0	H->D	6	2048	Normal Output	26.133 333 21 us	236	17.70
Read FPDMA Queued	0	H->D	5	2048	Normal Output	32.106 666 56 us	470	35.26
Identify Device	0	H->D	3	512	Normal Output	30.746 667 86 us	3	0.23
Check Power Mode	0	H->D	2	0	Normal Output	6.826 666 83 us	3	0.23
Execute Device Diagnostic	0	H->D	2	0	Normal Output	6.693 333 63 us	3	0.23
Flush Cache	0	H->D	2	0	Normal Output	6.746 666 91 us	10	0.75

Read Write Command Report View

To display the Read/Write Command Report view, click the **Read/Write Command** tab. The creation of this page can be enabled or disabled under Software Settings, see page 131. The Read/Write Command Report view displays information in the following columns:

- Time Stamp
- OpCode/Command
- LBA
- Sector Count
- Payload size
- Status
- Completion Time
- Count

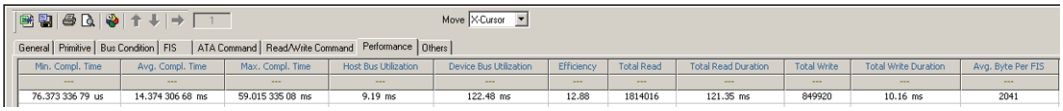


Time Stamp	OpCode / Command	LBA	Sector Count	Payload size	Status	Completion Time	Count
All	All	All	All	All	All	All	---
7.086 733 ms	Read DMA Ext	0x3e91089	0x4	2048	Normal Output	23.616 920 47 ms	1
36.192 829 ms	Read DMA Ext	0x2061be0	0x4	2048	Normal Output	24.622 480 39 ms	1
79.374 199 ms	Read DMA Ext	0x20c03dd	0x4	2048	Normal Output	5.868 813 51 ms	1
87.538 506 ms	Read DMA Ext	0x32218f2	0x4	2048	Normal Output	16.218 040 47 ms	1
103.870 239 ms	Read DMA Ext	0x3a43e32	0x4	2048	Normal Output	18.062 200 55 ms	1
122.003 601 ms	Read DMA Ext	0x30f9949	0x4	2048	Normal Output	16.309 919 36 ms	1
138.406 189 ms	Write DMA Ext	0x1d8901f	0x4	2048	Normal Output	6.931 439 88 ms	1
145.384 949 ms	Write DMA Ext	0x98739b	0x4	2048	Normal Output	7.563 373 57 ms	1

Performance Report View

To display the Performance Report view, click the **Performance** tab. The Performance Report view displays information in the following columns:

- Minimum Completion time
- Average Completion time
- Maximum Completion time
- Host Bus Utilization
- Device Bus Utilization
- Efficiency (%)
- Total Read (DWords)
- Total Read Duration
- Total Write DWords
- Total Write Duration (time)
- Average Byte Per FIS

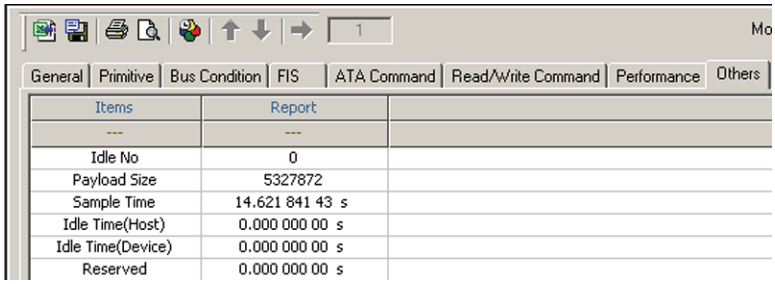


Min. Compl. Time	Avg. Compl. Time	Max. Compl. Time	Host Bus Utilization	Device Bus Utilization	Efficiency	Total Read	Total Read Duration	Total Write	Total Write Duration	Avg. Byte Per FIS
---	---	---	---	---	---	---	---	---	---	---
76.373 336 79 us	14.374 306 68 ms	59.015 335 08 ms	9.19 ms	122.48 ms	12.88	1814016	121.35 ms	849920	10.16 ms	2041

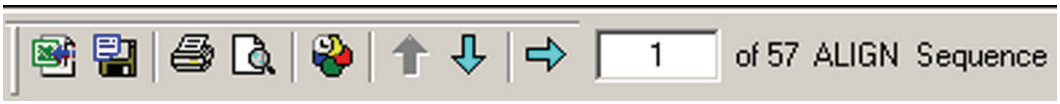
Others Report View

To display the Others Report view, click the **Others** tab. The Others Report view displays information in the following columns:

- Items
- Reports



Statistical Report Toolbar



The Statistical report toolbar provides the following functions accessible by buttons on the toolbar:

- Export to Excel
- Save as Text
- Print Report
- Print Preview
- Report Display Settings

Export as Microsoft® Excel file

Click the **Export to Excel** Button on the Statistical Report Toolbar to open the Export to Excel dialog.

Choose a folder to save the Excel file in and an appropriate file name and click **Save.**

Save as Text file

Click the **Save as Text** Button on the Statistical Report Toolbar to open the Export to Text dialog.

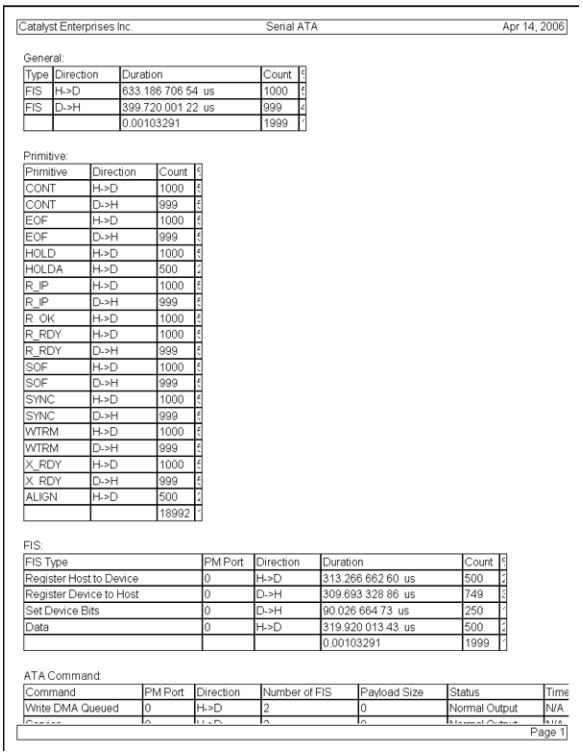
Choose a folder to save the Text file in and an appropriate file name and click **Save.**

Print Statistical Report

Click the **Print** Button on the Statistical Report Toolbar to open the select printer dialog. Choose an available printer and click **OK.**

Print Preview

Click the **Print Preview** Button on the Statistical Report Toolbar to display a preview of the report to be printed**.**



Catalyst Enterprises Inc. Serial ATA Apr 14, 2006

General:

Type	Direction	Duration	Count
FIS	H->D	633,186,706.54 us	1000
FIS	D->H	399,720,001.22 us	999
		0.00103291	1999

Primitive:

Primitive	Direction	Count
CONT	H->D	1000
CONT	D->H	999
EOF	H->D	1000
EOF	D->H	999
HOLD	H->D	1000
HOLDA	H->D	500
R_IP	H->D	1000
R_IP	D->H	999
R_OK	H->D	1000
R_RDY	H->D	1000
R_RDY	D->H	999
SOF	H->D	1000
SOF	D->H	999
SYNC	H->D	1000
SYNC	D->H	999
WTRM	H->D	1000
WTRM	D->H	999
X_RDY	H->D	1000
X_RDY	D->H	999
ALIGN	H->D	500
		18992

FIS:

FIS Type	PM Port	Direction	Duration	Count
Register Host to Device	0	H->D	313,266,662.60 us	500
Register Device to Host	0	D->H	309,693,328.86 us	749
Set Device Bits	0	D->H	90,026,664.73 us	250
Data	0	H->D	319,920,013.43 us	500
			0.00103291	1999

ATA Command:

Command	PM Port	Direction	Number of FIS	Payload Size	Status	Time
Write DMA Queued	0	H->D	2	0	Normal Output	N/A

Page 1

Figure 110 Sample Print Preview of Report

Report Display Settings

You may set up the report columns for display to suit a particular analysis need eliminating the need to show hide columns individually. Use the **Setting** dialog to configure the display for each page.

Click the **Setting** Button on the Statistical Report Toolbar to open the Setting dialog.

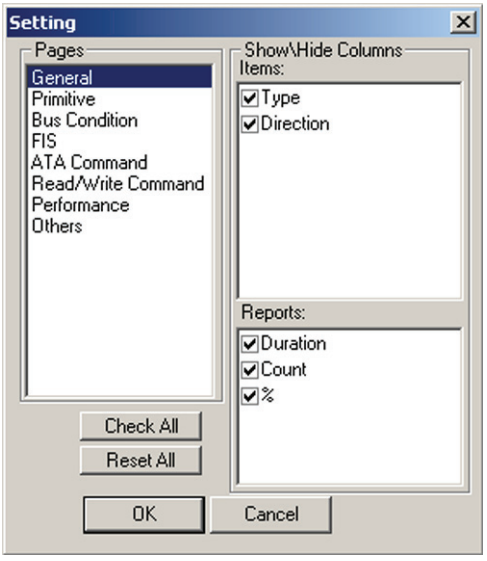


Figure 111 Statistical Report Column Setting

Link With Sample View

Whenever a Type is selected on any page of the statistical report, a set of navigation buttons is enabled allowing you to examine each instance of that type in the sample viewer.

Click the **Jump to Previous** Button on the Statistical Report Toolbar to go to the previous instance of the selected type in the Sample Viewer**.**

Click the **Jump to Next** Button on the Statistical Report Toolbar to go to the next instance of the selected type in the Sample Viewer**.**

Click the **Jump to Specific** Button on the Statistical Report Toolbar to go to the instance specified as N of M items on the Statistical Report Toolbar**.**

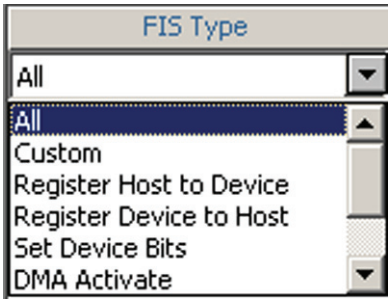
Formatting the Statistical Report View

Initially the Statistical Report View contains all of the information in columns, but you may customize the display for your needs by:

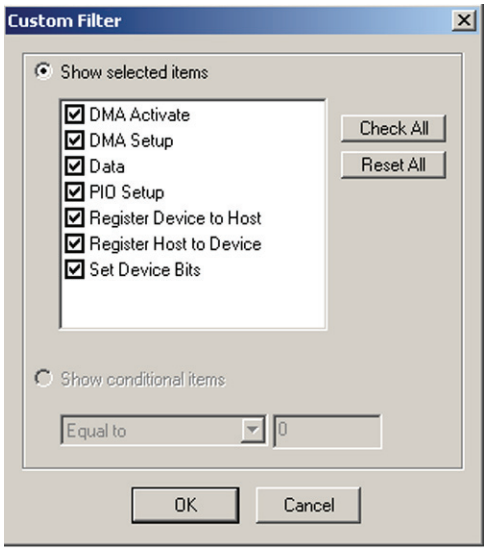
- Sorting items by column
- Filtering Columns by item
- Hiding any column on the display

Filtering Column Content

To filter column content click the down arrow in the heading for that column and choose the items that you wish to be displayed. The default is All. By checking a specific item you exclude everything but that item for display.



Choosing **Custom** allows you to specify more than one item for display.



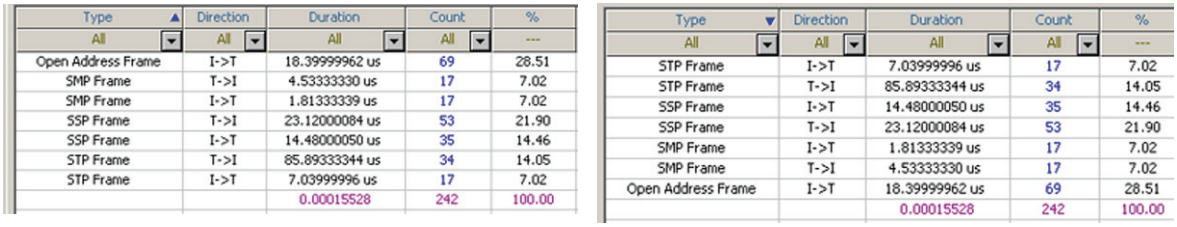
Check the items you wish to display and click **OK**.

Hiding Columns

To hide a column, right click in the column and choose Hide. To unhide a column, right click on any column and choose Unhide.

Sorting Column Content

To sort column content, click the heading for that column. Repeated clicking of the column heading will sort the column in ascending or descending order.



Type ▲	Direction	Duration	Count	%
All	All	All	All	---
Open Address Frame	I->T	18.39999962 us	69	28.51
SMP Frame	T->I	4.53333330 us	17	7.02
SMP Frame	I->T	1.81333339 us	17	7.02
SSP Frame	T->I	23.12000084 us	53	21.90
SSP Frame	I->T	14.48000050 us	35	14.46
STP Frame	T->I	85.89333344 us	34	14.05
STP Frame	I->T	7.03999996 us	17	7.02
		0.00015528	242	100.00

Type ▼	Direction	Duration	Count	%
All	All	All	All	---
STP Frame	I->T	7.03999996 us	17	7.02
STP Frame	T->I	85.89333344 us	34	14.05
SSP Frame	I->T	14.48000050 us	35	14.46
SSP Frame	T->I	23.12000084 us	53	21.90
SMP Frame	I->T	1.81333339 us	17	7.02
SMP Frame	T->I	4.53333330 us	17	7.02
Open Address Frame	I->T	18.39999962 us	69	28.51
		0.00015528	242	100.00

Figure 112 Toggling Type Sort Order

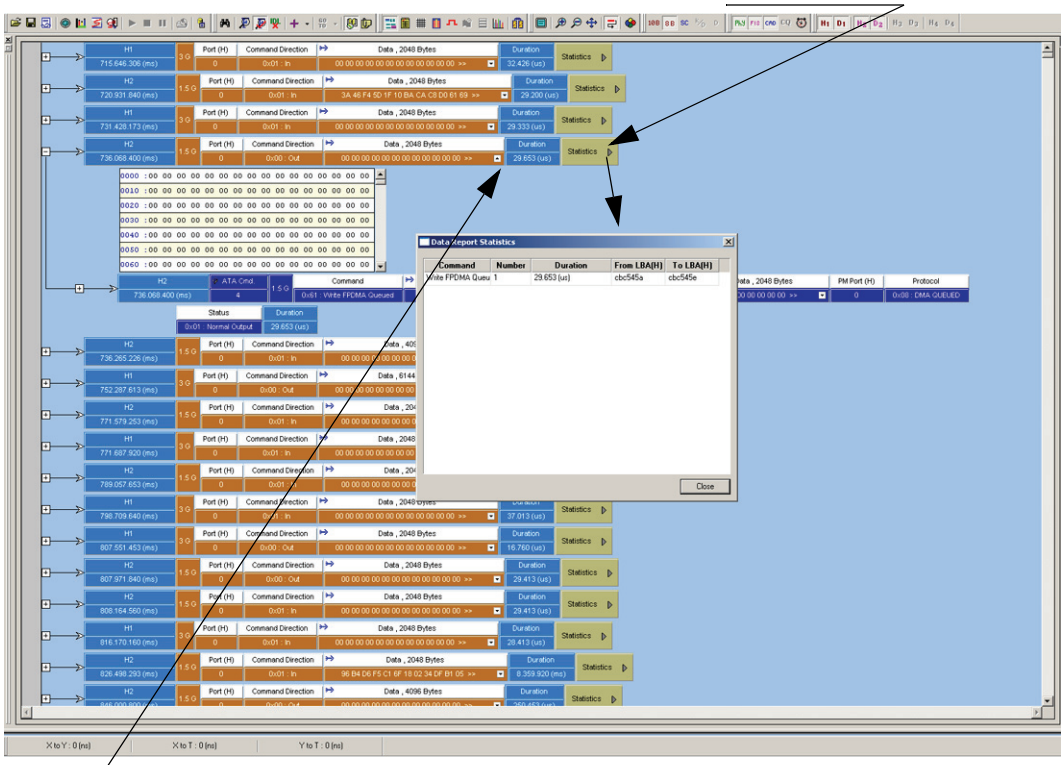
Data Report

Whenever a captured sample is displayed in the Sample Viewer, the Data Report Button is enabled on the Viewer toolbar and the Data Report selection is enabled in the Report menu.

The data report displays all of the data that is sent from the host to the device and from the device to the host. All PIO In =>In commands are grouped as a data packet until the occurrence of a PIO Out =>Out command creating a new data packet.

To display a **Data Report,** click the Data Report Button on the Viewer toolbar or choose Data Report from the Report menu.

Click the Statistics button to display data report statistics



Click the Down Arrow in a data field to display Data Report details

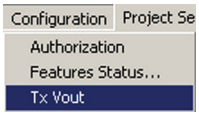
Figure 113 Data Report

Display Utilities

Tx Vout

Your analyzer incorporates the ability to select TX Vout for the transmitter on each port. Selecting TX Vout increases the output voltage swing above the nominal value for test and characterization purposes. This feature is also useful to compensate for line loss when driving long cables. The output range without TX Vout is 500 - 700 mv, nominally 600 mv (see Figure 115) and 800 - 1300 mv, nominally 1100 mv (see Figure 116) with TX Vout selected.

To select TX Vout click **Configuration** and then choose



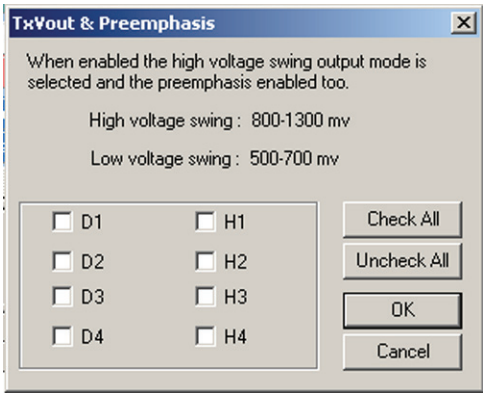


Figure 114 Choose Port for TX Vout

Check the Port(s) for which you wish to apply TX Vout and click **OK**.

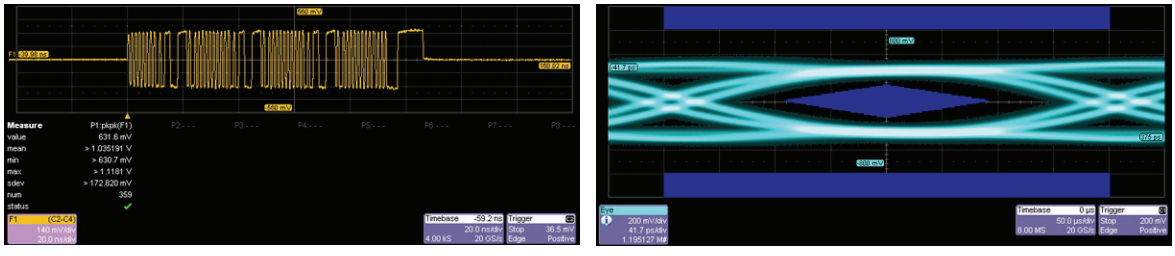


Figure 115 Waveform and Eye Diagram with no TX Vout

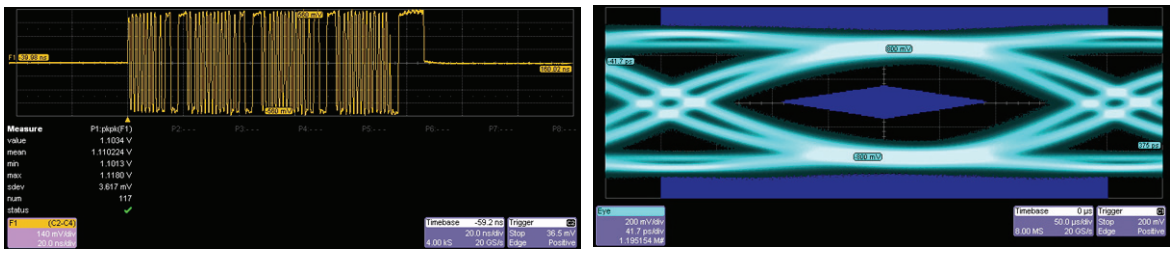
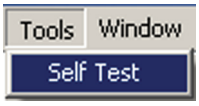


Figure 116 Waveform and Eye Diagram with Tx Vout

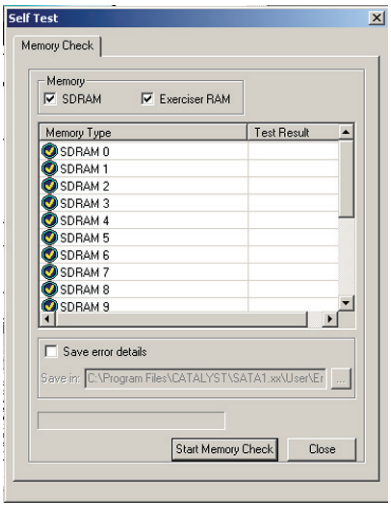
Memory Check

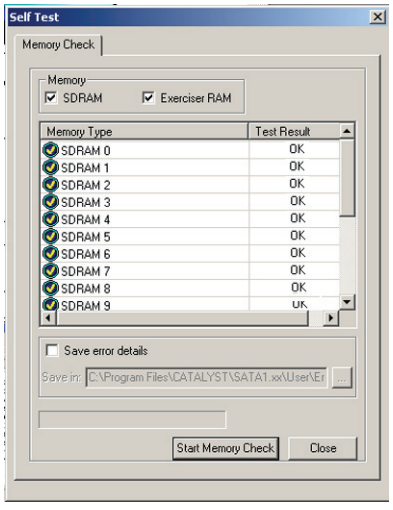
The STX230/STX430 has a built-in RAM self test utility.

Note: To be enabled in software version 2.x



Click Tools on the main menu bar and choose **Self Test** to open the Self Test dialog.



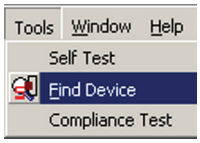


To perform a Memory check, choose the memory to be tested and click the **Start Memory Check** button. After a short time the Test result status will appear.

You may save the memory check result by checking the **Save error details** check box and specifying a destination file name.

Find Device

You may use this utility to obtain all of the vendor specific information and detailed device parameters.



Click Tools on the main menu bar and choose **Find Device** to open the Device Identifier dialog.

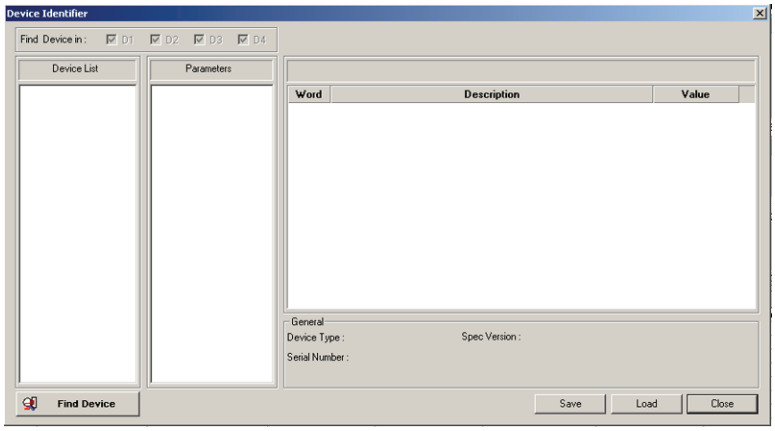


Figure 117 Device Identifier Dialog

Click the **Find Device** button to search for connected devices. After a brief period the dialog will display all of the device information.

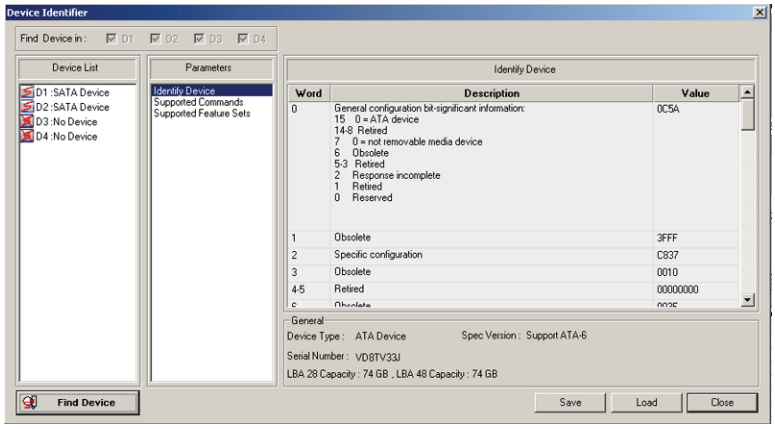


Figure 118 Identified Devices

Click on a device in the Device List to display information about that device.

Compliance Test

The Compliance Test consists of a set of selectable tests to verify compliance with the SATA specification.

To run a Compliance Test click **Tools** on the main toolbar and choose **Compliance Test**.

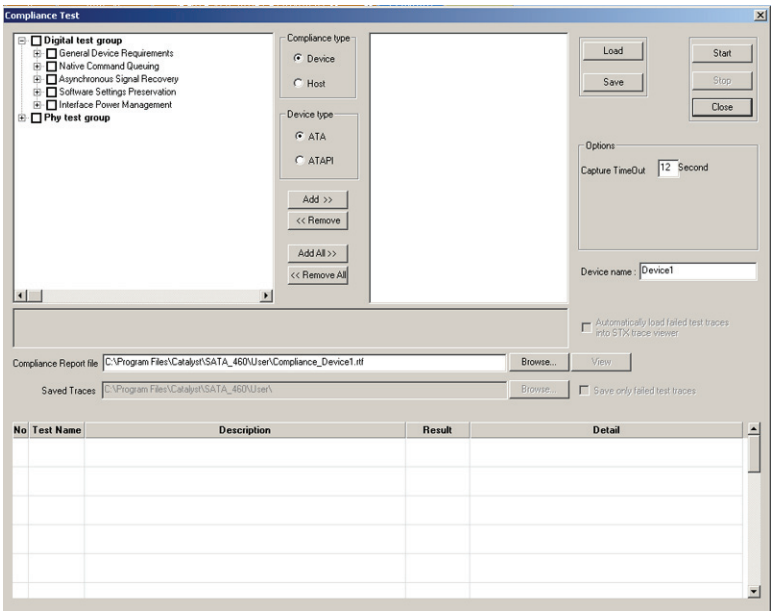


Figure 119 Compliance Test Selection

1. Choose the desired tests in the left pane of the dialog and click the **Add>>** button.
2. Enter a value for **Capture Timeout**.

Note: If a test running time exceeds the Capture Timeout value the test will be terminated.

3. To view failed test traces in sample viewer check, the **Automatically load failed test traces** check box.
4. To save the failed test traces only, check the **Save only failed test traces** check box.
5. When all of the desired tests have been selected click **Start**.
6. The test will run and after a brief period of time display the result.

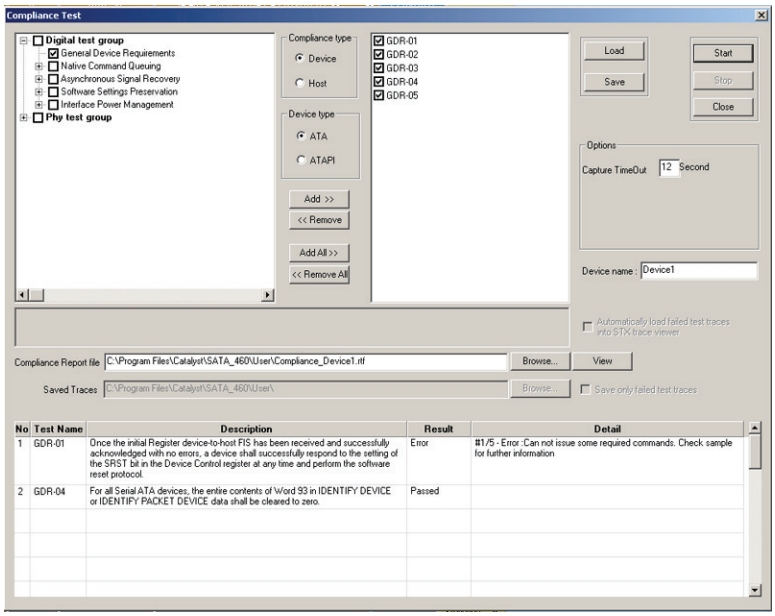


Figure 120 Compliance Test Result

7. You may save the current compliance setup for later use by clicking the **Save** button to open the Save As dialog. Assign a meaningful name to the setup and save it as an *cst compliance file.
8. You may also run a previously defined setup by clicking the **Load** button and choosing a previously defined setup to run.

Configuration Features Status

To get a comprehensive overview of the current configuration feature status, click **Configuration** and choose **Features status.**

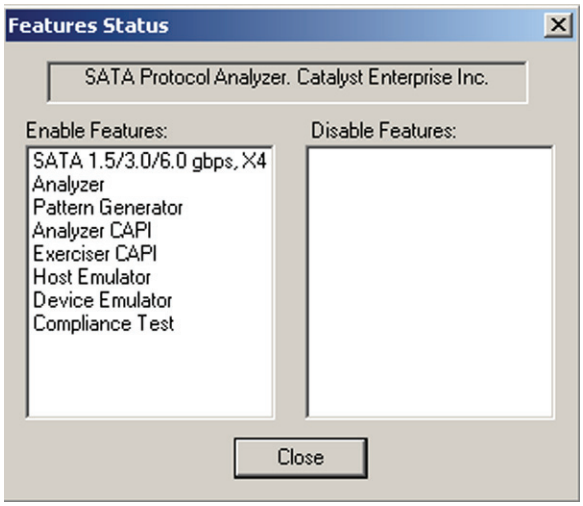


Figure 121 Configuration Feature Status

Appendix A

Creating a Pattern Generator File

You may use any text editor or word processor to create a pattern generator file (*.spg) using the following conventions:

Key words

ALIGN, CONT, DMAT, EOF, HOLD, HOLDA, PMACK, PMNAK, PMREQ_P, PMREQ_S, R_ERR, R_IP, R_OK, R_RDY, SOF, SYNC, WTRM, X_RDY, XXXX, LOOP, Enable, Disable, Host, Device, Scramble, Role, END_OF_FILE.

Comment format

/*Comment text*/

Primitive definition format

To add an ALIGN primitive, use ALIGN or 27.3 10.2 10.2 K28.5
To add a CONT primitive, use CONT or 25.4 25.4 10.5 K28.3

Loop definition format

You may write a defined pattern into memory repeatedly by enabling a loop.
Loop definition allows either “Enable” or Disable”. To enable looping use: Loop=Enable

Scramble definition format

Scramble definition allows either “Enable” or Disable”. To enable scramble use: Scramble=Enable

Role definition format

To specify SATA hardware role: Role=Host or Role=Device

END_OF_FILE definition

A pattern generator file must include END_OF_FILE as the last statement in the file.

Figure 122 illustrates a typical Pattern Generator file.

```
/*.....Set Device Bits.....*/
/*Device*/
23.2 23.2.21.5 K28.3              /*X_RDY*/
23.2 23.2.21.5 K28.3              /*X_RDY*/
25.4 25.4.10.5 K28.3              /*CONT*/
XXXX
XXXX
XXXX
XXXX
XXXX
XXXX
XXXX
23.1 23.1 21.5 K28.3              /*SOF*/
00 50 40 A1
E0 00 00 00
21.6 21.6 21.5 K28.3              /*EOF*/
24.2 24.2 21.5 K28.3              /*WTRM*/
24.2 24.2 21.5 K28.3              /*WTRM*/
25.4 25.4 10.5 K28.3              /*CONT*/
XXXX
XXXX
XXXX
XXXX
21.5 21.5 21.4 K28.3              /*SYNC*/
21.5 21.5 21.4 K28.3              /*SYNC*/
25.4 25.4 10.5 K28.3              /*CONT*/
XXXX
XXXX
XXXX
XXXX
Role=Device
Loop=Enable
Scramble=Disable
END_OF_FILE
/*.....*/
```

Figure 122 Sample Pattern Generator File *spg

Appendix B

WAN Operation

WAN connected operation is supported. Contact factory for details of operation.

Index

LeCroy Corporation
3385 Scott Blvd.
Santa Clara, CA 95054
Phone 1-800-909-7112 or 408-635-7085
Fax 408-727-6622
www.lecroy.com