# SDLA Visualizer Serial Data Link Analysis Visualizer Software Printable Application Help





SDLA Visualizer Serial Data Link Analysis Visualizer Software Printable Application Help

Supports SDLA Visualizer Firmware V2.0.1 and above



Copyright <sup>©</sup> Tektronix. All rights reserved. Licensed software products are owned by Tektronix or its subsidiaries or suppliers, and are protected by national copyright laws and international treaty provisions. Tektronix products are covered by U.S. and foreign patents, issued and pending. Information in this publication supersedes that in all previously published material. Specifications and price change privileges reserved.

TEKTRONIX and TEK are registered trademarks of Tektronix, Inc.

# **Contacting Tektronix**

Tektronix, Inc. 14150 SW Karl Braun Drive P.O. Box 500 Beaverton, OR 97077 USA

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

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

# **Table of Contents**

<b>W</b> 7-1	-
welcome	 V

# **Getting started**

Software updates from the Tektronix web site	1
Requirements and installation	1
Conventions	2
Application file types and locations	2
Moving between applications	3
Online help	4

# **Product overview**

SDLA visualizer product overview	5
Understanding the system	7
Understanding test points	11
Using DPOJET and SDLA visualizer together	16

# **Components and menus**

Main menu in detail	9
Test points	2
Test point and bandwidth manager 2	2
Saving test points	7
Exporting filters for use with a 32-bit oscilloscope	0
Save test point filters for multiple sample rates	1
Creating a custom bandwidth limit filter	2
De-embed block	4
De-embed block overview	4
De-embed-Embed menu 3.	5
How to re-normalize S-Parameters to different reference impedances	0
Configuring probes	2
Probe and tip selection 4	7
Block configuration menu	1
Load configuration menu	7
Plots	0
Plots	0
Using plots for troubleshooting s-parameters	6

Tx block (Transmitter modeling block) 69
Tx block overview
Tx configuration menu
Tx emphasis menu
Embed block
Embed block overview
Rx block (Receiver modeling block) 7'
Rx block overview
Rx configuration menu 79
Using CTLE to improve signal recovery
Using the PCIE3 option in CTLE
Using clock recovery for FFE-DFE equalization
Adjusting FFE-DFE to improve signal recovery
Using the PCIE3 option in FFE-DFE
Using the taps tab
Running the RX equalizer 92
AMI mode
Configure actions for the apply and analyze buttons
Creating filters for a sampling scope

# Running a test

Running a test: recommended order		101
-----------------------------------	--	-----

# **Examples and troubleshooting**

Examples of tasks and troubleshooting	109
Example of de-embedding cables	110
Example of embedding a serial data link channel	115
Example of de-embedding a high impedance probe	118
Example of de-embedding significant reflections with dual input waveforms	121
Example of removing a DDR reflection with a single input waveform	137

# **GPIB remote control**

Using GPIB remote control	143
GPIB commands	144
APPLICATION: ACTIVATE "Serial Data Link Analysis"	145
VARIABLE:VALUE "sdla", "p:exit"	145
VARIABLE: VALUE? "sdla"	145
VARIABLE:VALUE "sdla", "p:adapttaps: <value>"</value>	146

VARIABLE:VALUE "sdla", "p:bitrate: <value>"</value>	146
VARIABLE:VALUE "sdla", "p:ctletype: <value>"</value>	147
VARIABLE:VALUE "sdla", "p:dfestate: <value>"</value>	147
VARIABLE:VALUE "sdla", "p:RunEQ"	147
VARIABLE:VALUE "sdla", "p:source: <source/> "	148
VARIABLE:VALUE "sdla", "p:sourcetype"	148
VARIABLE:VALUE "sdla", "p:recall: <path and="" file="" name="">"</path>	148
VARIABLE:VALUE "sdla", "p:source2: <source2>"</source2>	149
VARIABLE:VALUE "sdla", "p:analyze"	149
VARIABLE:VALUE "sdla", "p:apply"	149

# Welcome



Figure 1: The Tektronix SDLA Visualizer offers a powerful, flexible set of modeling tools for de-embedding, embedding and equalizing high speed serial signals. Using a simple user interface with many configureable features, you can model a measurement circuit to de-embed the effects of scopes, probes, fixtures, cables and other equipment from the acquired scope waveform back to the transmitter block. Likewise, you can model and embed a simulation circuit from the transmitter block that simulates possible effects upon the signal. Both single and dual waveform input modes are available.

# **Getting started**

# Software updates from the Tektronix web site

Periodic software upgrades may be available from the Tektronix Web site.

To check for upgrades:

- 1. Go to the Tektronix Web site (*www.tektronix.com*).
- 2. Press on Support and select the item Downloads, Manuals & Documentation.
- 3. Enter "SDLA" in the MODEL OR KEYWORD text box.
- 4. Select Software in the SELECT DOWNLOAD TYPE drop-down list.
- 5. Press Go to find the available software upgrades.
- **6.** Press the appropriate software title. Read the application information to be sure that it is compatible with your instrument model.
- 7. Press Login to access this content and log in to access the download.
- 8. Press the Download File link.

# **Requirements and installation**

The SDLA Visualizer application is installed on Tektronix DPO/DSA/ MSO70000/C/D/DX Series oscilloscopes before they leave the factory. The installation provides ten free uses of the full featured SDLA Visualizer application.

#### **Requirements for Proper Operation**

The SDLA Visualizer application requires a Tektronix DPO/DSA/MSO70000/C/D/DX Series Oscilloscope with a single shot bandwidth  $\geq$ 4.0 GHz.

It also requires Tektronix DPOJET Jitter and Eye-diagram Analysis software in order to perform jitter and timing analysis.

To ensure accurate acquisitions, be sure to properly calibrate your oscilloscope by running the signal path compensation. The length of time between SPC and temperature changes at the instrument location dictate when this should be done.

#### Software Compatibility

Refer to the product Release Notes or the Optional Applications Software Installation manual for the compatible versions of oscilloscope software and for DPOJET.

#### **Option Key Requirement**

You must have a valid option key for the application. Without the key, there are ten free trials. Consult with your Tektronix Applications Engineer or Account Manager for details.

#### Reinstalling the SDLA Visualizer Software

To install the latest version of SDLA Visualizer software, press *Software Updates From the Tektronix Web Site*.

# Conventions

The online help uses the following conventions:

- DUT refers to the Device Under Test.
- When a step requires a sequence of selections, the > delimiter indicates the path from menus to sub-menus and to menu options.
- The directory path to support files is C:\Users\Public\TekApplications \SDLA.

# Application file types and locations

The software uses the following file types and locations. The support files are arranged in folders with descriptive names at C:\Users\Public\Tektronix \TekApplications\SDLA:

- Example waveforms Example waveform files to help you learn the application.
- Input filters FIR and IIR filter files
- Input S-parameters Touchstone 1.0 version
- Output filters where the software stores generated FIR filters when the Apply button is pressed. The filenames are overwritten each time you click the Apply button. You can rename the filter files to save a set of FIR filters for later use.

These filters are stored in the directory entitled C:/users/public/ Tektronix/TekApplications/SDLA/output filters.

#### **Default naming conventions:**

For Single Input mode, the filenames are:

Sdlatp1.flt, sdlatp2.flt, .... Sdlatp<n>.flt where n is the test point number.

For Dual Input mode: folders named

Tp1, Tp2, ... Tp<n>

are created, where n is the test point number. Inside each folder is the set of files.

Save recall – temporary location where software stores the SDLA Visualizer setup configuration files.

Your custom S-parameter files and filter files can reside at any path accessible to the instrument.

# Moving between applications

The quickest way to move between software applications is to hold down the keyboard Alt key and tap the Tab key to pick an application.



An alternative is to use the triangle buttons on the right side of the Main Menu to switch between the SDLA Visualizer, TEKScope and DPOJET applications:

- Press the left triangle to bring the oscilloscope waveform display to the foreground.
- Press the right triangle to bring the oscilloscope waveform display into view with SDLA Visualizer application still in the foreground. This option is handy when also using the DPOJET application.

You may bring all the SDLA Visualizer windows to the foreground by first pressing the minimize button at the upper right corner of the oscilloscope window to collapse it to the Windows tool bar. Then press the right triangle on SDLA to expand the scope back to full screen with SDLA in the foreground.



# **Online help**

# Help in Different Languages

If you would like to download a .PDF file of the Online Help that has been translated into Japanese, simplified Chinese, or Korean, visit www.tektronix.com and press on "Change Country" at the top. Then enter the search term "SDLA Visualizer".

Press the **Help** button in the upper right corner of the SDLA Visualizer Main Menu to bring up the online Help system. Pressing the **F1** key at any time also brings up the Online Help system.



# **Product overview**

# SDLA visualizer product overview



The Tektronix SDLA Visualizer offers a powerful, flexible set of modeling tools for de-embedding, embedding and equalizing high speed serial signals. Using a simple user interface with many configureable features, you can model a measurement circuit to de-embed the effects of scopes, probes, fixtures, cables and other equipment from the acquired scope waveform back to the transmitter block. Likewise, you can model and embed a simulation circuit from the transmitter block that simulates possible effects upon the signal. Both single and dual waveform input modes are available.

SDLA Visualizer offers full 4-Port S-parameter modeling support that takes into account the Tx and Rx impedance models, along with all transmission line characteristics. The signal path is fully represented by a unique cascading S-parameter feature; if any parameter changes anywhere in the cascade, it affects all test points in the cascade.

Many standards require that equalization is applied to the signal before measurements are taken. SDLA Visualizer provides CTLE, FFE and DFE equalization modeling tools with support for serial standards such as PCI Express 3.0, USB 3.0 and SAS. Also available is an IBIS-AMI model that lets you use equalization files supplied by a chip vendor.

Validation is simplified with a rich set of plotting tools, including S-parameter plots, time domain plots, Smith chart, and overlay tools. These plots are available starting with the cascade block configuration stage, providing confidence that the input models (i.e. S-parameters) are correct.

After the circuits are defined, SDLA Visualizer provides the ability to observe the signal via 12 user-defined test points, including 4 that are movable within the De-embed and Embed Blocks. You may view multiple test points simultaneously, and observe areas of the signal that you could not probe otherwise. Up to four math and two reference waveforms are visible on the scope graticule at one time. You are able to see the differential, common mode, or individual inputs of the signal at once, without having to create multiple models for each option. You can also create test point filter (transfer function) plots that allow for verification of the system setup. Magnitude, Phase, Impulse and Step plots are available.

SDLA is intended to be used along with Tektronix DPOJET Real-time Jitter and Timing Analysis software. Together, these tools provide deep insight and analysis capabilities so that you can visualize an entire signal processing path and accurately measure the true signal from the DUT.

Some tasks you can accomplish using SDLA Visualizer

- Remove the effects of reflections, cross-coupling, and loss caused by nonideal probe points, fixtures and cables
- Remove the effects of interposers using 3, 4, or 6-port S-parameter models
- Simulate and measure at test points using actual captured waveforms where physical probing is not practical
- Observe the signal at the end of the link by embedding user-defined channel models into the waveform at the transmitter
- Add or remove transmitter equalization, using 2 or 3-tap filter coefficients or FIR filter
- Open closed eyes using CTLE, clock recovery, DFE and FFE equalization
- Model silicon-specific receiver equalization algorithms using IBIS-AMI models, so you can virtually view the signal inside of the receiver
- De-embed high impedance or SMA probes
- Model RLC, TDT waveforms, and lossless transmission lines in the absence of S-parameters
- Create S-parameter plots, time domain plots, and Smith Chart plots for quick verification of S-parameters and test point transfer functions
- Perform quick analysis of jitter and timing parameters using integrated DPOJET support
- Work with DDR and next generation serial standards including PCI Express
   3.0, USB 3.0, SAS 6G, SATA, and DisplayPort (including interposer model)

#### For more information:

Understanding the System Using DPOJET and SDLA Visualizer Together Running a Test: Recommended Order **NOTE.** *Pressing the key at any time brings up the Online Help system.* 

**NOTE.** If you would like to download a .PDF file of the Online Help that has been translated into Japanese, simplified Chinese, or Korean, visit www.tektronix.com and press on "Change Country" at the top. Then enter the search term "SDLA Visualizer".

# SEE ALSO:

- Main Menu in Detail
- Examples of Tasks and Troubleshooting

# Understanding the system



SDLA Visualizer requires you to define two circuit models, the Measurement Circuit and the Simulation Circuit, that both connect to the **Tx Block**. The Tx Block makes use of Thevenin equivalent voltage to provide a point where the acquired waveform is passed into the simulation side of the system. (Thevenin's Theorem states that it is possible to simplify any linear circuit, no matter how complex, to an equivalent circuit with just a single voltage source and impedance.)

# The Measurement Circuit

The upper part of the Main Menu diagram stemming from the Tx Block represents the Measurement Circuit: the probes, scope, fixtures and the portion of the channel between the Tx and the fixture. (Note that this diagram changes, reflecting whether Single or Dual Input mode is specified.) This is where the S-parameter models that represent the physical test and measurement system used to acquire the signal need to be defined and loaded into the **De-embed Block**. In the absence of S-parameters, you can use RLC or lossless transmission line models.

The **test points** in this circuit represent simulated probing locations that allow visibility of the link at multiple test locations, including two movable test points within the De-embed Block. The software derives the transfer function(s) and creates FIR filters for each test point. When the filters are applied to the waveform(s) acquired from the scope, SDLA produces waveforms at the desired test points. The waveform with the loading of the Measurement Circuit can be viewed at **Tp1**, **Tp6**, or **Tp7**.

Tp6 and Tp7 always mean the signals at the right side of the block at the left of the test point. When s6p or s3p interposer is considered, to get the signals at the right side of the interposer, set the block at the right of the interposer to Thru, and Tp6 or Tp7 can be put to the right of the block. For example, if the HiZ probe is located between B2 and B3, to get the signal at the left of the interposer, then set Tp6 between B2 and B3. To get the signal at the right side of the interposer, set B3 to Thru, and Set Tp6 between B3 and B4.

#### The Simulation Circuit

The lower part of the Main Menu diagram stemming from the Tx Block represents the Simulation Circuit. Now that the waveforms have been deembedded back to the Tx Block, the Simulation Circuit is used to embed a simulated channel to the Tx Block. The S-parameter models for the link you would like to simulate need to be defined and entered into the **Embed Block**. Again, you may use an RLC or a lossless transmission line models when S-parameters are not available. The load of the receiver is also modeled in the Embed Block. The **Rx Block** allows you to specify Rx equalization. The **test points** in this circuit allow visibility in between link components, including two movable test points within the Embed Block. **Tp2** shows the Tx output waveform without the loading of the Measurement Circuit, but with the loading of Simulated Circuit.

**NOTE.** The arrows on the Main Menu circuit diagram show the order in which SDLA processes the transfer functions. For the Measurement Circuit part of the diagram, the ACTUAL signal flow is in the . For the Simulation Circuit, the actual signal flow direction is the same as the signal processing flow arrows.

# Using the Embed Block to Close the Eye and Rx Block to Open the Eye

The Embed Block lets you "insert" a simulated channel so that you can observe the closed eye (viewable at **Tp3**):



Now, you can use the Rx block to open the eye and observe the signal after CTLE (**Tp10**) or after FFE/DFE (**Tp4**) as been applied. The **Rx Block** allows you to specify Rx equalization. Serial data receivers typically contain three kinds of equalizers: a continuous-time linear equalizer (CTLE), a feed-forward equalizer (FFE), and decision feedback equalizer (DFE)). CTLE, clock recovery, DFE and FFE equalizers are available in the Rx Block; alternatively, IBIS-AMI models can be used to model silicon specific equalization algorithms. Also, three test points are available in the **Rx Block**. These allow for visibility of the waveform after CTLE and/or after FFE/DFE and recovered clock, or an IBIS-AMI model has been applied.

# **Test Points**

With 12 test points, SDLA Visualizer gives you visibility over multiple test points simultaneously, providing virtual "observation points" of the signal that you could not probe otherwise. You can view the transmitter signal with the loading of the measurement circuit at **Tp1**, and at the same time, view the deembedded measurement circuit at **Tp2** with an ideal 50 Ohm load. You have many flexible options for labeling test points, and for mapping test points to math waveforms. It is easy to put the test point labels onto the scope waveform display, so you can tell which waveform is which, and easy to apply the data to DPOJET, so that you know which waveforms in time with respect to each other. (By default, the delay is removed from the test point filters, so that events are close to being time-aligned.)

SDLA Visualizer provides up to 6 waveforms (four math and two reference) that are simultaneously visible on the scope graticule at one time, allowing visibility of the link at different locations. (You use the Test Point and Bandwidth Manager to map the SDLA test points to the math and reference waveforms.) The software allows for dynamic configuration of test points in order to best utilize the scope math channels (i.e. after de-embedding, CTLE, etc.) Also, four test points can be moved on the De-embed and Embed Menu cascade diagrams, providing maximum flexibility. Press here for a *deeper understanding of how test points work*.

Once the simulation and measurement circuits have been defined, you can easily save test point filters that can be used with the scope math system. For details, see *Saving Test Points*.

# **Modeling Block View**

Another way to view the system is as a series of modeling blocks for deembedding the effects of the waveform acquisition hardware setup, and modeling blocks for embedding link components that are not represented physically.

These diagrams illustrate the entire S-parameter processing path.

**Single Input Mode:** 







#### **Dual and Single Input Modes**

In some cases, it is desired to process each leg of the signal individually through the network, in order to completely take into account differences in the two sides of the signal. SDLA Visualizer offers a choice of Dual Input or Single Input modes on the Main Menu. In Single Input mode, the differential signal may be viewed at each test point. Dual Input mode allows the viewing of individual inputs, differential, or common mode. For additional information, see *Full 4–port Modeling*.

# Algorithms, theory and math derivations

For in-depth information on several advanced SDLA topics, including algorithms, theory, math derivations for re-normalizing S-parameters and converting single-mode S-parameters to mixed mode, see technical papers located at www.tek.com/sdla.

# SEE ALSO:

- Using DPOJET and SDLA Visualizer Together
- Product Overview

# **Understanding test points**

Test points output waveforms that represent the signal at a particular position in the system circuit diagram. Each test point waveform is obtained by applying at least one filter to the input waveform(s) acquired by the oscilloscope.

SDLA Visualizer provides up to 12 test points (when using the REF waveforms). Up to 6 test point outputs are viewable on the scope graticule at one time: four math and two reference. The SDLA processing and analysis operate only on waveforms that have been turned on and are displayed on the oscilloscope.



Press here for a Table of Test Point Descriptions.

Test point	Position	Description			
Tp1	Main	Measurement circuit loading the Tx block output			
Tp2	Main	Simulation circuit loading the Tx block output, measurement circuit de-embedded			
Тр3	Main	Rx block input. Simulation circuit loading the Tx block output, measurement circuit de- embedded			

Test point	Position	Description			
Tp4	Rx Eq Data	Data output of the Rx block after equalization			
Тр5	Rx Eq Clock	Test point for the recovered clock output of the Rx block			
Трб	De-embed Block	Movable test point with the measurement circuit loading the Tx block output			
Тр7	De-embed Block	Movable test point with the measurement circuit loading the Tx block output			
Тр8	Embed Block	Movable test point with the simulation circuit loading the Tx block output, measurement circuit de-embedded			
Тр9	Embed Block	Movable test point with the simulation circuit loading the Tx block output, measurement circuit de-embedded			
Тр10	CTLE	CTLE output			
Tp11	Тх	Thevenin equivalent voltage of the Transmitter model			
Tp12	Tx Test point Tx Empha				

# Test Point and Bandwidth Manager

Pressing on a test point on the Main Menu brings up the **Test Point and Bandwidth Manager**, which is used to configure test points and modes (Dual Mode only) and to save test point filters. For details, see *Test Point and Bandwidth Manager*.

🛃 SDLA Visualizer - Test Po	int and Bandwic	dth Confi	guration							
Tp1 - Map the desired t	est points to Ma	ath to see	e the processed w	aveform						
Tp On/Off	Map Tp to	o Math			Label	Filters	Global BW Limit	Delay		
◯ Math1	None	•			NA	Save	🔘 None	🔘 Keep Delay		
🔘 Math2	Tp1	•			Tp1	Save	Auto	Remove Delay		
🔘 Math3	Tp2	-			Tp2	Save	Custom	Adjust Delay		
Math4	ТрЗ	-			ТрЗ	Save				
Ref3	Tp5						BW: 25GHz			
O Ref4	Tp4		Scaling Factor					-		ок
			☑ 80 %	?		Export	filters for 32-bit scope	?	۶L	

# How Test Point Filters are Applied

The test point filters are derived from the S-parameter models that are contained in the De-embed, Tx, and Embed Blocks. These filters are of type FIR, which are convolved in the time domain with the source waveforms acquired on the oscilloscope. Press here for details on what generally happens when test point filters are applied.

- 1. First, you have to enter the S-parameters or models that will determine Sparameters for each of the blocks and terminations throughout the system using the Tx Block and De-embed/Embed Menu.
- 2. You also need to turn on and define the desired test points by pressing on a test point on the Main Menu and using the Test Point and Bandwidth Manager.
- 3. Finally, you press the Apply button in the SDLA Visualizer Main Menu. The software computes the filters (transfer functions) for each test point that has been turned on using the Test Point and Bandwidth Manager. These filters are then stored in the directory entitled C:/users/public/Tektronix/ TekApplications/SDLA/output filters. (You may also save the filters from the Test Point and Bandwidth Manager into files using your own names or folder.) Default Naming Conventions

For Single Input mode, the filenames are:

Sdlatp1.flt, sdlatp2.flt, .... Sdlatp<n>.flt where n is the test point number.

For Dual Input mode: folders named

Tp1, Tp2, ... Tp<n>

are created, where n is the test point number. Inside each folder is the set of files.

At the same time, SDLA loads the filters that have been turned on into the oscilloscope math menu, and creates a math expression that will display live waveforms for the selected test points on the oscilloscope graticule.

# **Crosstalk and Reflection Handling**

SDLA Visualizer uses all elements of the S-parameter models to compute the transfer functions for test points. Press here for an illustration of the signal flow graph.

Shown below is an example of the signal flow graph for three cascaded 4-port networks. This illustrates the effects that cross-talk paths, transmission paths, and reflection paths have on the overall transfer function from one point in the network to another point in the network. SDLA Visualizer uses all of these S-parameter paths to compute the transfer functions for test points.



### **Full 4-port Modeling**

This system maintains full 4-port modeling. Therefore, the test points are differential, and each contains a set of four possible waveforms (test point modes) to view.

Dual input mode:

Dual input selection takes two waveforms from two channels, math functions or reference waveforms in the oscilloscope, and processes them through the 4-port system to obtain test point waveforms. When Dual Input mode has been selected on the Main Menu, the Test Point and Bandwidth Manager will show the options for **Select Test Point Mode**. The options are:

- A: the waveform on the upper line of the test point
- **B**: the waveform on the lower line
- A B: the differential waveform and
- (A + B)/2: the common mode waveform.

In Dual Input mode, **each test point can output waveforms for all four modes described above**. Each of these four modes requires two filters applied to the two input waveforms. Press here for example math expressions that SDLA might set up in the oscilloscope Math menu.

An example math expression that SDLA might set up in the oscilloscope Math menu:

#### Math1 = arbflt1(ch1) + arbflt2(ch2)

Mathematically, only four filters are required for a differential test point. However, this would require two filters each for A and B modes, and all four filters for differential and common modes. In order to simplify to only two filters for any mode, an additional four filters are created from linear combinations of the four basic filters. Thus SDLA creates eight filters for each test point, as shown below:



# Single Input mode:

When Single Input mode is selected on the Main Menu, an assumption is made that a differential input waveform of form A - B is acquired on a single source of the oscilloscope (Src1). SDLA then splits this waveform mathematically into an exactly balanced A and B signal, which is then processed through the 4-port cascaded system.

For Single Input operation, the test points throughout the system only utilize the A - B Mode (differential waveform as output). Only one filter is required, and is applied to the input source waveform to obtain the output test point waveform. An example math expression that SDLA might set up in the Math oscilloscope menu:

Math1 = arbflt1(ch1)

# **SEE ALSO:**

- Test Point and Bandwidth Manager
- Saving Test Points
- Main Menu in Detail
- Product Overview.

# Using DPOJET and SDLA visualizer together

Together, SDLA Visualizer and DPOJET provide a complete solution for highspeed serial measurement and analysis. DPOJET operation is integrated right into the SDLA Visualizer Main Menu **Analyze** and **Config** buttons. DPOJET gives you the flexibility to analyze and compare the results at multiple points on the link. What's more, it allows multiple measurement configurations; for example, you could easily compare standard-specific vs. silicon-specific clock recovery measurement parameters.

The figure below shows an example whereby the Analyze button has been configured to automatically run DPOJET without changing the SDLA setup. Here, the PCI Express 3.0 configuration has been defined by the user. Notice how using DPOJET and SDLA Visualizer together gives you full link visibility of the eye diagram and associated measurements for each of the desired test points. The eye diagram on the top left shows the acquired waveform and the input into SDLA. The eye diagram on the top right shows the Simulation Circuit loading the Tx block output (**Tp3**). The eye diagrams on the bottom show the signal after CTLE (**Tp10**) and after FFE/DFE (**Tp4**).

te ter vers	oni Harizikog Trig Disple	e Carsons			ope Aneyze	CORPORT LINES			1.66		3
BROJET PM	6									20	8
20 8 8	电行电电 体田;	0.00.00.0	14 2 1	민동민		ାଷଷଣ					
VWalker	n Reight)	Eye Disgram	100 A	1 Terre	Y Vue	-07	Bright L fa	e Diegram		of Terror	
										A DESCRIPTION OF THE OWNER OF THE	0
\$20as//		-	-	-						-	
								-			
ev					. ev			1.11		100	
					102	ALC: NO IS					
ALL	LOOK OW		A DESCRIPTION OF		- SALEY DOL	C. R. HOMENING	-			_	
Calcon a		L. C. C.	1000		Seate	and a state of the					
V Notice	And the second	- Date Distances		a Tarrer			Mana Sec. 1			1 Tana	
								and the other Designation of the local division of the local divis			0
				•	200-01						
200mily					and the second se	Statement of the local division of the local					
20544											
20144 V CV			Concession in which the		ov 🗮						
						Dice-					
	04a 644000-5100 2 31 99503, Tahu (0000 3499035					1000 211 7521 Televit	000.017904				
	04a 200000-100 2 31 99333, Tatus (10000-31/9833 24, AC-D 1/194 ave)					INCOME AND TRANSPORT	000:317931 at				
or Contraction Con	den Statistics Statistics Statistics AC 2004 million AC 2004 million							fa Zipa			
	Aller Aller Aller Eye Diagram Analysis	Tools				NO.217501, Table 2 PCE ACC TABLE	a a a	De Ster			
Sitter and	Aller Aller Aller Constant Aller Constant Eye Diagram Analysis Operall Text Result	Tools	2001 40					Cyfors J			
Jitter and	Ca Statute of the mode several and a statute office statute File Statute Eye Diagram Analysis Overall Test Result	Tools @Pass	ALS AN			Contrast Transit Total and Total a	an la san an a	te Xee Options			
Jitter and	Caracteria Second Second Secon	Tools Pass Passfal	Mean	Stol Dev	or United	The particular of the second s	w Summary P-P	te - 20e Options • Expan Population			
Jitter and Select	Cramero Contractor Allow Contractor Allow Contractor Allow Contractor Allow Contractor Eye Diagram Analysis Overall Test Result S Description • OS LIVAL Anapolit	Tools Pass Passfall Pass	Alexan Monari Manari Mariana	Stat Dev	or Carlor Access Access	Vie Min Min Min	w Summary P-P access	ta ⊃tea Options ▼ (Espan Population			V A (s)
Jitter and Select	Criminal Annu Criminal Crimina	Tools Pass Passfal Passfal Pass Pass	Meser 158.13ev/	Stal Dev B. DOORV	An and a second	View Min Min 153, 134, 134, 134, 134, 134, 134, 134, 13	w Summary P-P 1-0005 0.00074	Cyclons Cyclons V Coano Population			VA(s)
Jitter and Select Configure	Constant of the second of the	Tools PassFal PassFal Pass Pass Pass Pass Pass	Mouri 152,13mV 152,13mV 152,14mV	501 Dev 0.0000V 0.0000V 0.0252m/	Annual States	Vie Min 158, 138, 138, 1 158, 1 1	W Summary P-P Construction 0.0000V 285.000V	Container Options Population 1 1 1		Char Char Char Char Char Char Char Char	VA(s)
Jitter and Select Configure Besuits	Constant of the second of the	Tools PassFal PassFal Pass Pass Pass Pass Pass Pass Pass	Mouri 158.13mV 158.46mV 255.62mV	501 Dev 0.0000V 03.257mV 02.57mV	New Arrest States of States of State	Vie Min 158. Servi 158. Servi 158. Servi 158. Servi 152. Servi 152. Servi 152. Servi 152. Servi 152. Servi	W Summary 9-9 100007 0.00007 0.00007 285.00m/ 543.00m/	Continues Options Continues Population 1 1 1 536481 154502			VA(s)
Jitter and Select Configure Results	Eye Diagram Analysis Overall Test Result Science Control (1997) Control (1997) Eye Diagram Analysis Overall Test Result Science (1997) Control (1997) Contro	Tools Pass Pass Pass Pass Pass Pass Pass Pa	Mouri 158.13mV 158.45mV 158.45mV 1.0462V 608.65mV	504 Dev 0.0000V 0.0000V 09.252mV 0.0000V	Next Market Mark	Vie Min 153, 154, 154, 154, 154, 154, 154, 154, 154		Or Options Options Population 1 1 153648 154502			
Jitter and Select Configure Pote	Constant of the second se	Tools Pass Fal Pass Fal Pass Pass Pass Pass Pass Pass	Mown 158.13mV 158.13mV 158.46mV 325.62mV 104.62mV 101.78ps	501 Dev 0.0000V 0.0000V 0.0000V 0.0000V 0.0000V 0.0000V 0.0000V 0.0000V	0 2000 2000 2000 200000000	Vie Min 158, 13mV 158, 13mV 152, 46mV 152, 46mV 152, 56mV 152, 56mV 152, 56mV 152, 56mV 152, 56mV 152, 56mV	P-P 3.0007/ 5.0007/	0 200 Options Coptions Population 1 1 153048 154502 1 1			VA(s)
Jitter and Select Configure Pots	Alexandrometry   Alexandr	Tools Pass Passfad Pass Pass Pass Pass Pass	Mourr 158.13mV 158.13mV 158.13mV 158.45mV 15845V 688.68mV 101.78ps 0.8000V	501 Dev 0.0000V 0.0000V 0.0000V 0.0000V 0.0000V 0.00000 0.00000	0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Vie Nin 152, 154, 154, 154, 154, 154, 154, 154, 154	P-P 0.0007/	Or 200 Options Population 1 1 153648 153648 153648 1 1 1 1 1			

To switch between SDLA Visualizer and DPOJET, use the Alt Tab keyboard combination or the navigation buttons (< and >) on the SDLA Main Menu. Use the TekScope application minimize button to minimize the scope window to view the DPOJET and SDLA applications.

# **SEE ALSO:**

- Configure Actions for Apply and Analyze Buttons
- Product Overview
- Understanding the System

# **Components and menus**

# Main menu in detail

Use the SDLA Visualizer **Main Menu** to configure the blocks, models, and test points, and to apply, plot and analyze the data.

The upper part of the circuit diagram shows the Measurement Circuit model, and the lower part shows the Simulation Circuit model. The arrows show the order in which SDLA processes the transfer functions. Note that for the Measurement Circuit part of the diagram, the ACTUAL signal flow is in the *opposite direction of the arrows*. For the Simulation Circuit, the actual signal flow direction is the same as the signal processing flow arrows.



#### Inputs

You can use either one or two inputs with SDLA Visualizer by selecting either Single Input or Dual Input mode. Changing these radio buttons will change the configuration panels here and elsewhere. The image above displays Dual Input mode. Press here to view Single Input mode.



# **Global BW Limit**

This displays the current bandwidth. Pressing on the **BW** button brings up the *Test Point and Bandwidth Manager*, where you can set or and *create custom BW limit filters*.

### Sources

The SDLA processing and analysis operate only on waveforms that are displayed on the oscilloscope. You can select from actively acquired channel signals, Math waveforms or reference waveforms. For a live acquired waveform, select its channel number. To recall a reference waveform, select **File>Reference Waveform Controls** in the oscilloscope menu. Then press **Recall** in the Reference menu to bring up the Recall browser.

# **De-embed Block**

The De-embed Block contains the circuit models that represent the actual hardware probe, fixtures, etc. that were used to acquire the waveforms with the oscilloscope acquisition system. Here, you can define the effects of the fixture, probe, scope and other acquisition and measurement hardware upon the DUT signal, re-normalize the S-parameter reference impedance, perform singled-ended to mixed mode conversion, reach the Block Configuration menu for Thru, File, RLC and T-line options, add and configure High Z, SMA probes, or interposer, and many other tasks. For more information, see the *De-embed/Embed Menu*.

# **Test Points**

Test points output waveforms that are displayed live on the oscilloscope. You may bring up the **Test Point and Bandwidth Manager** by pressing a test point on the system circuit diagram on the Main Menu. From here, you can configure the individual output waveforms and save test point filters. (When Dual Input mode has been selected on the Main Menu, you can also select test point modes.) You can also set a Global BW limit and create a custom BW limit filter. For more information, see *Test Point and Bandwidth Manager*.

# **Tx Block (Transmitter Modeling Block)**

The Tx Block represents the model of the serial data link transmitter that is driving both the Measurement Circuit model and the Simulation Circuit model. Pressing Tx on the Main Menu brings up the Tx Configuration Menu, where you can select files and view plots. It also gives you access to the Tx Emphasis Menu, where you can select emphasis, de-emphasis or pre-emphasis filters, read from FIR filters and make other choices. For more information, see the *Tx Block Overview*.

# **Embed Block**

The Embed Block allows the user to "insert" the channel based on its Sparameters, as a lossless transmission line, or as an RLC model, in order to observe the waveforms at the various test points on the Simulation Circuit model. Pressing **Embed** on the Main Menu brings up the *De-embed/Embed Menu*,. Use this for the same tasks as the De-embed Block above, except you cannot configure a probe.

# **Rx Block (Receiver Modeling block)**

The Rx Block represents the model for the serial data link receiver for the simulation side of the circuit drawing. Pressing **Rx** on the Main Menu brings up the Rx Configuration Menu. Here, you may apply CTLE equalization, perform clock recovery, and apply FFE/DFE equalization. Alternatively, you may set up an AMI model that uses imported equalization files to emulate actual silicon. For more information, see the *Rx Block Overview*. Note: the Rx load is defined in the Embed Block, not the Rx Block.

# Apply, Config and Analyze buttons

# Apply

By default, this computes test point filters and applies them to the scope. If any SDLA configuration is changed, run Apply to get updated results. Some configuration options are available, as described below.

#### Analyze

Pressing **Analyze** performs waveform analysis with the DPOJET application. The SDLA application is put into a sleep state and then the DPOJET application is started with the test point signals, and the recovered data and clock signals selected for analysis. The SDLA software may configure the DPOJET application to analyze the link quality with eye diagrams and jitter measurements. Note that you must first press the **Apply** button and wait for filter processing to complete before pressing the **Analyze** button. The DPOJET application must be installed for this transfer to work.

# Config

This lets you configure the action of the Apply button as well as the Analyze button with DPOJET, and to determine whether to use a new or a previously acquired waveform. *Press here for Apply and Analyze button configuration options*.

# Plot button

Press to show the results of running the enabled test points. *Press here for more about plots*.

# **Default button**

Press to restore the SDLA Visualizer system to its default settings.

#### Save button

Press to save the current SDLA Visualizer setup to a file with a .sdl file extension in the directory SDLA\Save recall.

**NOTE.** Only the SDLA setup is saved and recalled, not the entire oscilloscope setup.

# **Recall button**

Press to recall saved setup files and to return the software to a previous configuration.

# **SEE ALSO:**

- Product Overview
- Running a Test: Recommended Order
- Solving Problems with SDLA Visualizer

# **Test points**

# Test point and bandwidth manager

SDLA Visualizer provides up to 12 test points (when using REF for two), including four test points that can be moved on the schematic drawing. Up to six test point outputs are viewable on the scope graticule at one time (math plus reference). Press here for a Table of Test Point Descriptions.

**NOTE.** For a conceptual overview of how test points work, see Test Point Locations.



Test point	Position	Description
Tp1	Main	Measurement circuit loading the Tx block output
Tp2	Main	Simulation circuit loading the Tx block output, measurement circuit de-embedded
Тр3	Main	Rx block input. Simulation circuit loading the Tx block output, measurement circuit de- embedded

Test point	Position	Description				
Тр4	Rx Eq Data	Data output of the Rx block after equalization				
Тр5	Rx Eq Clock	Test point for the recovered clock output of the Rx block				
Тр6	De-embed Block	Movable test point with the measurement circuit loading the Tx block output				
Тр7	De-embed Block	Movable test point with the measurement circuit loading the Tx block output				
Тр8	Embed Block	Movable test point with the simulation circuit loading the Tx block output, measurement circuit de-embedded				
Тр9	Embed Block	Movable test point with the simulation circuit loading the Ty block output, measurement circuit de-embedded				
Тр10	CTLE	CTLE output				
Tp11	Тх	Thevenin equivalent voltage of the Transmitter model				
Tp12	Тх	Test point for the output of the Tx Emphasis block (if on)				

The Test Point and Bandwidth Manager is reached by pressing any test point on the Main Menu. Use this to configure the individual output waveforms, to save test point filters, to set the Global BW limit or to create a custom BW limit filter. You can also select test point modes if Dual Input is selected on the Main Menu. (If Single Input is selected, the **Select Tp Mode** column will not appear.) Scroll down for descriptions of each feature.

lap the desired to	est points to Math to see t	the processed waveform					
Tp On/Off	Map Tp to Math	Select Tp Mode	Label	Filters	Global BW Limit:	Delay	
Math1	None 💌	А-В 💌	NAA-B	Save	None	Keep Delay	
Math2	Tp1: Main 💌	А-В 🔻	Tp1A-B	Save	Auto	Remove Delay	
Math3	Tp2: Main 💌	А-В 🔻	Tp2A-B	Save	Custom	Adjust Delay	
Math4	Tp3: Main 💌	А-В 🔻	ТрЗА-В	Save			
Ref3	Tp5						
Ref4	Tp4						

# Tp On/Off

Controls which of the six (4 math and 2 reference) active test point waveforms are on or off. Each radio button lists the name of one of the available Math functions or a Ref memory waveform in the oscilloscope. If the button is off, then the waveform on the oscilloscope screen is turned off. If the button is on, then the waveform on the oscilloscope screen is turned on.

# Map Tp to Math

This drop-down menu allows a specific test point to be assigned to a math function of Math1, Math2, Math3, or Math4. The same test point may be assigned to more than one math slot.

**NOTE.** SDLA only processes and creates test point filters for the enabled test points. An enabled test point is a Tp that has been mapped to a Math or Ref waveform, and the corresponding Math or Ref is turned on.

# Select Tp Mode

This column is only visible when **Dual Input** has been selected on the Main Menu. Press here for more information.

This system maintains full 4-port modeling. Therefore, the test points are differential, and each contains a set of four possible waveforms (test point modes) to view. The options are:

- A: the waveform on the upper line of the test point
- **B**: the waveform on the lower line
- $\mathbf{A} \mathbf{B}$ : the differential waveform and
- (A + B)/2: the common mode waveform.

# Label

A label for the test point waveform can be entered into this box. It will appear on the oscilloscope screen along with the waveform.

# Save Filters

You can save each test point filter into the file folder you specify by pressing the **Save** button next to the test point label. For more information, see, *Saving Test Points*.

# **Filter Scaling Factor**

Filter Scaling Factor is located at the bottom of the configuration menu in the single input case only. It scales the test point filter coefficient according to the value.

SDLA Visualizer - Test Poi	nt and Bandwi est points to Mi	dth Configu ath to see ti	uration he processed wave	form					_	
Tp On/Off	Map Tp t	to Math		L	abel	Filters	Global BW Limit	Delay	١	
🔘 Math1	None	•			NA	Save	🔘 None	🔘 Keep Delay	ы	
🔘 Math2	Tp1	•			Tp1	Save	Auto	Remove Delay		
🔘 Math3	Tp2	•			Tp2	Save	Custom	Adjust Delay		
🔘 Math4	ТрЗ	-			ТрЗ	Save				
C Ref3	Tp5						BW: 25GHz		11	
O Ref4	Tp4	5	Scaling Factor	-						ОК
		[	V 80 % 🧾	2		Export f	ilters for 32-bit scope	2	J	

The small square check box is used to enable or disalbe the Scaling Factor. The Scaling Factor value is not effected whether the check box is on or not.

After the scaling factor is enabled and the main menu Apply is finished, the scaled filter coefficient value can be saved. The range of the scaling factor is between 20% to 200%. The default is 90%.

# **Plotting Test Points**

To plot test point transfer functions, return to the Main Menu and press **Plot**. Magnitude, Phase, Impulse and Step graphs are available. Press here for more information.

It is useful to always check these plots AFTER the **Apply** button on the Main Menu has been pressed, in order to verify that the results appear as expected. This helps ensure that no errors were made in setting up the configuration of the S-parameter blocks throughout the system.

For cases where the auto bandwidth limit setting has been used (see below), the plot will reveal whether or not the auto bandwidth limit is sufficient. If not, you may select **Custom** bandwidth and specify a more appropriate bandwidth limit filter. Then press **Apply** once more, and re-check the plots.



# **Global Bandwidth Limit**

This allows you to set up how the global BW limit filter will be applied to all test point waveforms. Under the **Global Bandwidth Limit** label, three options are available, including the option to create a custom filter. Press here for more information..

- **None.** No bandwidth limit filter will be applied to test points.
- Auto. All test point transfer functions will be checked. The one that crosses the -14 dB point at the lowest frequency will be determined. The bandwidth limit filter cutoff frequency will be set to that value.
- Custom. Allows you to create a bandwidth limit filter. The Custom option is most useful when the Auto bandwidth filter is not appropriate for your input data, or your test has specific bandwidth requirements. For more information, see *Creating a Custom Bandwidth Filter*.

# Delay

This allows you to control how SDLA Visualizer handles absolute and relative delay for the test points. By default, the absolute delay is removed.

Keep Delay: The absolute delay between all test point waveforms is maintained.

**Remove Delay:** This is the default setting. The absolute delay of the test point filters is removed, so that the test point waveforms all have the same events close to being aligned in time.
Adjust Delay: This button is only visible when the **Remove Delay** radio button is selected. Pressing that will bring up the Test Point Filter Delay Slider.

#### **Test Point Filter Delay Sliders**

The Delay Slider menu allows the relative delay of each test point filter applied to Math to be adjusted over a range of -1 ns to +1 ns.

There are four delay sliders, one for each math waveform on the oscilloscope display.

SDLA Visualizer - Test Point Delay Configuration	
Specify Delay for Test Point Filters	
	ns
M1 NA	0.100
	-0.137
M3 Tp2	-0.4276
M4 Tp3	0.277 OK
-1ns -0.5 0.0 0.5	+1ns

There are several ways to control the relative delay using a slider:

- enter a number in the text edit box next to the slider
- drag the slider button with a mouse
- fine position by pressing or holding down the arrow buttons
- course position by pressing or holding down on the space between the arrow button and the slider button.

Sliders that are assigned to the same test point will operate together, with their delays set to the same value.

As the delay is adjusted, the test point filters will be recalculated and will update live on the oscilloscope display. Hint: to obtain a more lively interaction, you can make the record length shorter temporarily while setting up delay.

#### SEE ALSO:

- Understanding Test Points
- Creating a Custom Bandwidth Limit Filter
- Saving Test Point Filters (Transfer Function)

**Saving test points** There is a separate **Save** button on the Test Point and Bandwidth Manager that is associated with each of the four possible test points that may be active. Simply press on **Save** next to the test point you are interested in.

In order to save a test point that was not enabled before Applying the model, you must return to the Main Menu and press **Apply** to recompute the test point filters.

**NOTE.** Test point filters are intended to work on oscilloscopes that have a 64-bit processor. However, if you wish to export these filters for use with a scope that has a 32-bit processor, then you'll need to edit the file to make it compatible. For more information, see . You can also press ? on the Test Point and Bandwidth Manager.

SDL	A Visualizer - Test Poi 1 - Map the desired to	int and Bandwidth est points to Math	Configuration to see the processed waveform					(	
ſ	Tp On/Off	Map Tp to N	Math .	Label	Filters	Global BW Limit	Delay		
	🔘 Math1	Tp1	•	Tp1	Save	🔘 None	🔘 Keep Delay		
	Math2	Tp1	•	Tp1	Save	Auto	Remove Delay		
	Math3	Tp2	•	Tp2	Save	Custom	Adjust Delay		
	Math4	ТрЗ	•	ТрЗ	Save				
	Ref3	Tp5				BW: 20GHz			
	C Ref4	Tp4	Scaling Factor						ок
			90 % ?		Export filters 1	for 32-bit scope ?		し	

Pressing any **Save** button in the Test Point and Bandwidth Manager will open up a folder browser. You may then either select a folder or specify a new folder:

Select Directory to Open			x
😳 🖉 🕨 🖉 Usen 🕨 P	ublic + Tektronix + TekApplications + SDLA + outpr	t filters • + Search output filters	P
Organize - New folder			
4 🔆 Favorites	- Norre	Date modified Type	
Desktop	1	No items match your search.	
38 Recent Places			
# 🧮 Desktop			
4 📷 Libseries			
Documents			
P # Music			
4 🛃 Videos			
My Videos	* 4		1
Folder	output filters		
		Select Folder Can	cel _

#### **Dual Input Mode**

If you have selected **Dual Input** on the Main Menu, then SDLA Visualizer will save 9 files into the specified folder. Eight of the files will each contain one of the test point filters. One of the files will contain all eight of the filters. If using math as the input source, make sure the test point is output to a different math.

The test point filter filename convention for Dual Input mode is:

<foldername>\_Tp<X><mode><source>.flt. File details.

<Foldername>: entered by user

<X>: test point number

<mode>: either A, B, Diff, or Cm

<source>: either Src1 or Src2, where Src1 relates to src1 on the Main Menu, and Src2 relates to src2.

A test point single file contains ASCII characters. The first character is "#" to identify a comment line. Numerous comments can be included. Variables and parameters can be included in comment lines of these forms:

# TpX differential test point filters

# [ DELAY ] 1e-09 is the delay parameter same as current arbflt format.

# [ SAMPLERATE ] 50e9

Eight lines contain the coefficients for the 8 filters, i.e.:

Line 1: TpXASrc1

Line 2: TpXASrc2

Line 3: TpXBSrc1

Line 4: TpXBSrc2

Line 5: TpXDiffSrc1

Line 6: TpXDiffSrc2

Line 7: TpXCMSrc1

Line 8: TpXCMSrc2

**NOTE.** For future releases of scope firmware it is planned that this file may be loaded into a new math function that can apply the filters according to the selected mode and sources.

#### Single Input Mode

For Single Input mode, only one filter file is saved for each test point when you press **Save** on the Test Point and Bandwidth Manager. This is for mode **A-B**, **differential**. You may save test point filters to a file with the following name format: <filename>.flt.

The ASCII file format contains comment lines that start with a "#". A line with **[DELAY] <value>** may be present in the file. The filter line contains a sample rate number followed by a ";" and the coefficients separated by commas.

#### **SEE ALSO:**

- Exporting Filters to Use with a 32-bit Oscilloscope
- Test Point and Bandwidth Manager
- Understanding Test Points

## Exporting filters for use with a 32-bit oscilloscope

Test point filters are saved to an arbflt ASCII file format, in order to allow them to be loaded into the oscilloscope's arbflt function in the math menu. These filters are intended to work on oscilloscopes that have a 64-bit processor. However, if you wish to export these filters to an oscilloscope with a 32-bit processor, then you'll need to edit the file to make it compatible.

The file format contains lines with comments preceded by the # symbol.

Next, there is a line that contains the sample rate value for the first entry, followed by ";" followed by the filter coefficients for the remaining entries separated by commas. (For further information on the filter file format, see *Understanding Test Points*.)

**NOTE.** If the radio button is selected on the Test Point and Bandwidth Manager, then the waveform timing may be off by one sample period.

To edit the file:

- 1. Open it up using Windows Notepad.
- 2. Add a comment line at the top of the file in order to document what sample rate the filter was designed to operate at. Enter # <sample rate value> where the sample rate value is the first element of the filter coefficient line.
- 3. Next, on the filter coefficient line, edit the first sample rate number to be an @ symbol. The @ symbol indicates that the filter will operate at all sample rates with the same set of coefficients.

Make sure that if you use this filter on a 32-bit scope, that the oscilloscope is set to the sample rate specified in the comment line above. The arbflt math function was designed to run only at the sample rate in the coefficient line and will normally blank out the waveform if the oscilloscope sample rate is changed to some other value. However, when the @ symbol is present, then the filter will run at all sample rates, but its response will be normalized to the sample rate. In other words, the filter will only work as desired when the scope is set to the sample rate the filter was designed for.

For example:

# Tp1 filter

# sample rate 50GS/s

@ <coeff1>, <coeff2>, <coeff3>, ... <coeffn>



**CAUTION.** Note that if you are using this filter with a scope with a 32-bit processor, and the scope is operated in IT mode (interpolated sample rate), then the sample rate readout on the screen is not actually the interpolated sample rate, but rather is the base sample rate before the interpolation. The filter would be operating at the interpolated sample rate.

In order for the filter with an (a) as the sample rate to operate with the correct response, the interpolated sample rate must be set to the rate for which the filter was designed. The user must manually do this when exporting to a scope using a 32-bit processor. You may determine the IT sample rate by computing 1 divided by the sample interval readout in seconds per point on the scope display.

Save test point filters for multiple sample rates	On real-time scopes, there may be a need to save a single Test Point Filter to cover multiple sample rates. SDLA can create one Test Point Filter for one sample rate. The following steps can be used to combine multiple Test Point Filters for each sample rate to a single Test Point Filter that covers all the sample rates that are needed.			
	To cover m number of sample rates SR1, SR2,, SRm:			
	<ol> <li>Set the sample rate of SDLA input to SR1, run SDLA to create the Test Point Filter for Tp<x>. Rename it to sdlaTpxxxx.flt.</x></li> </ol>			
	2. Set the sample rate of SDLA input to SR2, run SDLA to create the Test Point Filter for Tp <x>. Copy the sample rate and coefficient part of sdlaTp<x>.flt and paste it to the end of sdlaTpxxxx.flt.</x></x>			
	<b>3.</b> Repeat Step 2 for the remaining sample rates. The final Test Point Filter covering all the sample rates will look like:			
	# tp1 filter coefficients			
	5.000000E+10; <coeff1>, <coeff2>, <coeff3>, <coeffn1>,</coeffn1></coeff3></coeff2></coeff1>			
	1.000000E+11; <coeff1>, <coeff2>, <coeff3>, <coeffn2>,</coeffn2></coeff3></coeff2></coeff1>			
	2.000000E+11; <coeff1>, <coeff2>, <coeff3>, <coeffn3>,</coeffn3></coeff3></coeff2></coeff1>			
	SRm; <coeff1>, <coeff2>, <coeff3>, <coeffnm>,</coeffnm></coeff3></coeff2></coeff1>			
	SEE ALSO:			
	Test Point and Bandwidth Manager			
	<ul> <li>Understanding Test Points</li> </ul>			

Saving Test Points

# Creating a custom bandwidth limit filter

When de-embedding fixtures, cables or other equipment, a bandwidth limit filter is usually necessary to obtain a usable result. In such cases, a bandwidth limit filter can reduce the gain on noise by filtering out the high frequency components. SDLA Visualizer gives you control over the pass band, transition band and stop band responses, which affect noise attenuation, rise time, preshoot and overshoot.

Follow these steps to create a custom filter:

1. Press a test point on the Main Menu to bring up the Test Point and Bandwidth Manager. (Pressing **Global BW** on the Main Menu also brings up the Test Point and Bandwidth Manager.)

LA Visualizer - Test Poi 1 - Map the desired to	int and Bandwidtl est points to Math	h Config n to see t	uration he processed waveform					, 0
Tp On/Off	Map Tp to	Math	Select Tp Mode	Label	Filters	Global BW Limit	Delay	
Math1	None	•	A-B(DIFF) 💌	NA	Save	None	🔘 Keep Delay	
C Math2	Tp1	•	A-B(DIFF)	Tp1	Save	Auto	Remove Delay	
🔘 Math3	Tp2	•	A-B(DIFF)	Tp2	Save	🔘 Custom	Adjust Delay	
Math4	ТрЗ	•	A-B(DIFF)	ТрЗ	Save			
Ref3	Tp5					BW: 5GHz		
C Ref4	Tp4							ОК
					Export fi	Iters for 32-bit scope	2	

2. Under Global BW Limit, select Custom and then press Setup BW. This brings up the Bandwidth Limit Filter Design Menu.



- 3. Set values in the BW GHz, Stopband GHz and Stopband dB fields.
- 4. Press **Apply** to generate the bandwidth filter and save it for use in SDLA's internal data base. The filter response is plotted for review. Optionally, press the **Export** button to save the filter to a file for uses outside SDLA.
- 5. Press Close to return to the Test Point and Bandwidth Manager.

#### **SEE ALSO:**

Test Point and Bandwidth Manager

### **De-embed block**

#### **De-embed block overview**

The **De-embed Block** contains the Measurement Circuit models that represent the actual hardware probe, fixtures, etc, used during waveform acquisition. Press **De-embed** on the Main Menu to bring up the *De-embed Menu*, which represents a cascade of 4-port S-parameters. These menus provide multiple ways to model the blocks, as shown below:



**NOTE.** If the DUT has large attenuation, de–embedding results will have limited bandwidth, ringing, and slower rise-time. If you increase the bandwidth limit filter transition band, it can reduce ringing at the expense of phase and magnitude error at the higher frequencies, and at the expense of increased noise.

SDLA Visualizer handles *cross-talk and reflections* in both directions through the cascade.

The De-embed Block lets you model a variety of different configurations. Here are some possibilities.

#### **De-embed Block – Possible Configurations**

- 4-port single-ended S-parameter file
- 4-port differential S-parameter file
- Two 2-port S-parameter files
- FIR filter files (time domain)
- Transfer function files (frequency domain)
- SMA probe model
- Interposer/probe/scope model
- Mixed-mode S-parameter files
- Various RLC series or parallel configurations
- Lossless Transmission line model
- 3-port probe model file
- 1-port load S-parameter file
- 2-port load S-parameter file
- Nominal load values
- TDT waveform

**NOTE.** For step-by-step examples of de-embedding and embedding, see *Examples* and troubleshooting on page 109.

#### **SEE ALSO:**

De-embed/Embed Menu

#### De-embed-Embed menu

The De-embed/Embed Menu allows you to define where the acquired waveforms enter the signal flow path of the system.

**NOTE.** For step-by-step examples of de-embedding and embedding, see *Examples* of Tasks and Troubleshooting.



Pressing **De-embed** on the Main Menu brings up the De-embed Menu; pressing **Embed** brings up the Embed Menu. These menus display a diagram of 8 cascaded 4-port S-parameter block models, plus a load model at the end, which you may configure, plot and save. You may also select the locations of two movable test points, configure the load, and configure probes (de-embed menu only).

**NOTE.** Parameter changes in the De-embed Block may affect all test points in the De-embed Block as well as in the Embed Block. However, parameter changes in the Embed Block cannot affect test points in the De-embed Block.

#### Differences between De-embed Menu and Embed Menu

In the De-embed cascade model diagram, shown above, note that the processing arrows in the diagram flow from right to left. In the Embed cascade model diagram, shown here, the arrows flow from left to right. Also, the De-embed Menu includes probe options, while the Embed Menu does not. Each menu has its own specialized Load Block (the final block) with a menu for configuration options. Other than that, the functionality is the same.



On both menus, there are four tabs on the left side of the screen:

#### Cascade Tab

The cascade diagrams show the S-parameter modeling blocks. The two arrow buttons under **Move** may be used to move **Tp6** and **Tp7** on the De-embed Menu, and **Tp8** and **Tp9** on the Embed Menu. Pressing on these arrows repositions the movable test points. In the De-embed Menu only, you may configure SMA or High Z probe options. For details, see *Configuring Probes*.

Pressing on any of the cascade blocks **B1-B8** brings up the individual block's configuration menu. For details, see *Block Configuration Menu*.

The final block of the cascade is labeled either **Scope**, **SMAProbe** or **Load** in the De-embed Menu, and labeled **Rx Load** in the Embed Menu. When you press this block, the appropriate configuration menu comes up where you may determine the load of the output ports. For details, see *Load Configuration Menu*.

#### Normalize Tab

SDLA requires all ports to have a reference impedance of 50 Ohms. You can use the Normalize Tab to re-normalize the S-parameters to the correct reference impedance for each port before reading them into SDLA Visualizer De-embed or Embed Blocks. For details, see *How to Renormalize S-Parameters to Different Reference Impedances*.

SDLA Visualizer - De-embed				
Renormalize S-parameter refere	ence impedance for each port			
Cascade	Tx+	Port 1 50 Browse <filename></filename>	Port 2 50	Apply
Resample	Tx-	50 Port 3	50 Port 4 Rx-	Save

#### **Convert Tab**

Here, you can set up singled-ended to mixed mode S-parameter conversion. Once you load a file, the **Save** and **Plot** buttons become available.

**NOTE.** It is preferred practice to leave the data in single-ended format, not mixedmode, for uses that are internal to SDLA.



#### **Resample Tab**

SDLA requires all S-parameter files to be uniformly sampled. Files with nonuniform sampling can be resampled using the Resample tab, which works with any number of ports.



After loading a file you can change the suggested uniform sampling interval, then plot and save the uniformly sampled version of your data.

If you click the Plot button on the right, three new windows appear with the following graphs.

- dB magnitude overlays of the original and resampled data.
- Phase overlays of the original and resampled data.
- dB magnitude plots of the resampled data with the standard subsidiary plotting options.

The two overlay windows display the original S-parameters in red and the resampled S-parameters in green. With a good resampling frequency spacing all the red will be covered by green. The following graph illustrates a case in which the lowest frequencies were oversampled and some information was lost in resampling. While in this graph the discrepancy is acceptably small, it demonstrates what to look for when evaluating the accuracy of the resampling.



#### **Control Buttons**

On the right side of the screen, the buttons vary depending on the context. On the Cascade Tab, these buttons appear:



#### Save

When you press Save, you can select from two options.

- Cascade Setup: Saves the parameters for the current cascade into a setup file. The cascade setup can be recalled later by pressing Recall. Note the cascade setup file is different from the main SDLA setup file, as the cascade setup file only contain the cascade configuration.
- 4-port s4p file: Allows you to save a single 4-port S-parameter set for the combination of all the blocks in the cascade, excluding the load (final) block and the Tx Block. This allows for general purpose cascading of 4-port S-parameters exported to a file that may be used in other simulation tools, or may be loaded back into a cascade block in order to combine with additional blocks. This is useful if more than 8 blocks need to be combined together.

SDLA Visualizer - Save Configuration	
Select which file to save	
<ul> <li>Cascade Setup</li> <li>4-port s4p file</li> </ul>	Save

#### Recall

Recalls a setup file saved using the Save button.

#### OK

Returns you to the Main Menu.

#### **SEE ALSO:**

- Block Configuration Menu
- Load Configuration Menu
- De-embed Block Overview
- Embed Block Overview

#### How to re-normalize S-Parameters to different reference impedances

Use the **Normalize Tab** on the *De-embed/Embed Menu* to take an S-parameter set that was not normalized to 50 Ohms and normalize it to 50 Ohms, which is the reference impedance required by SDLA Visualizer. The reference impedance is the value that ports are loaded with at the time the S-parameters are measured. (Reference impedance should not be confused with the Load impedance in the cascade, which may be any value you desire.)

The tab may be used to normalize the port reference impedances to any value for any port for uses external to SDLA, or for analyzing the effects by looking at the plots.

#### Using the Normalize Feature

- 1. Press on the Normalize Tab.
- 2. Press **Browse** and select the 4-port S-parameter file that is to be renormalized.

🚺 🛛 SDLA Visualizer - De-em	bed	
Renormalize S-paramete	er reference impedance for each port	
Cascade Normalize Convert Resample	Dr.+     Port 1     Port 2       50     50       errorse       filename>       50     50       Port 3     Port 4   Rx-	Apply Plot Save OK

- **3.** Touchstone 1.0 format only supports one impedance value for all ports. That number will be displayed in the port edit boxes. Now, edit each box to 50 Ohms for internal use with SDLA. Edit any port to desired values for external use outside of SDLA or for observing differences in the plots.
- 4. Press the local **Apply** button to compute the re-normalized set of Sparameters. Note that this data is not used by any SDLA blocks, unless you save the data to a file and then load it into a block in the Cascade Tab.
- **5.** Press **Plot** to observe the original S-parameters overlaid on the re-normalized set. The plot shows the original S-parameter data in gray traces and the new re-normalized plots in various colors. Use the zoom tool on the plot tool bar to zoom in on the detail of each individual plot. A cursor tool allows the read out of trace data to be performed. There is also a trace marking tool.



6. Press **Save** to create a Touchstone 1.0 file containing the re-normalized data. If all four ports have the same reference impedance, then a standard file will be written with that value in the option line. However, if some ports have an impedance setting that is different than the other ports, the system will place 1 as the impedance in the options line and write a comment line with

```
! [ IMPEDANCE ] <value1> <value2> <value3> <value4>
```

You can load the re-normalized file into any block in the Cascade Tab diagram.

If required, the re-normalized file can be read back into the tool and restored back to the original reference impedance values.

#### **SEE ALSO:**

De-embed/Embed Menus

**Configuring probes** You may choose from three probe options on the De-embed Menu:

#### None

For this option, no probe is used. The input waveform signals from the scope are represented at the load block on the Cascade Tab diagram (the final block), and are indicated by the labels **src1** and **src2**. A typical use case might be where a fixture and two cables are attached to the transmitter in order to acquire signals into **Ch1** and **Ch2** of the oscilloscope. To model this, you would choose **Dual Input** on the Main Menu, and then select **Ch1** and **Ch2** as the waveforms that are labeled **src1** and **src2**.

#### SMA Probe

This option can only be selected if **Single Input** has been chosen on the Main Menu. This selection specifies that a single waveform will be obtained from an SMA probe to be input into the De-embed Block. SDLA will assume this waveform was acquired through a 3-port SMA probe which had equal but opposite polarity signals at the input to each cable. The SMA probe S-parameter set includes the cable pair that comes with the probe. The load (final) block of the Cascade Tab diagram will only allow the SMA probe model for termination of the cascade, and will be labeled "SMAProbe".

Pressing **SMAProbe** on the Load block (final block) of the De-embed Menu Cascade Tab diagram brings up this menu:



#### Options

- Port Numbers: Select the port numbers to make the correct connection polarity for the probe.
- Probe model: Use this drop-down menu to select the probe model that is connected to the oscilloscope. This selection will cause a file browser window to open so that the correct S-parameter file may be selected. Pressing the Load button on this menu will also open the file browser menu to the same folder determined by the probe model selection.
- Scope Browse: Press this button to load an S-parameter file for the scope. This will open a browser and the user can select the correct file according to the scope model in use.
- General Purpose Block Browse: Press this button to load an S2p file for the general purpose block. This general purpose block may be used to model a RF switch.
- General Purpose Block Thru: Press this button to make the general purpose block be ideal Thru.

#### High Z Probe

This option can only be selected if the **Single Input** radio button has been selected on the Main Menu. This selection specifies that a single waveform will be obtained from a High Z probe to be input to the De-embed Block. When you press the **High Z** radio button on the left, a **Probe** button appears on the Cascade Tab diagram. A right arrow ">" button also appears below the words High Z. Pressing on the arrow button moves the probe to a different location on the diagram.



Pressing the **Probe** button on the Cascade Tab diagram brings up the Probe Path Configuration Menu:



Under **Configure**, select the circuit configuration desired. There is choice of **Probe/Scope**, shown above, or **Interposer/Probe**, shown below:







#### Or Interposer s3p, shown below:



Options

- **Probe model:** Use this drop-down menu to select the type of probe in use. This opens a file browser to the correct probe directory. The user must then select the correct file according to the probe attenuation setting and according to the tip in use. For help selecting the correct tip name for the file, see *Probe and Tip Selection*. You may also press the ? button on the Probe panel.
- **Probe Load**: Press this to load a probe file according to the currently selected model.
- **Port numbers:** Select the desired port numbers to obtain the correct polarity connection for the probe.
- Scope Browse: Press this button to load an S-parameter file for the scope. This will open a browser and the user can select the correct file according to the scope model in use.
- Scope Ideal: Press this button to use an ideal 50 Ohms termination to model scope.
- Interposer Browse: Press this to load 4-port S-parameters for the interposer model. This supports a simplified interposer model for a differential clock or strobe line pair. Where the memory input and controller input are assumed to be one port per line, and the connect to the probe are the other two ports.
- Interposer Thru: Press this to set the interposer to model two isolated ideal three way line connections.
- Interposer s6p Browse: Press this to load 6-port S-parameters for the s6p interposer model. This supports a complete two line differential interposer model for a differential clock or strobe line pair. Where the memory input and controller input are modeled as two ports per line, and the connection to the probe are the other two ports. The port number assignments for the s6p parameter is as follows: the Tx side positive leg is connected to the top port on the left, the Tx side negative leg is connected to the second port from the bottom on the left, the Rx side positive leg is connected to the bottom port on the left, the probe positive leg is connected to the bottom port on the left, the probe positive leg is connected to the bottom port on the left, the probe positive leg is connected to the port on the right, the probe negative leg is connected to the bottom on the right.
- Interposer s6p Thru: Press this button to set the interposer to model two isolated ideal three way line connections.
- Interposer s3p Browse: Press this to load 3-port S-parameters for the s3p interposer model. This supports an interposer having the same 3-port model on the positive leg and on the negative leg, and there is no coupling between the two lines. This configuration is useful for single line signaling. The port number assignments for the s3p parameter is as following: the Tx side line is connected to the top port on the left, the Rx side line is connected to the bottom port on the left, and the probe tip is connected to the port on the right
- Interposer s3p Thru: Press this button to set the interposer to model two isolated ideal three way line connections.
- Block 1 Browse: Press this button to load an S4p file for general purpose block 1. This general purpose block may be used to model extra probe tip resistors.

- Block 1 Thru: Press this button to make general purpose block 1 be ideal Thru.
- Block 2 Browse: Press this button to load an S2p file for general purpose block 2. This general purpose block may be used to model an RF switch.
- Block 2 Thru: Press this button to make general purpose block 2 be ideal Thru.
- Label: Edit this label to change the probe block label in the De-embed menu.
- **Filenames:** The filenames for the loaded S-parameter files are listed at the bottom of the menu.
- **Interposer in Plot:** Press this button to plot the interposer data. The interposer data could be s4p, s6p or s3p.
- **Block1 in Plot:** Press this button to plot the s4p data in Block1.
- Probe in Plot: Press this button to plot the s3p data of the Probe
- Block1 in Plot: Press this button to plot the s2p data in Block2.
- Scope in Plot: Press this button to plot the s1p data of the scope.

#### SEE ALSO:

De-embed/Embed Menu

**Probe and tip selection** This topic explains how to identify probe tips and then select the proper file name for the configuration in use.

For the P7313 and 7380 probe models, the following set of tips is available; however, only four of these tips are supported with S-parameter sets. These are:

- HBW Right Angle Flex,
- HBW Straight Flex,
- Medium Flex Small Resistor,
- and Short Flex Small Resistor.



Shown below are the tips for P75xx family of probes. However, only the Performance Solder Tip is currently supported with S-parameter sets.



#### Choosing the correct 3-port S-parameter file

Press **De-embed** on the Main Menu. On the Cascade Tab, under **Probe**, select the **High Z** radio button. Then press the **Probe** button on the diagram. This brings up the Probe Configuration menu:

SDLA Visualizer - Probe Configuration				
Define S-parameter files for the Probe and Scope				
Configure Probe/Scope	Probe + Port 1 v ? Ideal v Port3 Port 2 v Filename	Scope Browse Ideal Ch1	Label Probe	Piot Probe Scope OK

On the **Probe** drop-down menu, choose the correct model that is in use. For example, P7380 was selected in the image above. The browser will then open the correct directory for the selected probe model:

Choose File for model, tip, and attenuation		
♥ P7380 ► P7380_solderTips	✓ 4y Search P7380_solderTips	0
Organize 🔻 New folder	1==	
Favorites     Desktop     Downloads	Name P7380_5X_Nominal_HBWRightAngleFlex.s3p D7380_5X_Nominal_HBWStraightFlex.c2p	
Recent Places	<ul> <li>P7380_5X_Nominal_MediumFlexSmallResistor.s3p</li> <li>P7380_5X_Nominal_ShortFlexSmallResistor.s3p</li> </ul>	
<ul> <li>Elbraries</li> <li>Documents</li> </ul>	<ul> <li>P7380_25X_Nominal_HBWRightAngleFlex.s3p</li> <li>P7380_25X_Nominal_HBWStraightFlex.s3p</li> <li>P7380_25X_Nominal_MediumFlexSmallResistor.s3p</li> </ul>	
<ul> <li>My Documents</li> <li>Public Documents</li> <li>Music</li> </ul>	P7380_25X_Nominal_ShortFlexSmallResistor.s3p	
▷ 🔄 Pictures File name:	• • (".s3p) •	Þ
	Open 🔽 Cancel	4

Choose the correct file for the probe's current settings. Note that the file name contains the probe model, and a gain number such as 5X or 25X, and it also contains a name for the tip that is in use. You'll need to correlate the tip name in the filename using the probe tip images shown above.

The gain setting for a probe is indicated by LEDs on the probe comp box that plugs into the oscilloscope input channel connectors.

**NOTE.** Tri-mode probes, such as the 75xx models, are able to operate under 4 possible modes. However, for SDLA, the S-parameter support currently only allows a choice for, which is identified as selection on the probe comp box that plugs into the scope.

#### SMA Probes

The SMA probe S-parameter files include the matched pair of SMA cables that come with the probe. There are no tip selections. The attenuator choice is included in the file name and the LEDs on the probe comp box indicate which setting is in use.

#### Scope Settings when Using a Probe with SDLA

The oscilloscope DSP filters must be turned on while using the probe in conjunction with SDLA. To check that they are turned on, go to the oscilloscope's Vertical menu. Make sure that **Digital Filters (DSP) Enabled** radio button is selected. In addition, go to the scope's Vertical menu **Chan X** tab on the left, and select the channel which the probe is connected to. Then, on the scope menu, select **Vertical > Probe Cal**. Press the **Select** button to bring up the probe tip selection menu. Make the radio button selection in this menu match the tip that is in use on the probe. This insures that probe DSP is turned on. (Note: not all tips are supported with S-parameters, as mentioned above.)

Then, SDLA will correctly de-embed the current probe DSP response, and replace it with the probe combined with user data, and produce the results in a math waveform slot on the scope display. This way, channel X of the scope will have the nominal filter response that does not include user data for the probe connection, but the test point math waveform created by SDLA will represent the response of the system that includes both the probe and its actual connection to the DUT.

#### P7520A, P7625, P7630 and P7633 probes

For these probes, the S-parameters are stored internally in the probe. Therefore, they are loaded into SDLA directly from the scope. No file browser is opened. A probe must be plugged into the oscilloscope source channel in order for the S-parameter set to be loaded into SDLA.

#### **User DATA probes**

In some cases, such as for interposer setups where custom modified probe tips are sometimes used, it may be that the oscilloscope does not have nominal DSP probe data. In this case, on the scope menu, select Vertical > Probe Cal. Press the Select button to bring up the probe tip selection menu. Choose Other Tip (no DSP). This way, the scope will not apply DSP for the given probe model. Instead, you may use the user probe data that is selected from the probe drop-down menu under User. A file browser will open, and you may load the custom probe data S-parameter set for use with the interposer. This data may be provided on a custom basis from Tektronix or from other sources, such as a custom simulation model.

NOTE. You must have saved S-parameter data there prior to loading.

#### **SEE ALSO:**

- Configuring Probes
- De-embed/Embed Menu

#### Block configuration menu

Use the Block Configuration menu to configure the cascaded S-parameter modeling blocks **B1–B8** in the De-embed/Embed Menu. (To configure the final block, use the *Load Configuration Menu*.)

The Block Configuration menu has four tabs that offer different categories of models:

#### Thru Tab

The Thru block model is used when the block is not a necessary part of the cascade. It represents an ideal model that has no effect upon signals in the system.

SDLA Visualizer - Block Configurat	ion			<b>— — X</b>
Select B1 model and the applicable fil	es if required			
Thru File RLC	Port1 v	Port2 V	Label B1	
	Port3 💌	Port4 💌		ок

File Tab

🛃 SDLA Visualizer	- Block Configuration		
Select B1 model	and the applicable files if required		
Thru	Model 4-Port Single-ended	Filename Port1 • Port2 •	Label B1 Check
RLC T Line	Scaling Factor	Port3  Port4  Port4	Passivity Port assignments OK

The File Tab allows you to choose from six models represented by data read from files:

#### 4-Port Single-ended

Models a single 4-port S-parameter set, as shown above. You can load a 4-port Sparameter file to represent the block. All S-parameter terms in the data are taken into account when computing test point transfer functions.

#### 4-Port Differential

Models a mixed mode S-parameter set. Press here for more information.

You can load a 4-port S-parameter file that has been saved with a mixed mode format for the data. The differential block allows the filename and path for the mixed mode S-parameters to be specified using a **Browse** button. The block is shown with two differential ports, but physically it still has 4 single-ended ports. SDLA will convert the mixed mode data into a single-ended data format for use within the cascade of blocks.

SDLA supports two ways of organizing the S-parameter data in the matrix. **Typical** shows a typical arrangement of mixed mode S-parameter data read from the file. **Alternate** provides a second arrangement.

**NOTE.** The mixed mode representation of S-parameter data is not supported by the Touchstone 1.0 file format. Therefore, only two organizations of the file data is supported, as shown below:



#### 2-Port

Models two 2-port S-parameter sets. Press here for more information.

This allows two 2-port S-parameter files to be loaded into the 4-port block model. SDLA will convert these into one 4-port S-parameter set with ideal cross-coupling terms set to zero.

A common use for this choice would be to represent a pair of shielded cables connected between a fixture and the scope.

🛃 SDLA Visualize	r - Block Configuration		
Select B1 mode	el and the applicable files if required		
Thru File	Model 2-Port 🔹	Filename Port1  Port1  Port2  Port2	Label B1 Check
T Line	Scaling Factor ☑ 90 % ?	Port1  Filename	Passivity Plot

#### **Transfer Function**

Models a frequency domain set of complex data. Press here for more information.

This allows two files containing transfer function data in the frequency domain to be loaded to represent the block. The file will be in the s1p Touchstone 1.0 format. This format contains a frequency column, a real or magnitude column and an imaginary or phase column. SDLA will convert these into a single 4-port S-parameter set with ideal cross-coupling and reflection coefficient terms set to zero. A common use for this choice would be to represent a pair of shielded cables connected between a fixture and the scope.

🛃 SDLA Visualizei	r - Block Configuration		
Select B1 mode	l and the applicable files if required		
	Model	Filename	Label
File		Port1 v Port2 v	Check
T Line	Scaling Factor	Port3 v Port4 v	Passivity Plot OK
		rilename	

FIR

Models a time domain impulse response. Press here for more information.

This allows two files containing FIR filter coefficients data in the time domain to be loaded to represent the block. The file shall be in the oscilloscope arbflt() ASCII format. SDLA will convert these into one 4-port S-parameter set with ideal cross-coupling terms set to zero. The reflection coefficient terms are also set to zero.

The file may contain comment lines with # as the first character. The file may contain multiple lines of filter coefficients, where the first number followed by a ";" is the sample rate. The remaining numbers are the filter coefficients separated by space comma.

ick Configuration		
the applicable files if required		
Model	Filename	Label
FIR 💌	Port1 - Port2 -	B1
Scaling Factor	Port3  Port4  Filename	Plot
	Model       Fire       Scaling Factor       V     90     % ?	Model       Filename         FIR       Port1         Port1       Browse         Port2       Filename         Scaling Factor       Port3         Ø       % ?         Port3       Filename         Filename       Filename

#### High Z probe

Models the loading of the probe. Press here for more information.

This allows a 3-port S-parameter set representing a probe to be loaded. This model places the probe tips on line A and line B in parallel with the signal path lines. A potential use would be to observe how oscilloscope or logic analyzer probes load the system.

SDLA Visualizer -	Block Configuration		<b>-</b> X
Thru File RLC T Line	Model High-Z probe	Filename Port1  Filename Port2  Filename Port2  Filename Port2  Filename	Label B1 Plot OK

**NOTE.** This model is representing probe loading effects only. There is no acquired waveform entering the simulation system from this model.

#### **S** Parameter Scaling

S Parameter Scaling is used to scale the S parameter. The small square check box is used to enable or disable the Scaling Factor. The Scaling Factor is not changed with it.

When enabled, the scaled S parameter is plotted and applied to the cascading. Be aware, the crossing and reflection terms are zeroed out during the sacling process, even at the 100% scaling. The default is 90%. Scaling is from 20% to 200%.

#### **RLC** Tab

This allows RLC (resistor, inductor, and capacitor) elements to be used to model the block. SDLA computes a set of 4-port S-parameters that will be used in the cascade. The cross-coupling terms will be set to zero.

SDLA Visualizer - B	Block Configuration		
Thru File RLC	Model Series1  Rohms 50 LnH I CoE	 Label B1	
T Line	Image: Spin for the spin f		Plot

The RLC Tab **Model** drop-down menu offers six different series and shunt configurations of RLC networks. Press here for more information.

#### Series 1

Represents a series RLC network in series with each line (shown above).

#### Series 2

Represents a parallel RLC network in series with each line:

SDLA Visualizer - Blo	ock Configuration	
Select B1 model and	the applicable files if required	
Thru File RLC T Line	Model Series2 ▼ Rohms ✓ 50 L nH ✓ 1 C pF ✓ 1 Freq Space MHz 10	Label B1 Piot OK

#### Shunt 1

Represents series RLC networks in shunt with each line:

🛃 SDLA Visualize	er - Block Configuration		
Select B1 mod	el and the applicable files if required		
Thru File RLC T Line	Model Shunt1 Rohms Shunt1 Shunt1 Rohms So LnH 1 CpF 1 Freq Space MHz 10	Label B1	Piot

#### Shunt 2

Represents parallel RLC networks in shunt with each line:

SDLA Visualizer -	Block Configuration Ind the applicable files if required		
Thru File RLC T Line	Model          Shunt2       R ohms         Image: Shunt2       50         L nH       1         Image: C pF       1         Image: Image: Shunt2       1         Freq Space MHz       10	Label B1	Plot

#### CRLC

Represents a shunt capacitor, series inductor, series resistor and a shunt capacitor with each line:

SDLA Visualizer - Blo	ock Configuration the applicable files if required		
Thru File RLC T Line	Model CRLC ▼ S0 L nH ✓ 1 C pF ✓ 1 Freq Space MHz 50	Label B1	Piot OK

#### RLCC

Represents a series resistor, series inductor and two shunt capacitors with each line:

🛃 SDLA Visualizer - B	llock Configuration		
Select B1 model and	d the applicable files if required		
Thru File RLC T Line	Model RLCC ▼ Rohms 50 LnH ✓ 1 CpF C2pF ✓ 1 Freq Space MHz 50	Label B1 C C2 Flot OK	

#### R, L, C, C2 edit boxes

These determine the values for these components.

#### R, L, C, C2 check boxes

When checked, the **R**, **L**, **C**, or **C2** value will be included in the circuit. When not checked, the R, L, C, or C2 element will be replaced with a short if it is a series element, or with an open if it is a shunt element.

#### Freq Space MHz

This edit box specifies the frequency spacing for the S-parameter set that will be computed for this network. Frequency spacing determines the time duration the S-parameter covers.

#### Label

This edit box determines what label will appear on the block diagram in the De-embed/Embed Menu, depending on which block this is located in.

#### Plot

This will open a new window containing all the plot menus for the various ways in which the 4-port S-parameter set may be viewed.

#### OK

This menu button closes the block menu and returns to either the Embed or De-embed menu, depending on which one was used to open the block.

SDLA Visualizer - Block Configur	ation			X
Select B1 model and the applicable	files if required			
Thru File RLC T Line	Z0 ohms 50 Delay ns 1 Freq Space MHz 10.0	1   Transmission Line     3   Transmission Line	2 Label B1	Piot OK

This allows you to define a lossless transmission line. SDLA computes a set of 4port S-parameters depending on the parameter settings in the edit boxes. The cross-coupling terms shall be set to zero. Press here for more information.

#### Z0 ohms

This allows you to specify the characteristic impedance of the transmission line pair.

#### Delay ns

This allows the delay through the transmission lines to be specified in ns.

#### **Freq Space MHz**

This specifies the frequency spacing for the 4-port S-parameter set that will be created for the transmission line model.

#### Plot

This may be used to view the characteristics.

#### **SEE ALSO:**

- De-embed/Embed Menu
- Load Configuration Menu

# **Load configuration menu** The final block in the De-embed/Embed Menu cascade diagram is where you determine what will load the output ports of the cascaded circuit. Pressing it brings up the Load Configuration menu. (To configure blocks **B1-B8**, use the *Block Configuration Menu*.)

#### **De-embed Cascade Load Block**

The final block on the **De-embed Menu** cascade diagram is labeled either **Scope**, **SMAProbe** or **Load**. When you press it, a different Load Configuration Menu will come up depending on how you have configured your model. For example, the image below shows the default menu that comes up without a probe:

SDLA Visua	alizer – Load Block Configuration			x
Select th	ne load model and the applicable file(s) if re-	quired		
	Model Nominal	ohm Port 1 v 50 ohm Port 2 v 50	Label Load	

If, instead, you have configured it for an SMA Probe, the following menu comes up. The middle block allows for the effects of cabling between the probe and the scope. For further information, see *Configuring Probes*.



If you have configured it for TDT High Z the following menu comes up. Blocks B1 through B7 are ignored and B8 is defined by time domain waveforms.



Clicking the B8 button displays the following menu in which the B8 block can be defined by a reference and a TDT waveform.

🛃 SDLA Visualiz	er - Block Configuration				
Select B8 model and the applicable files if required					
	Model	Filename	Label		
Thru	TDT Waveform 💌	Ref	B8		
File			Check		
T Line	Scaling Factor	тот	Plot		
		Filename			

#### **Embed Cascade Load Block**

The final block on the **Embed Menu** cascade diagram is labeled **Rx Load**. Pressing on it brings up the Load Configuration Menu show below, which allows you to determine the model that will load the output ports of the simulation circuit. In many cases, this would model a physical receiver.

The impedance can be modeled as a nominal value, one 2-port S-parameter block, or two 1-port S-parameter blocks by choosing one of the options in the **Model** drop-down list. By default, SDLA assumes 50 Ohm impedance. Below is the menu that comes up with one 2-port S-parameter block:

SDLA Visualizer – Block Configuration     Select the load model and the applicable file(s) if	required	
Model 2 Port	Filename	Label Plot
	Port 2 V	ок

#### **SEE ALSO:**

- De-embed/Embed Menus
- Block Configuration Menu

## Plots

**Plots** Plots are essential aids for setting up and verifying the system. SDLA Visualizer provides a variety of context-sensitive plots that show the results of running the enabled processing blocks and the test points. These can be used along with DPOJET and oscilloscope plots for tasks such as analyzing the quality of the S-parameters, verifying the configuration of each block as you configure the circuits, or determining port numbers.

Navigation features include zoom (+) and pan tools; some plots also include measurement cursor tools.



For some examples of how to troubleshoot using S-parameter plots, see *Using Plots for Troubleshooting S-parameters*.

Here are just some of the SDLA Visualizer plots available:

- Comprehensive S-parameter plots. Extensive plots of the S-parameters allows for full insight into the system characteristics. An Overlay feature is available for some plots. Some examples of S-parameter plots:
  - 6-port, 4-port, 3-port, 2-port, 1-port plots

Use these plots to:

- Quickly verify port assignment. You can tell by the shape of the curve which one is insertion loss.
- Validate that the magnitude is correct. Typically, the insertion loss should be higher at high frequencies; reflection and cross talk is typically higher at higher frequencies.
- Do a passivity check. None of the S-parameters of a passive system should go above 0 dB in the frequency domain. All data can be less than 0d and yet the system may still violate passivity where the total output power is more than the input power. Note that viewing the plots is a first step only; for a more extensive check, go to a file load tab in a block menu. For cases where the S-parameter data is loaded from a file, a passivity check button becomes available.
- Check whether models are Single-ended or Mixed Mode. For a single-ended system, transmission terms such as s21, s12, s34, and s43 are equal for an ideal passive system. However, if the data had been converted to mixed mode, the common mode transmission response is generally noticeably different than the differential transmission. In other words, S34 and S43 would equal common mode, and would look different than the differential mode response plotted in S21 and S12.



Impedance vs. magnitude (Plot Z(f) button)

These plots let you quickly observe how impedance varies over frequency for the reflection coefficients in the S-parameter set.



Smith Chart impedance (Plot Z button)

A Smith Chart provides impedance, phase, and magnitude information, including impedance in polar or rectangular format, markers readout, start and stop frequencies, and overlay:



Time Domain plots (Plot TD button), including impulse response vs. time and step response vs. time
In the time domain, all the plots should be settled by the end of the time record. If not, this could mean that the measurement was performed with too short of a time interval. Note that if a pulse is close to the beginning of the record, such as a typical **t11**, then it should be expected that a portion of the non-causal response at the front is wrapped to the very end of the record. SDLA handles the wrapped-end part internally.

This non-causal part is not real for the analog circuit that was measured; it is an artifact of the band-limiting effect of performing the IFFT operation to transform the S-parameter set to the time domain.

If the frequency spacing is too wide, the time domain will show aliasing where the pulse is wrapped in the time domain to an aliased position.



Pressing Overlay on the left side lets you select up to 16 plots in one visual display. The overlay plot is performed on four axes: Magnitude vs. Frequency, Phase vs. Frequency, Impulse vs. Time and Step Response vs. Time:



The **Overlay Menu** is on the left side, with a **Plot** button at the bottom. Pressing **Plot** opens a window with four overlaid axes: dB vs. GHz, Phase in Degrees vs. GHz, Amplitude vs. Time, and Step Response vs. Time.



Test point filter (transfer function) plots. Plots of test point filter responses allow for verification of the system setup. Any problems typically show up in the plots. Magnitude, Phase, Impulse and Step graphs are available.



**Tx Emphasis plots.** For details, see *Tx Emphasis Menu*.

Additional plots are available via:

- DPOJET eye diagram plots. When the Main Menu Config button is set for Auto Configure, then when the Apply button is pressed, DPOJET will produce eye diagram plots for the test point waveforms that are turned on. These will include the source waveforms and waveforms from one or more of the test points that are enabled. DPOJET can create a maximum of four plots. Therefore, it may be necessary to go into the DPOJET menus to reassign the plots to view the waveforms desired. (To do this, from the scope menu, press Analyze>Jitter and Eye Analysis (DPOJET).)
- Scope waveform plots. The waveforms that represent the test points appear on the oscilloscope screen. They may be controlled for viewing by using the standard oscilloscope controls. Cursor measurements and standard measurements may also be applied.

# **SEE ALSO:**

- Using Plots for Troubleshooting S-parameters
- Understanding Test Points

# Using plots for troubleshooting sparameters

SDLA Visualizer S-parameter plots include an Overlay tool, located on the left side, that allows you to view any selected plots on one display. This can be helpful in many scenarios, such as:

Viewing a DUT with mismatched differential pairs

# Viewing a DUT with Mismatched Differential Pairs

In this example, impulse plots reveal that the delay between transmission line pairs are not matched in length. In the overlay plot below, the two lines have similar magnitude responses. However, the difference in delay between the two lines shows up in the phase, impulse, and step response plots.



Troubleshooting bad VNA measurements using overlay plots

# **Troubleshooting Bad VNA Measurements Using Overlay Plots**

In this example, there is an open connection on one leg of the fixture and cables when measuring the S-parameters on a VNA. (Also note the bad S11 and S22.) The other leg of fixture looks okay. All four plots circled on the image below should be similar-looking for a good fixture.



Troubleshooting bad phase response

# **Troubleshooting Bad Phase Response**

In this example, the wrong calibration kit on VNA was used, which resulted in a bad phase response. The phase did not start at zero degrees at DC. This, in turn, resulted in bad step response.



Verifying mixed mode vs. single-ended mode

# Verifying Mixed Mode vs. Single-ended Mode

You can use these plots to verify the difference between mixed mode and singleended.



Troubleshooting bad step response

### **Troubleshooting Bad Step Response**

In this case, the step response has a non-casual dip before the rising edge. The cause of this is revealed by observing the magnitude response zoomed in at DC. The S-parameters commonly do not contain DC, because VNAs cannot measure it. SDLA must then extrapolate the data to DC before it can be transformed to the time domain for processing to make a transfer function filter.

Extrapolation can be problematic, as can be seen below, where the first data sample in the S-parameter set was lower than the previous one due to noise or a bad VNA measurement. This results in the extrapolated portion of the curve being slightly offset by a few tenths of a dB. This is enough to cause the preshoot dip in the step response, as shown below.



# **SEE ALSO:**

- Plots
- Examples of Tasks and Troubleshooting

# Tx block (Transmitter modeling block)

**Tx block overview** The Transmitter Modeling Block is the only block that contains a double definition: it is defined once for the Measurement Circuit model and again for the Simulation Circuit model.

A simple model assumes a perfect 50 Ohm environment. But what if the transmitter environment is not 50 Ohm? Using the **Tx Block**, you can model different configurations, such as setting a nominal impedance. You can represent the transmitter as one 2 port S-parameter model, or as two 1-port models. You can also use it to configure **Emphasis** options.

The source impedance is specified by the user, while the Thevenin equivalent voltage is computed by the software, based on the input waveforms and system models defined by the user.

**Thevenin Equivalent voltage:** The Tx Block contains the Thevinin equivalent voltage, which has special importance to the system. It is the point that contains the common waveform voltage that is shared by BOTH the Measurement Circuit (de-embed) path and the Simulation Circuit (embed) path. In other words, this is the point where the acquired waveform is passed into the simulation side of the system. Since a voltage source has zero impedance, this point provides isolation that prevents the Simulation Circuit from loading the Measurement Circuit configuration.

Pressing on the Tx block brings up the *Tx Configuration Menu*. From there, you can select the **Emphasis** radio button and press on the **Emphasis** button that appears on the diagram. This brings up the *Tx Emphasis Menu*.

# **SEE ALSO:**

- Tx Configuration Menu
- Tx Emphasis Menu.

# **Tx configuration menu** Use this menu to model different transmitter configurations. You can reach it by pressing **Tx** on the Main Menu.

The transmitter representation is split into two Thevenin equivalent circuit models with a common differential voltage source. The top section is the side for the Measurement Circuit (de-embed) path and the bottom is for the Simulation Circuit (embed) path. For example, the graphic below shows a transmitter model using nominal impedance:

4	SDLA Visualizer - Tx Configuration						
	Setup Thevenin Equivalent model f	or de-embed and embed paths					
		°s	NA O Tp11	NA Tp12	50 b1 ohms 50 b2 50 o1 ohms 50 o2	Impedance Nominal v Impedance Nominal v	Piot

The following image represents the transmitter as one 2-port S-parameter model in the lower (embed) part of the systems, and two 1-port S-parameter blocks in the upper (de-embed) part of the system. These are shown differently to illustrate the choices; normal usage is for both the upper and lower sections to be set up as identical to each other.

*	SDLA Visualizer - Tx Configuration				
	Setup Thevenin Equivalent model for de-embed and	embed paths			
	e e	S Filename Filename Filename NA Tp11	Browse b1 Browse b2 NA Port 1 c1 Browse Tp12 Port 2 c2	Impedance s1p Files v Impedance s2p File v	Plot
	Cinpidsis				

Select the **Emphasis** radio button. Press the **Emphasis** button that appears in the diagram, which brings up the *Tx Emphasis Menu* :

📣 SDLA Vis	ualizer - Tx Configuration						
Setup The	evenin Equivalent model fo	r de-embed and embed pa	ths				
			ŧ		50 b1 ohms 50 b2	Impedance Nominal 💌	
	Emphasis		NA FIR Emphasis Tp11	NA O Tp12	50 c1 ohms 50 c2	Impedance Nominal 💌	Plot

# **SEE ALSO:**

- Tx Block Overview
- Tx Emphasis Menu.

# **Tx emphasis menu**The Tx Emphasis Menu lets you specify, remove or add emphasis, de-emphasis<br/>or pre-emphasis filters. You may also read data from a FIR filter file. You reach<br/>this menu by pressing **Tx** on the Main Menu, selecting the **Emphasis** radio<br/>button, and then pressing the **Emphasis** button that appears on the diagram. Note<br/>that the Emphasis feature only appears in Simulation Circuit path.

SDLA Visualizer - Tx Equalization C	ionfiguration		
Create Emphasis filter or load f	from file		
	Emphasis	Specification	
	🔘 Thru	dB	
	O De-	Add 5	Plot
	O Pre-	Bit Rate Gb/s	
	3-tap FFE		
	Read From FIR File		ок

Four types of filter response are available. Each option offers the ability to either remove the effects of a component or to simulate one. The Pre/De-emphasis units are in dB. You can use the typical 3 dB setting or enter a custom dB setting. To see the results of the filter on the source signal, press **Apply** on the Main Menu to recompute the system's test point filters.

- Thru removes the effect of de-emphasis added by another circuit block or device.
- De- adds de-emphasis: it attenuates the low frequency components to compensate for high-frequency loss through the Channel. Shown in the image above.
- Pre- adds pre-emphasis: it amplifies the high frequency components to compensate for high-frequency loss through the Channel.
- 3-tap FFE: High speed serial data standards require 3-tap FFE at the transmitter to compensate for the channel loss. When 3-tap FFE is selected, SDLA shows three numerical controls for the three taps c-1, c0, and c1. You can enter in values for the three taps in linear scale. When you Apply, SDLA computes the filter for the 3-tap FFE Tx equalizer. Make sure the DC term c0 is bigger than both c-1 and c+1. Otherwise the equalizer will not be stabilizing.

SDLA Visualizer - Tx Equalization Configuration						
Create Emphasis filter or load fr	Create Emphasis filter or load from file					
	Emphasis	Specifi	ication			
	○ Thru ○ De-	Add	c-1         c 0         c+1           0.1         0.775         0.125			
	© Pre-	🔘 Remove	Bit Rate Gb/s	Plot		
	<u>B-tap FFE</u> Read From FIR File			ок		

• **Read from FIR File**: The Emphasis block may be set up from a FIR filter file as follows:

Select the **Read from FIR file** radio button. Browse to the location of your filter file. The Emphasis FIR filter is combined into the test point transfer function using the current sample rate setting of the oscilloscope. The file format contains comment lines starting with #. It then contains at least one line formatted as: <sample rate>; coef1, coef2, ... coefN.

SDLA Visualizer - Tx Equalization Configuration	SDLA Visualizer - Tx Equalization Configuration				
Create Emphasis filter or load from file	Create Emphasis filter or load from file				
	Emphasis				
	🕐 Thru				
	🔘 De-	Plot			
	O Pre-				
	🔘 3-tap FFE				
	Read From FIR File     Browse	ок			
	NULL				

**NOTE.** The filter setup need not be an emphasis type. It may be of any type required to better simulate your system.

# **Specification Settings**

Use the Specification settings that appear when you select the De- or Pre- radio button to add or remove emphasis values. The bit rate is the bit rate of the source signal. It determines the frequency range increase or decrease in the magnitude response of the emphasis filter. The magnitude frequency response is periodic, with a period determined by the bit rate. The peak value of the filter magnitude response is set by the dB value you chose.

# Plot button

Pressing **Plot** opens a window that contains four graphs: Magnitude vs. Frequency, Phase vs. Frequency, Impulse Response vs. Time, and Step Response vs. Time. The image below shows a plot where de-emphasis with a setting of 3 dB was added:



Press below for more examples of plots.





Plot with 3 dB of de-emphasis removed.







# **SEE ALSO:**

- Tx Block Overview
- Tx Configuration Tool.

# Embed block

**Embed block overview** The Embed Block represents a cascade of 4-port S-parameter models for simulating a desired system to be connected to the Tx model. Models can be loaded from S-parameter files as 4-port, 3-port, 2-port, 1-port, or transfer functions. Models can be created from RLC combinations, or a lossless transmission line. Probe load models are also included.

In a typical usage case, the testing of some serial standards requires the embedding of a compliance channel. But probing at the Rx pins is often not possible, requiring simulation of the channel. The Embed Block lets you "insert" a simulated channel so that you can observe the closed eye. Then, you can use the Rx Block to open the eye.



Create a simulated channel to embed by pressing **Embed** on the Main Menu. This brings up the Embed Menu. Using this, you may configure the blocks in many ways. Press here for some possible configurations. For more information, see *De-embed/Embed Menu*.

#### Embed Block – Some Possible Configuration

- 4-port single-ended S-parameter file
- 4–port differential S-parameter file
- Two 2-port S-parameter files
- FIR filter files (time domain)
- Transfer function files (frequency domain)
- Various RLC series or parallel configurations
- Lossless Transmission line model

- 3-port probe load model file
- 1-port load S-parameter file
- 2-port load S-parameter file
- Nominal load impedance
- TDT waveform block

#### **SEE ALSO:**

De-embed/Embed Menu

# Rx block (Receiver modeling block)

**Rx block overview** Configure the Rx Block by pressing **Rx** on the Main Menu. This brings up the *Rx Configuration Menu*.

The Rx Block represents the model for the serial data link receiver for the Simulation Circuit. It restores the integrity of the data stream and recovers the embedded clock. It can serve as a "reference receiver" in that it performs at the minimal acceptable level, as defined by a standard, for a serial data receiver. (The analog part of a receiver, including the package and termination model, can be modeled using the *Embed Block*, which contains the S-parameter files, transmission line, and RLC circuit models.)

The Rx Block contains:

- Mode choice of User, AMI or Thru
- CTLE equalizer (User mode)
- Clock recovery (User mode)
- FFE / DFE equalizer (User mode)
- Output of CTLE to oscilloscope Math channel (User mode)
- IBIS-AMI model of equalizer (AMI mode)

The Rx Block equalizer compensates for the loss, cross-talk, reflection or noise in the link. It attempts to optimize the signal/noise ratio of the eye opening, among other targets, in order to improve the link performance in aspects such as bit error rate. In the oscilloscope measurement context, this equalization allows you to accurately simulate the signal timing and amplitude parameters at the receiver. Sometimes, this measurement point may be referred as a "virtual Rx", reflecting the signals.

The receiver has a comparator, or "slicer", that determines whether a bit with value 0 or 1 is received in any unit interval. The exact timing at the slicer is determined by the clock recovery in the receiver. But the virtual Rx is typically not directly accessible by probing or other methods, so the Rx equalization must be simulated in order to get adequate measurements. Often the signal at the input of the equalizer will have a "closed" eye. When correctly designed, the equalization will "open" the eye and increase the eye height, width, or both.

Use the Rx Block AFTER using the Embed Block to create and insert a channel, so you can observe the eye closure at the Rx load. The Rx Block will then show what the signal will look like inside the Rx where the decision 0 or 1 is made by the comparator (aka the "slicer") after the Rx equalization.

The Rx Block provides three equalizer modes: User, AMI, and Thru. In User mode, continuous-time linear equalizer (CTLE), feed-forward equalizer (FFE), and decision feedback equalizer (DFE)) models are provided for you to try as serial data receivers typically contain them. Also, user mode supports various equalization adaptation/optimization requirements, such as the LMS-based optimization criteria from SAS 6G, and peak-to-peak based optimization criteria from PCI Express 3.0, as these standards call for "behavioral equalizers" that can be modeled using CTLE and/or DFE. Note that SDLA not only provides implementation of such behavioral equalizers for various standards, but it goes beyond them to simulate much more capable signal conditioning, and allows you to compare them.

AMI Mode is also available; this allows you to emulate IBIS-AMI models, which are descriptions of the equalizers provided by chip designers and manufacturers, as well as EDA tools that provide similar plug-in functionality. This results in more precise simulated Rx waveforms for measurements, comparison and validation.



# **SEE ALSO:**

Rx Configuration Menu

# **Rx** configuration menu

Press Rx on the Main Menu to bring up the Rx Configuration Menu offers. This offers you three modes in the radio buttons on the left: User, AMI, and Thru.

#### **User Mode**



User Mode gives you access to equalization tools and options for recovering the data stream and clock by correcting for the effects of insertion loss, cross talk, reflection, and noise. Many **CTLE**, **clock recovery** and **FFE/DFE** parameters can be specified, plus Taps definition and training sequence detection. This mode also implements the behavioral equalizers of PCI Express, Gen3, SAS 6G, USB3, etc.

# **Config Tab**

On the Config Tab, the equalization processing runs from left to right. The CTLE and FFE/DFE equalizers may be enabled separately. When both sets of equalizers are enabled, the CTLE equalization occurs first, followed by the FFE/DFE equalization.

The following sections provide usage details:

- Using CTLE to Improve Signal Recovery
- Using PCIE Option in CTLE
- Using Clock Recovery for FFE/DFE Equalization
- Adjusting FFE/DFE to Improve Signal Recovery
- Using PCIE Option in FFE/DFE
- Using the Taps Tab
- Running the Rx Equalizer in User Mode

# AMI Mode

Load the .am	ni and .dll for the AMI model. Click Results buttor	to view the AMI output		-
User AM Thru Config Mig Log	Bit Rate Gols 6 Samples Per Bit 16	filename ani Browse all Browse Filename	Data  Remove ignore_Bits  Block Processing Size (K Samples)  Clock	Run Eq AM Output Results Pre-AM OK

AMI Mode allows you to emulate IBIS-AMI models, which are descriptions of the equalizers provided by chip designers and manufacturers, as well as EDA tools that provide similar plug-in functionality. Note that SDLA only emulates the digital part of the IBIS-AMI model in the Rx block. The analog part of the model is neglected in the Rx block, but can be modeled using the **Embed block**, where the S parameter file, T-line model, and RLC circuit models can be used to model the Rx package and terminations.

For more details, see Rx AMI Mode

#### Thru Mode

When using Thru mode, the output of EQ **Tp4**, is the same as the input to EQ **Tp3**. Note that selecting **Thru** on the Rx Configuration Menu does not alter the panels displayed.

# **SEE ALSO:**

- Rx Block Overview
- Rx AMI Mode

# Using CTLE to improve signal recovery

To use the CTLE equalizer, press **Rx** on the Main Menu to open the Rx Configuration Menu. Select **User**. On the **Config tab**, the **Equalizer: CTLE** panel is on the left.



Turn CTLE on or off using the radio buttons on the left. Four CTLE types are available on the drop-down menu:

- Standard
- FIR
- IIR
- PCIE3

The CTLE output waveform is Tp10. Pressing **Tp10** in the CTLE panel brings up the Test Point and Bandwidth Manager, where Tp10 can be assigned to a Math channel like other test points.

You may need to adjust the CTLE settings in order to recover the data and clock signals. You can plot the CTLE by pressing the **Plot** button on the right. It shows both the frequency domain and time domain responses of CTLE.

# Second-order CTLE

Many standards such as PCIE Gen3 and USB 3.0 define a second-order CTLE. The function of most of the key parameters of the second-order CTLE described here are shown in the following illustration. Refer to it as you review the parameter descriptions in the list that follows.



Most of the following parameters are defined in serial data standards.

$$H(s) = \frac{A_{dc}\omega_{p1}\omega_{p2}}{\omega_{z}} \cdot \frac{s + \omega_{z}}{(s + \omega_{p1})(s + \omega_{p2})}$$
(CTLE - 1)

where  $\omega_z = 2\pi f_z$ ,  $\omega_{p1} = 2\pi f_{p1}$ ,  $\omega_{p2} = 2\pi f_{p2}$  and

# $\mathbf{A}_{\mathsf{dc}}$

This is the DC gain of the CTLE transfer function. It is a positive number in linear scale. The default value is 0.8.

# $\mathbf{F}_{\mathbf{z}}$

This is the zero frequency of the CTLE transfer function. The value must be within the range of 1 MHz to 20 GHz. The default value is 750 MHz.

# F<sub>p1</sub>

This is the frequency of the first pole of the CTLE transfer function. The value must be within the range of 1 MHz to 20 GHz. The default value is 3.75 GHz.

# F<sub>p2</sub>

This is the frequency of the second pole of the second order CTLE transfer function. The value must be within the range of 1 MHz to 20 GHz. The default value is 5 GHz.

Another convention on the second order CTLE is used in standards such as SuperSpeed Gen 2 reference CTLE uses the following convention:

$$H(s) = A_{ac} \omega_{p2} \frac{s + \frac{A_{dc}}{A_{ac}} \omega_{p1}}{(s + \omega_{p1})(s + \omega_{p2})}$$
(CTLE - 2)

where  $A_{ac}$  is the high frequency peak gain

A<sub>dc</sub> is the DC gain

 $\omega_{p1} = 2\pi f_{p1}$  is the first pole frequency

 $\omega_{p2} = 2\pi f_{p2}$  is the second pole frequency

If the  $\omega_z$  in (CTLE – 1) is set to be

$$\omega_z = \frac{A_{dc}}{A_{ac}} \omega_{p1}$$

then formula (CTLE - 1) and (CTLE -2) are identical with

$$A_{ac} = \frac{A_{dc}\omega_{p1}}{\omega_z}$$

**NOTE.** Both formulas (CTLE -1 and CTLE - 2) have four independent variables to configure.

#### IIR

This option, found in the **CTLE Type** drop-down menu, loads a custom IIR filter file that sets the CTLE parameters. The IIR filter file is an ASCII text file with file extension .tsf. The file uses a polynomial transfer function to define the IIR filter. There is no limit on the order of polynomials. The file uses # to indicate a comment line; **Numerator** as the key word for the numerator polynomial, and **Denominator** as the key word for the denominator polynomial. For example, if an IIR is a first order filter having the pole at 4 GHz, then the denominator should be written as 1, 2\*pi\*4\*1e6.

The following is an example IIR filter file definition:

```
# IIR CTLE Filter
# defined by a polynomial transfer function
#
#
        b1s^{(n-1)}+b2s^{(n-2)}+...+bn
#
        a1s^{(m-1)}+a2s^{(m-2)}+...+am
#
#
# using the following format
#
#[Numerator]
#b1, b2, ..., bn
#[Denominator]
#a1, a2, ..., am
#
# Note that unit is radian/second, not Hz
[Numerator]
5.026548245743669e+010, 3.158273408348595e+020
[Denominator]
1, 6.283185307179587e+010, 6.316546816697189e+020
```

**NOTE.** *The frequency unit is radians/second, not Hz.* 

SDLA generates FIR filters (based on the IIR filter definition and complete signal path ) on pressing the **Apply** button in the Main Menu or the **Run Eq** button in the Rx Configuration menu. If CTLE output Tp10 is assigned to a Math waveform and turned on, then SDLA writes the **Tp10** FIR filter file (sdlatp10.flt) to C:\TekApplications\SDLA\output filters and configures the Math setup to utilize that filter.

# FIR

This button opens a file browser to load a custom FIR filter to set the CTLE parameters.

# PCIE3

When PCIE Gen3 option is selected (using the **CTLE Type** drop-down menu), SDLA Visualizer runs an optimization process to find the best CTLE setting to maximize the eye area, as per the PCIE Gen3 specification. For details, see *Using the PCIE Option in CTLE*.

# **SEE ALSO:**

- Using Clock Recovery for FFE/DFE Equalization
- Adjusting FFE/DFE to Improve Signal Recovery
- Rx Block Overview

# Using the PCIE3 option in CTLE

To use the PCIE3 option with CTLE, press **Rx** on the Main Menu to open the Rx Configuration Menu. Select **User**. On the Config tab, in the Equalizer: CTLE panel, choose **PCIE3** from the **CTLE Type** drop-down menu.

When this option is selected, SDLA runs an optimization process to find the best CTLE setting to maximize the eye area as per the PCIE Gen3 specification. The PCIE Gen3 specification defines 7 CTLE presets. The DC gains are -6, -7, -8, -9, -10, -11, and -12 in dB. When **PCIE3** is checked, the UI changes as shown in the following image. Press here to see settings for the following image.

🚺 SDLA Visuali	SDLA Visualizer - Rx Configuration					
Single run	n completed.					
User     AMI     Thru     Config     Taps     TrainSeq     Error Log	● On Equalizer: CTLE $r_{p10}$ ○ Off CTLE Type PCE3 $r$ A <sub>DC</sub> $f_2$ $f_{p1}$ $f_{p2}$ 0.8 A <sub>DC</sub> 3.75 $f_{p1}$ GHz 0.75 $f_2$ GHz 5 $f_{p2}$ GHz	Clock Recovery Bit Rate: Auto Detect Nominal 8 Gb/s PLL Type: I 2 10 JTF BVV MHz 0.7 PLL Damp 0 Clk Delay ps	On     Equalizer: FFE / DFE     Off     FFE/DFE Type     Adapt Taps     PCE3     Auto     O     FFE Taps     1     DFE Taps     1     Sample/bit     1     Amplitude     1     Ref Tap     O     Threshold     Use TrainSeq     Autoset V	Run Eq PCIE Output Results CTLE Plot Pre-DFE OK		

- PCIE3 is selected in the CTLE panel CTLE Type drop-down menu.
- Note that Adc, fz (GHz), fp1 (GHz), fp2 (GHz) are grayed out, as they are set by the SDLA optimization routine.
- fp1 = 2 GHz, fp2 = 8 GHz, and fz = fp1\*Adc. The value of Adc and fz are updated after running the CTLE optimization.
- Clock Recovery sets the Nominal bit rate to 8 Gb/s, PLL Type to 1, PLL BW MHz to 10.
- PCIE3 is selected in the FFE/DFE Panel FFE/DFE Type drop-down menu. FFE/DFE can be turned On or Off. When FFE/DFE is turned off, it means that only CTLE is active. When FFE/DFE is turned on, it means the Rx Equalizer has CTLE plus one-tap DFE selected. (See PCIE Gen3 Option: DFE.)
- Auto is selected on the Adapt Taps drop-down menu.

#### Sequence of operations when CTLE Type is set to PCIE3

When PCIE3 is selected in the **CTLE Type** drop-down menu, use this sequence of operations.

- 1. SDLA generates an optimal setting after the **Apply** button is pressed in the Main Menu or the **Run Eq** button is pressed in the Rx Configuration menu.
- 2. SDLA updates the settings of Adc, fz (GHz), fp1 (GHz), fp2 (GHz).
- **3.** SDLA creates the file pcieAdaptationEQ.txt at C:\TekApplications\SDLA \taps.
- 4. When DFE is off, the DFE Tap value should be 0. When DFE is on, the DFE Tap value should be between -30 mV and 30 mV.

#### PCIE Output Results

Press **Results** in the far right panel to view the contents of the optimization results file pcieAdaptationEQ.txt. All 7 pre-settings are listed. The best CTLE setting is labeled with \*\*\*, where the best setting has the maximum Eye Area value. See file example.

📃 pcieAdaptatio	onEQ - Notepad					×
File Edit Form	iat View Help					
# PCIE equa # Time: 04	lizer adapta -Nov-2012 19	tion results :04:35				Â
# CTLE(dB), -6 -7 -8*** -9 -10 -11	CTLE(lin), 0.501 0.447 0.398 0.355 0.316 0.282	DFE Tap(mV), 30.00 29.96 26.87 25.51 22.06 18.73	Eye Area(UI*mv), 44.20 51.84 52.73 43.34 35.65 29.44	Eye Height(mV), 84.13 91.02 88.26 81.33 75.01 69.44	Eye Width(UI) 0.53 0.57 0.60 0.53 0.48 0.42	- III
×					+	н

# **SEE ALSO:**

- Using CTLE to Improve Signal Recovery
- Adjusting FFE/DFE to Improve Signal Recovery
- Rx Configuration Menu
- Rx Block Overview

# Using clock recovery for FFE-DFE equalization

To use the clock recovery function, press  $\mathbf{Rx}$  on the Main Menu to open the  $\mathbf{Rx}$  Configuration Menu. Select User. On the **Config** tab, the Clock Recovery panel is in the middle.



Clock recovery is used for FFE/DFE equalization in the Rx Block. The software performs clock recovery by emulating a phase locked loop (PLL) circuit. Use the data rate defined for the serial standard you are testing. If you are testing a new serial line, you may need to measure the bit rate near the transmitter, or after some equalization (i.e., Tx emphasis equalization, Rx equalizations such as CTLE) so that the eye is opened for bit rate measurement. You can also use the **Auto Detect** option in the Clock Recovery panel to identify the bit rate.

Scroll down for information on Troubleshooting Clock Recovery.

# **Nominal Bit Rate**

The nominal bit rate of the signal. It is typically specified by a serial data standard if the signal is generated from a device designed based on standards. If it may vary, then the **Auto Detect** option could be helpful.

**NOTE.** The nominal bit rate you enter must be accurate, or you'll need to press the radio button to recover the data and clock signals.

# Auto Detect

When this radio button is selected, SDLA searches in the neighborhood of the nominal bit rate to detect the correct bit rate of the signal. After using the detect bit rate to open the eye, you can further tweak the nominal bit rate based on the detected bit rate to tune the clock recovery settings.

# PLL Type

The software supports Type I and Type II PLL clock recovery. Each serial standard specifies the type of PLL to use for clock recovery.

### PLL BW

The loop bandwidth of the PLL is defined as the -3 dB frequency of the error transform function of the PLL. The value should be specified in the serial standard.

# PLL Damp

This is the damping ratio of the Type II PLL. The value should be specified in the serial standard.

#### Clk Delay (ps)

The clock delay is a specific delay added to the recovered clock after the PLL result. The value adjusts the clock offset to optimize the equalization result and achieve the best data recovery.

# **Troubleshooting Clock Recovery**

If clock recovery fails, your bit rate might not be what you expect. One solution is to select **Auto Detect** in the Clock Recovery panel, as described above. Another solution is to measure the bit rate as near to the transmitter as possible. You can use the DPOJET application running on the oscilloscope to accurately measure the bit rate.

#### **Training Sequence Functions**

Another technique is to use the Training Sequence functions to help the Rx Equalizer identify the correct bit sequence before again running your test signal through the Rx Equalizer. Press here for more information.

The image below shows the TrainSeq tab of the Rx Configuration menu:

🚺 SDLA Visualizer - Rx Configur	SDLA Visualizer - Rx Configuration					
Training bit sequence dete	ection completed					
<ul> <li>User</li> <li>AMI</li> </ul>	Bit Sequence	Training Sequence	Sequence File			
🔿 Thru	1: 0 2: 1 3: 1	127 Pattern Length	Browse Save			
Config	4: 1 5: 0 6: 0	Detect				
TrainSeq	7: 0 8: 1 9: 0					
Error Log	10: 1 11: 1			ок		

- 1. Use a signal source with the same data pattern as the signal you plan to test, but with a clean, open eye pattern. This signal could be one acquired close to the transmitter, or a slower speed version of the original signal, or the original signal compensated using Tx emphasis or Rx CTLE to improve the eye opening.
- 2. On the Rx Configuration menu, select the **TrainSeq** tab. Set the correct **Pattern Length** according to the standard; for example, 127 for a PRBS7 data pattern.
- **3.** Press the **Detect** button. You should see a Bit Sequence displayed in the left field, which should be the same bit sequence as in your original signal.
- 4. With the correct bit sequence in place, select the Config tab and select the original test source.
- 5. Select (enable) the Use TrainSeq box if not already enabled. Enter the correct bit rate if you changed it in a preceding step.
- 6. Press the **Run Eq** button.
- 7. Check the results on the oscilloscope display. You should see a recovered data signal, though it may not meet the standard specifications. You may need to address other design issues to correct any problems with the recovered data.

# **Checking Test Point Filters**

Another area for investigation is whether your test point filters are correct. Review the plots for those filters to determine whether high-frequency noise or other aberrations are corrupting the signal. Use the *global bandwidth filters* to reduce such noise.

# **SEE ALSO:**

- Using CTLE
- Adjusting FFE/DFE to Improve Signal Recovery
- Rx Configuration Menu
- Rx Block Overview

# Adjusting FFE-DFE to improve signal recovery

To use the FFE/DFE equalizers, press **Rx** on the Main Menu to open the Rx Configuration Menu. Select **User**. On the Config tab, the **Equalizer: FFE/DFE** panel is on the right.

You may adjust the Rx Equalizer settings in order to recover the data and clock signals by using many of the same techniques used to optimize a hardware receiver.

On	Equalizer: FFE / DFE				
Off	FFE/DFE Type	Adapt Taps			
	Custom 🔻	Auto 💌			
	0 FFE Taps	3 DFE Taps			
	1 Sample/bit	0.181 Amplitude			
	1 Ref Tap	0.000773 Threshold			
	📃 Use TrainSeq	🔽 Autoset V			

# **FFE/DFE Type**

The following selections are available in the drop-down menu:

- **Custom:** You may configure every parameter in the FFE/DFE panel.
- PCIE3: This is a special DFE option defined in the PCIE Gen3 specification. For details, see Using the PCIE Option in FFE/DFE.

# Adapt Taps

The following selections are available in the drop-down menu:

- Auto: The adaptation routine starts by identifying initial Tap settings and then adjusts them to optimize recovery of the data and clock.
- From Current: The adaptation routine uses current Taps values as the initial Taps settings, then adjusts them to optimize recovery of the data and clock. The initial Taps settings might be those saved from an earlier test.
- None: The Rx equalizer uses the current Taps either from your inputs or from a previous adaptive session. Use the entered values without changes. This option is useful when you want to load a known Taps file in the Taps tab to resume a test started earlier.

Most of the following parameters are defined in a serial data standard:

# FFE Taps

The Feed-Forward Equalizer tap number is normally set to a number defined by the serial data standard. A value of FFE Taps = 0 means the FFE has one Tap with Tap coefficients fixed to 1, signifying that FFE is off. The default value is 0.

# Sample/bit

Sample per bit specifies the number of FFE Taps per bit. If set to >1, it implies an FFE with fractional spaces. The default value is 1.

# Ref Tap

The Reference Tap for the FFE indicates the number of precursor Taps. It must be set to one (1) more than a multiple of the number of FFE Taps per bit. The default value is 1.

# **DFE Taps**

The DFE Taps number is normally set to a number defined by the serial data standard. For example, the setting for SAS is 3, and the setting for PCIE Gen3 is 1.

# Amplitude

The Amplitude is the target output amplitude for the Rx Equalizer. When you select Autoset Voltages (Autoset V checkbox), the adaptation routine adjusts this value automatically to optimize the recovery of the data signal. The default value is 0.15 V.

# Threshold

The Threshold is the middle voltage level of the signal, which may be the transition between logic levels. For biased signals, enter the mid-level value. For differential signals, the value should be close to 0 V. The default value is 0 V. Lacking clear knowledge of the correct voltage, use the Autoset Voltages function to determine the optimal value.

# Use TrainSeq

Enables the Rx Equalizer to optimize its adaptation routine over a specific pattern the length of which is defined on the TrainSeq tab.

# Autoset V

When Autoset Voltages is enabled, the Rx Equalizer adaptation routine adjusts the Amplitude and Threshold values to optimize recovery of the data and clock.

# **SEE ALSO:**

- Using CTLE to Improve Signal Recovery
- Using Clock Recovery for FFE/DFE Equalization
- Rx Configuration Menu
- Rx Block Overview

# Using the PCIE3 option in FFE-DFE

To use the PCIE3 option with FFE/DFE, press **Rx** on the Main Menu to open the Rx Configuration Menu. Select **User**. On the Config tab, in the **Equalizer: FFE/DFE** panel, choose **PCIE3** from the **FFE/DFE Type** drop-down menu.

The PCIE Gen3 specification defines a one-Tap DFE with no FFE. The DFE Tap value is limited to a value between -30 mV and 30 mV.

) On	Equalizer: FFE / DFE				
Off (	FFE/DFE Type	Adapt Taps			
	PCIE3 👻	Auto 👻			
	0 FFE Taps	1 DFE Taps			
	1 Sample/bit	1 Amplitude			
	1 Ref Tap	0.000787 Threshold			
	Use TrainSeq	🗸 Autoset V			

When **PCIE3** is selected in the drop-down menu, the following field values are grayed out and their values set by internal algorithms:

- FFE Taps = 0
- Sample/bit = 1
- Ref Tap = 1
- Use trainSeq = unchecked
- DFE Taps = 1
- Amplitude (V) = 1
- Autoset Voltages = checked
- Clock recovery is set per PCIE3 specification: nominal bit rate is set to 8 Gb/ s, PLL Type to 1, PLL BW MHz to 10.

These fields are enabled again when **Custom** is selected in the FFE/DFE Type drop-down menu.

# **SEE ALSO:**

- Adjusting FFE/DFE to Improve Signal Recovery
- Rx Configuration Menu
- Rx Block Overview

**Using the taps tab** The settings on the **Taps Tab** (on the *Rx Configuration Menu* with User mode selected) reflect the settings on the **Config Tab**. For example, in the following figure, the FFE Taps have a value of 1, and the DFE field shows 3 Taps with different values. This state results from settings on the Config tab, where FFE is set to 0 and DFE set to 3. If this was the result of setting Adapt Taps to **Auto**, you could save the results in a Tap file for use in a later Rx Equalizer run.

E/DFE			
FFE Taps	DFE Taps 1: 0.068 2: 0.023 3: 0.006	Taps File Browse Save	ок
	E/DFE FFE Taps 1: 1.000	EXDFE  FFE Taps  DFE Taps  1: 0.068  2: 0.023  3: 0.006	FFE Taps         DFE Taps         Taps File           1: 1.000         1: 0.068         1: 0.068           2: 0.023         3: 0.006         Browse

When **PCIE3** is selected in the FFE/DFE panel (in the drop-down menu under **FFE/DFE Type**), and **Auto** or **From Current** is selected under **Adapt Taps**, the DFE adaptation algorithms attempt to maximize the eye area. The resulting DFE tap value is shown in the Taps tab.



The Config Tab **Results** button is enabled on the right panel (under **PCIE Output**) when **PCIE3** is selected, and **Auto** or **From Current** is selected under **Adapt Taps**. After adaptation is finished, press **Results** to open the adaptation results file *pcieAdaptationEQ.txt*. This results file has better numerical resolution for DFE Tap(mv) than what is shown in the Taps tab. Note that the DFE tap value is between -30 mV and 30 mV per PCIE Gen3 specifications.

📃 pcieAdaptatio	onEQ - Notepad					×			
File Edit Format View Help									
# P⊂IE equa # Time: 04	lizer adapta -Nov-2012 19	tion results :04:35				Â			
# CTLE(dB), -6 -7 -8*** -9 -10 -11	CTLE(lin), 0.501 0.447 0.398 0.355 0.316 0.282	DFE Tap(mV), 30.00 29.96 26.87 25.51 22.06 18.73	Eye Area(UI*mV), 44.20 51.84 52.73 43.34 35.65 29.44	Eye Height(mV), 84.13 91.02 88.26 81.33 75.01 69.44	Eye Width(UI) 0.53 0.57 0.60 0.53 0.48 0.42				
1						-			
					4				

For a complete description of AMI files, visit the IBIS Open Forum at http:// www.eda.org/ibis. See especially the I/O Buffer Information Specification for IBIS 5.1, AMI Executable Model File Programming Guide (section 10) and AMI Parameter Definition File Structure (Section 10A) at http://eda.org/pub/ibis/ ver5.1/ver5\_1.pdf.

# **SEE ALSO:**

- Adjusting FFE/DFE to Improve Signal Recovery
- Rx Configuration Menu
- Rx Block Overview

# **Running the RX equalizer**

The following steps describe how to make a first run of the Rx Equalizer to determine whether further adjustments are necessary.



- 1. First, configure your input or inputs using the Main Menu.
- Next, you need to turn on Tp3 as it is required for the Rx Equalizer. On the Main Menu, press Tp3 and use the *Test Point and Bandwidth Manager* to turn it on (A-B mode if it is the Dual Input case).

- 3. Press **Rx** on the Main Menu. On the Rx Configuration Menu, select the User radio button. On the **Config Tab**, enter the FFE and DFE Taps and configure the PLL fields for a Receiver as defined in the standard you are testing. (Alternatively, you can load a setup file by pressing **Recall** on the Main Menu that sets SDLA as per the standard you are testing against.)
- 4. Set the bit rate in the Clock Recovery panel (if it has not already been set by a standards file). Select **Auto Detect** if the nominal bit rate may not be accurate.
- 5. Press the **Run Eq** button.
- 6. To view the output waveforms, go to the oscilloscope display. The Ref4 waveform is the Data signal and is labeled **Tp4 R4**. The Ref3 waveform is the recovered Clock waveform and is labeled **Clk R3**.

# **SEE ALSO:**

- Rx Configuration Menu
- Rx Block Overview

# AMI mode

Many chip and system vendors use the IBIS Algorithmic Modeling Interface (IBIS-AMI) standard for designing and simulating multi-gigabit serial links. These models capture the behavior of transmitters and receivers as well as channels of particular design more precisely than generic implementation (for example, User Mode equalizer with CTLE and FFE/DFE). It is especially useful to apply the receiver behavioral model to the waveform captured by the scope. It allows evaluating the signal at the slicer of the receiver using various measurements and comparing it to simulation results obtained with the same model.

IBIS-AMI model is typically implemented as a combination of an .ami and a .dll file (often referenced from an .ibs file). SDLA Visualizer supports both 32 and 64 bit windows dll binaries. The .ami contains parameter definitions, their expected range and default values. SDLA Visualizer will extract the necessary information and condense the file to be used as the input string to the model. Parameters can be changed either by editing the original .ami file or the condensed file (which is easier to handle). The definition of parameters is vendor-specific and additional documentation may be needed for parameter tuning.

Version 2.1 of SLDA supports both full and condensed .ami files. If a full .ami file is specified the file is automatically condensed. The condensing operation reduces the Model\_Specific section of the .ami file to essential fields, to a string that is passed to the .dll. In order to vary parameters the model uses when evaluating the Get\_Wave function, edit the condensed .ami file by changing the respective parameter values, or edit the default values in full .ami.

Since the Model\_Specific parameters are model-specific, consider comments or vendor documentation for additional information.

**NOTE.** For specific information on how to modify an IBIS (.ami) parameter file for use by SDLA Visualizer, please refer to the readmeAmi.txt file available in C: \Users\Public\Tektronix\TekApplications\SDLA\AMI



AMI mode is provided as an option for Rx modeling in SDLA Visualizer. It can be activated by choosing the AMI option on the Rx Configuration panel. Press **Browse** under .ami and load the ami file you want to use. Press on the lower **Browse** button to select the .dll file. For valid results the .ami and .dll files must be compatible.

**NOTE.** signal labels **Tp3:** *A***-***B***, Data** and **Clock** refer to I/O labels on the Main Menu Rx block.

**Samples per Bit** settings control the interpolation ratio. Most AMI simulators and models expect a certain number of samples per unit interval (bit) to work properly. Please consult specific AMI model documentation to make sure whether there are specific requirements for this parameter.

**Remove Ignore\_Bits** checkbox controls whether certain bits in the beginning of the waveform are ignored in the output waveform and measurements. The number of bits is specified as the Ignore\_Bits parameter in the Reserved\_Parameter section of the model (.ami file). Checking this box will eliminate the adaptation region of the waveform from the output. Sometimes, however, looking at the adaptation process may be of interest. Then the setting can be left unchecked.

**Block Processing Size (K samples)** specifies whether the block processing is used and the block size in K samples. Block processing is useful with long waveforms and complicated models. Block processing helps save memory. The use of block processing also may allow observing the AMI-processing in more detail. Some models implement saving of the adaptation record or log other data on each call to the Get\_Wave function.

**Pressing Run EQ** takes the data from Tp3, processes it together with the information from the .ami and .dll files, and then loads the new waveform with equalized data into Tp4 (Ref4) and the recovered clock information into Tp5 (Ref3), if configured. Pressing **Run Eq** in the Rx block allows you to run this particular block over again without rerunning the preceding chain of operations.

**NOTE.** For Run Eq to work properly, the Measurement and Simulation circuit modeling portion of SDLA (De-embed, Tx and Embed blocks) must run at least once after startup and each update (Press **Apply** on the main menu).

Pressing **Results** (under **AMI Output**) opens a .txt file (AMI\_out.txt), which is written to the folder where the .dll file is located. This file may contain output parameters from the AMI model such as the adaptation results on Taps (model-specific). Press the **Msg Log** tab to check the output message output by the AMI model.

Note that ami files may produce additional results like an adaptation record or debug information in other files. These are either stored in the location of the ami files or a specific location in the file system determined by the respective ami parameter.

**Pre-AMI** checkbox allows you to enable export of interpolated waveforms before AMI processing. The interpolation ratio is determined by the **Samples per Bit** and **Bit Rate** parameters (source waveform). The file can be found in *C:* \*Users\Public\Tektronix\TekApplications\SDLA\output filters* folder and it contains time voltage pairs. The ASCII format is the most universal but also has an effect on performance, so use it only when needed. This data can be used for comparison in EDA tools, MATLAB or other purposes.

For a complete description of .ami files, visit the IBIS Open Forum at http:// www.eda.org/ibis. See especially the I/O Buffer Information Specification for IBIS 5.1, AMI Executable Model File Programming Guide (section 10) and AMI Parameter Definition File Structure (Section 10A) at http://eda.org/pub/ibis/ ver5.1/ver5\_1.pdf.

# **SEE ALSO:**

- Rx Configuration Menu
- Rx Block Overview

# Configure actions for the apply and analyze buttons

You have some flexibility for determining what happens when you actually run the models using the **Apply** and **Analyze** buttons on the Main Menu, including how SDLA will work with DPOJET, and whether a new waveform is to be acquired or a previous one is to be used. When you press the **Config** button on the Main Menu, this menu comes up:



# Configuring the Apply button

Check **Automatically run Analyze when pressing Apply** to combine the actions of the Apply and Analyze buttons. This will use the current configuration settings of the Analyze button as described below.

# Configuring the Analyze button

There are three different options for configuring the Analyze button:

**Recalculate:** This recalculates the results in DPOJET without changing the DPOJET setup or clearing previous results. This can be used to process the measurements on multiple acquisitions. This option should not be used when new models are being applied, as it could lead to inconsistent results.

**Clear and Recalculate:** This clears the previous data results and automatically runs DPOJET without changing the DPOJET setup.

**Auto Configure:** This auto-configures DPOJET for all test points that are turned on, does TIE measurements and plots eye diagrams.

**NOTE.** If custom settings have been defined in DPOJET, Auto Configure should not be used, as it will re-configure DPOJET.

#### Waveform to use for Apply and Analyze

This option allows you to select whether a new waveform should be acquired each time the Apply or Analyze button is pressed. By default, a new waveform will always be acquired. If you desire to use the already acquired waveform, select **Use Current**.

**NOTE.** It is important to properly configure the data rate when using the DPOJET Analyze function, as the clock recovery configuration in DPOJET is based on the data rate.

The figure below shows the Analyze button configured to **Clear and Recalculate**.


#### **SEE ALSO:**

- Understanding the System
- Using DPOJET and SDLA Visualizer Together

# Creating filters for a sampling scope

The menu below shows the configuration for creating filters for a sampling scope.



In the input source pull down box of the main menu, select Sampling as the source waveform.

When Sampling is selected as the source, Extend to 32 bits is disabled. Deselecting Sampling enables Extend to 32 bits. This applies to all of the following cases.

#### Single input case

Select Sampling as the source in default mode, single input and Remove Delay remain the same. When the test point configuration menu should have Remove Delay on and the Adjust Delay disabled. The Export filters for 32 bit scope is also disabled as shown below:

🕗 SDLA Visualizer - Test Po	oint and Bandwic	dth Configu	ration						- • •
Tp2 - Map the desired t	test points to Ma	ath to see th	e processed waveform						
Tp On/Off	Map Tp to	o Math		Label	Filters	Global BW Limit	Delay		
◯ Math1	None	•		NA	Save	None	🔘 Keep Delay		
Math2	Tp1	•		Tp1	Save	Auto	Remove Delay		
◯ Math3	Tp2	•		Tp2	Save	Custom	Adjust Delay		
🔘 Math4	ТрЗ	•		Тр3	Save				
C Ref3	Tp5					BW: 5GHz			
O Ref4	Tp4		Scaling Factor						ок
			90 % ?		Export filters	s for 32-bit scope <sub>?</sub>		ЛL	

#### **Dual input case**

In Dual Input mode, Sampling is not displayed as the input source since it does not apply for Dual Input case.

When Sampling is selected as the source, Dual Input mode is disabled as shown in the main menu above.

The sample rate is always automatically updated with the bit rate in this case. When the user inputs a new bit rate anywhere in the SDLA, the sampling rate is changed to be  $100 \times bit$  rate.

All probe options are disabled when Sampling is the source. Probes are not applicable to a sampling scope.



# Running a test

## Running a test: recommended order

Welcome to SDLA Visualizer, which offers a powerful, flexible set of modeling tools for de-embedding, embedding and equalizing high speed serial signals. This section describes the recommended order for running a test. The general steps and primary menus are listed below, and include links to fuller usage details within each step. For conceptual explanations, see *SDLA Visualizer Product Overview*. For component details, see *Main Menu in Detail*.

**NOTE.** If you would like to download a .PDF file of the Online Help that has been translated into, visit www.tektronix.com and press on "Change Country" at the top. Then enter the search term "SDLA Visualizer".

- 1. First, **connect the fixture and oscilloscope to the DUT**, using a probe or direct connection. Connect the source signal to an oscilloscope input channel. Adjust the oscilloscope trigger, vertical, and horizontal settings to capture signals with good fidelity. Using the oscilloscope Autoset function can simplify this adjustment. Ensure that the sample rate is set to a value for which its DSP calibration filters are on. (See the scope vertical menu.)
- **2.** Verify that DPOJET is installed and that it runs correctly. You can leave DPOJET running.
- **3.** Bring up SDLA Visualizer by pressing Analyze on the TekScope menu and selecting the program. (Use the Alt Tab keys to switch between programs.)



- 4. Now, configure the elements of the Measurement Circuit (the upper portion of the diagram), which represents the equipment that needs to be deembedded or removed from the acquired signal, such as fixtures, cables, and probes. Start by defining the first element, the **De-embed Block**, where you can use a cascade of S-parameter blocks, an RLC Circuit, or a lossless transmission line to represent the model. Each element can be validated with a plotting function. The load of the circuit, in many cases the oscilloscope, is also defined here. Steps
- a. Press De-embed on the Main Menu. The De-embed/Embed Menu comes up.

🛃 SDLA	Jalizer - De-embed	
Casc Norm Conv Resa	Move $1$ <td< th=""><th>Save Recall OK</th></td<>	Save Recall OK

- **b.** On the **Cascade** tab, select a probe option if applicable (SMA or High Z).
- c. Press on the first block B1. The Block Configuration Menu will come up.

SDLA Visualizer - Block Configuration				
Select B1 model and the applicable files if required				
Thru File RLC T Line	Port1 💌	Port2 💌 Port4 💌	Label B1	ОК

**d.** Select either the **Thru**, **File**, **RLC**, or **T Line** tab to choose how you'd like to model the element.

If you are selecting the **File** tab, choose a model type from the **Model** drop down list. For example, select "4-Port Single-ended" when using a 4-port single-ended S-parameter model. Press on the **Browse** button to load the S-parameter model and re-assign the port assignments if necessary.

Optionally, press Plot to view the S-parameter plot of the selected model.

- e. Press OK to return to the Block Configuration Menu.
- f. Repeat this procedure for as many of blocks B2 B8 as your Measurement Circuit model needs. It is important to load each element in the correct order as SDLA Visualizer takes reflections and cross coupling terms into account. Unused blocks are simply treated as "thru".
- **g.** Press on the final block of the **Block Configuration Menu**, which will either be labeled **Scope**, **SMAProbe**, or **Load**. This will allow you to determine what will load the output ports of the measurement circuit. In many cases, this is the oscilloscope or an SMA probe connected to a scope, or an Rx input (or something else) when a High Z probe has been selected.

The impedance can be modeled as a nominal value, one 2-port S-parameter block, or two 1-port S-parameter blocks by choosing one of the options in the Impedance drop-down list. By default, SDLA assumes 50 Ohm impedance.

**h.** Optionally, you may plot the S-parameters that represent the model if the **Plot** option is available on the right.

For fuller details, see *De-embed/Embed Menu*.

- 5. Next, define the second element of the Measurement Circuit, the **Tx Block.** Use this to set up the Thevenin Equivalent model for the de-embed path and to model the transmitter output impedance. This should be modeled either nominally to represent the actual transmitter, with two 1-port S-parameter models, or with one 2-port S-parameter model. Steps
- a. First, configure the transmitter impedance by pressing on Tx on the Main Menu. (By default, SDLA assumes 50 Ohm impedance.) The Tx Configuration Menu will come up. The controls on the top are for the Measurement Circuit.



**b.** Next, choose one of the options in the upper **Impedance** drop-down list. Press **OK**.

- **c.** Optionally, you may plot the S-parameters representing the model at this point by pressing **Plot** on the right.
- d. The Thevenin equivalent voltage will be computed by SDLA.

For fuller details, see *Tx Configuration Menu*.

- 6. Now, if applicable, define the elements of the Simulation Circuit Model, which simulates elements that are not physically present. Start by using the **Tx Block once again.** Steps
- a. First, configure the transmitter impedance by pressing on Tx on the Main Menu. (By default, SDLA assumes 50 Ohm impedance.) The Tx Configuration Menu will come up (shown above). The controls on the bottom are for the Simulation Circuit.
- **b.** Next, choose one of the options in the lower **Impedance** drop-down list. The impedance can be modeled either as a nominal value, or as one 2-port S-parameter model, or two 1-port S-parameter models. Press **OK**.
- c. Optionally, you may add or remove Emphasis by pressing on the Emphasis radio button in the Tx Configuration Menu. An Emphasis button will appear in the circuit diagram. Press on it to bring up the Emphasis Menu. Press OK.

For fuller details, see *Tx Configuration Menu*.

- Next, define the second element of the Simulation Circuit Model, the Embed Block. Use this to define the elements that make up the simulation path. The Embed Block has similar functionality to the De-embed Block described above. Steps
- **a.** Press on **Embed** on the Main Menu. The De-embed/Embed Menu comes up. Select the **Cascade** tab.



- b. Press on the first block B1. The Block Configuration Menu will come up.
- c. Select either the **Thru**, **File**, **RLC**, or **T Line** tab to choose how you'd like to model the element.

If you are selecting the **File** tab, choose a model type from the **Model** drop down list. For example, select "4-Port Single-ended" when using a 4-port Single-ended S-Parameter model. Press on the **Browse** button to load the S-parameter model and re-assign the port assignments if necessary.

Optionally, press **Plot** to view the S-parameter plot of the selected model.

- d. Press OK to return to the Block Configuration Menu.
- e. Repeat this procedure for as many of blocks **B2 B8** as your Simulation Circuit model needs. It is important to load each element in the correct order as SDLA Visualizer takes reflections and cross coupling terms into account. Unused blocks are simply treated as "thru".
- **f.** Press on the final block of the **Block Configuration Menu**, which will be labeled **Rx Load**. This will allow you to determine what will load the output ports of the Simulation Circuit. In many cases, this would model a physical receiver.

The impedance can be modeled as a nominal value, one 2-port S-parameter block, or two 1-port S-parameter blocks by choosing one of the options in the Impedance drop-down list. By default, SDLA assumes 50 Ohm impedance.

**g.** Optionally, you may plot the S-parameters representing the model at this point, if the **Plot** option is available on the right.

For fuller details, see *De-embed/Embed Menu*.

- 8. Define the third element of the Simulation Circuit Model, the **Rx Block.** Use this to simulate the equalization inside the Rx block, allowing you to virtually observe the waveform at the receiver pins. CTLE, FFE/DFE and IBIS AMI models are available. Steps
- a. Press on Rx on the Main Menu. The Rx Configuration Menu comes up.



b. Choose the type of equalizer to model: User, IBIS AMI, or Thru (no EQ).

- c. If you select User, CTLE/FFE and DFE equalizers can be used. For CTLE, under CTLE Type, choose the method to define the CTLE (Standard, IIR, FIR, or PCIe 3.0). To view the output of the CTLE, select the **Tp10** radio button and assign Tp10 to an available math channel.
- **d.** To use DFE and FFE, Clock Recovery must be configured. After configuring the Clock Recovery, select the **On** radio button to turn on the FFE/DFE block. Under **FFE/DFE Type**, choose either Custom or PCIE3, and configure the FFE/DFE settings.
- e. If you have previously pressed Apply on the Main Menu, and the only changes to the model are in the Rx Block, you only need to press on the Run Eq button to apply the Equalizer. If other changes have been made, press OK in the Rx Block and then press Apply in the Main Menu.

For fuller details, see Rx Configuration Menu.

**9.** Now that the model has been defined, **configure the Test Points**. Simply press on any of the test points on the Main Menu to bring up the Test Point and Bandwidth Manager, where you can assign it to a Math channel and other options. SDLA has 12 test points, with up to four math and two reference waveforms visible on the scope graticule at one time. For details, see *Test Point and Bandwidth Manager*.

SDLA Visualizer - Test Poi	nt and Bandwidth Configuration					×
Tp1 - Map the desired te	st points to Math to see the processe	ed waveform				
Tp On/Off	Map Tp to Math	Label	Filters	Global BW Limit:	Delay	
C Math1	None	NA	Save	None	🔘 Keep Delay	
Math2	Tp1: Main 💌	Tp1	Save	Auto	Remove Delay	
Math3	Tp2: Main 🔻	Tp2	Save	Custom	Adjust Delay	
C Math4	Tp3: Main 💌	Tp3	Save			
C Ref3	Тр5					
C Ref4	Tp4					ОК
	2 Export filters for 32-bit scope					

10. After the model has been configured,

**a.** Press the **Config** button on the Main Menu. On the resulting menu, configure the actions you'd like to happen when the **Apply** and **Analyze** buttons are pushed. For details, see *Configure Actions for Apply and Analyze Buttons*.



- b. Press the Apply button on the Main Menu. Wait for the status bar at the bottom to show that processing is complete. Pressing Apply creates a transfer function for each enabled test point based on the measurement and simulation circuit models defined above. The math and reference waveforms associated for each enabled test point will be automatically updated on the oscilloscope. If the Rx Block was enabled above, the clock and data output of the block will be stored in Ref3 and Ref4.
  DPOJET will be run automatically you configured it to do so in the prior step.
- Once the model has been applied, you may plot the test points by pressing Plot on the Main Menu. You may view the Magnitude, Impulse Response, Step Response, and Phase Plots to ensure that they have the response you expect. For details, see *Plots*.
- 12. If the plots did not show you the response you expected, you may fine-tune the bandwidth, by pressing the Global BW limit button on the Main Menu. For details, see *Test Point and Bandwidth Manager*.
- **13.** Optionally, you may **save the test points** filters that represent the transfer function of each of the enabled test points. After applying the model, press on any test point to bring up the Test Point and Bandwidth Manager. Any enabled test point can be saved by pressing the **Save** button (not be confused with the Save button on the Main Menu, which saves the setup.) For fuller details, see *Saving Test Points*.

This completes the general procedure for running the SDLA Visualizer Software. Each block has many configuration parameters not covered in this procedure. Explore the details of each processing block to get the most out of the SDLA Visualizer Software.

## **SEE ALSO:**

- Examples of Tasks and Troubleshooting
- Main Menu in Detail
- SDLA Visualizer Product Overview

# **Examples and troubleshooting**

# Examples of tasks and troubleshooting

### Tasks

Here are some examples of the many tasks that SDLA Visualizer can perform: *De-embedding Cables Embedding a Serial Data Link Channel De-embedding a High Impedance Probe De-embedding Significant Reflections with Dual Input Waveforms Removing a DDR Reflection with a Single Input Waveform* 

#### **Troubleshooting using S-parameter plots**

SDLA's S-parameter plots can be helpful in many scenarios, including:

- Viewing a DUT with mismatched differential pairs
- Troubleshooting bad VNA measurements using overlay plots
- Troubleshooting bad phase response
- Verifying mixed mode vs. single-ended mode
- Troubleshooting bad step response

For details, see Using Plots for Troubleshooting S-parameters.

### SEE ALSO:

Running a Test: Recommended Order

# Example of de-embedding cables

This example provides step-by-step instructions on how to de-embed a pair of cables. In this example, the transmitter impedance is assumed to be 50 Ohms. The scope Rx load is also assumed to be 50 Ohms. Each cable is represented by 2-port S-parameter models (one for each leg). The goal is to view the signal from the transmitter with the cable de-embedded, and the transmitter driving a 50 Ohm load.

This example uses Single Input mode.

1. First, select the Single Input radio button on the Main Menu.



- **2.** The next step is to define the Measurement Circuit model. This is made up of the transmitter source impedance, cable, and scope.
  - a. Press on Tx on the Main Menu to bring up the *Tx Configuration Menu*. On the top row, select Nominal in the Impedance drop-down list, and enter 50 Ohms in the b1 and b2 text boxes. Press OK.



**NOTE.** The impedance value should model the actual impedance of the transmitter. This can be done using either actual S-parameters or nominal values. It is important that this value match the transmitter impedance for the most accurate de-embedding.

**b.** Press **De-embed** on the Main Menu to bring up the *De-embed Menu*. Select the **Cascade** Tab.

🛃 🛛 SDLA Visualize	er - De-embed		
Cascade Normalize Convert Resample	Move Tp6 Tp7 Probe None SMA TDT High Z High Z	$\begin{array}{c ccccccccccccccccccccccccccccccccccc$	Save Recall OK

c. To model the input impedance of the scope, press on the last block of the cascade, which will be labeled Scope. This brings up the *Load Configuration menu*, shown below. Under "Model", select Nominal. Enter 50 Ohm for the impedance on both ports. Press OK.

SDLA Visualizer – Load Block Configuration		
Select the load model and the applicable file	(s) if required	
Model Nominal •	ohm Port 1 v 50 ohm Port 2 v 50	Label Load

d. Press on block B8. This brings up the *Block Configuration Menu*, shown below. On the File tab, under "Model", select 2-Port. Press the upper Browse button and select "twoPortActive.s2p" from the drop-down menu. Do the same with the lower Browse button. Under Label, change "B8" to "Cable". Press Plot to see the S-parameter plot of both the cables. To check the port assignments and passivity of the S-parameter files, press Check. Press OK.

SDLA Visualizer	- Block Configuration and the applicable files if required		
Thru File RLC T Line	Model 2-Port 💌	twoPortActive.s2p Port1  Browse Port2 Port1 Port1 Frowse Port2 Frowse Frowse Port2 Frowse Frowse Port2 Frowse	Label Cable Check Passivity Plot OK

**NOTE.** If your cable model is represented as a 4-port S-parameter block, you may select 4-Port as the model type and load an .s4p file.

- **3.** The next step is to define the Simulation Circuit model. In this example, we will drive the transmitted signal into an ideal 50 Ohm load.
  - a. Press Tx on the Main Menu once again, which brings up the Tx Configuration Menu (shown in step 1a above). On the bottom row, select Nominal in the Impedance drop-down list, and enter 50 Ohms in the c1 and c2 text boxes. Press OK.

**NOTE.** *The impedance value should match the value in the de-embed path.* 

**b.** Press **Embed** on the Main Menu, which brings up the Embed Menu, with the **Cascade** Tab displayed. Press on the last block of the cascade, which will be labeled **RX Load**.



c. This brings up the *Load Configuration menu*, shown below. Under "Model", select Nominal. Enter **50** Ohms for the impedance on both ports. Press **OK**.

4	SDLA Visualizer – Load Block Configu	ration			
	Select the load model and the app	licable file(s) if required			
	Model Nominal	Port 1	ohm 50 ohm 50	Label Load	ОК

- **d.** Leave all the other blocks in the Embed cascade diagram set to Thru, since only the ideal load block will be used for this example. Press **OK**.
- 4. Once the Measurement Circuit and Simulation Circuit models have been defined, the desired test points can be enabled. (For more information on test points, see *Understanding Test Points.*) Since the goal is to observe the signal at the output of the transmitter driving an ideal 50 Ohm load, we want to enable Tp2.

**NOTE.** It may seem that Tp1 will provide the de-embedded signal, but Tp1 will show the signal at the output of the Tx with the loading of the Measurement Circuit. Since we want to completely remove the effects of the Measurement Circuit, is the correct test point.

- **a.** Press on **Tp2** on the Main Menu, which brings up the Test Point and Bandwidth Menu.
- b. Under Map Tp to Math, select Tp2: Main in the first drop-down list.
- **c.** Under **Tp On/Off**, select the radio button next to **Math1** Math1 will be mapped to Tp2, so that when the model is finally applied, the filter will be configured automatically in Math1.
- d. Press OK.

DLA Visualizer - Test Poi	nt and Bandwidth Co est points to Math to s	nfiguration					
To Op/Off	Man To to Matt		Label	Filters	Global BW Limit	Delay	
Math1	Tp2		Tp2	Save	None	🔘 Keep Delay	
Math2	Tp1 💌		Tp1	Save	Auto	Remove Delay	
Math3	Tp2 💌		Tp2	Save	🔘 Custom	Adjust Delay	
Math4	ТрЗ 💌		ТрЗ	Save			
Ref3	Tp5				BW: 25GHz		
C Ref4	Tp4	Scaling Factor		Europet fillerer	far 22 bit anna 🛛		ОК
		- an % X		Export filters	TOF 32-DIT SCOPE		

5. Now, configure the system so that when you press Apply on the Main Menu, you will also automatically run DPOJET. On the Main Menu, press Config. On the menu that comes up, shown below, check the box that says Automatically run Analyze when pressing Apply. Under Configure Analyze Button, select Auto Configure. Press OK.

_	_			_	_
Config	ure Apply Butto	on			
	Automatically ru	n Analyze when pres	sing Apply		
Config	ure Analyze Bu	tton			
C	Recalculate				
	Use current filte	rs and recalculate without	changing DPOJET setup	, ,	
C	Clear and Recal	culate			
	Use current filte	ers and clear data and rec	alculate, without changing	g DPOJET setup	
۲	Auto Configure		000157		
	Use current filte	rs and automatically setu	p DPOJET parameters		
Wavet	form to use for	Apply and Analyze			
0	New		Use Current		
	Acquire new wfm	is and apply filters	Apply filters to	current wfms	

- 6. Now, process the models by pressing **Apply** on the Main Menu. This will compute the transfer function at **Tp2**. The resulting waveform will be visible on the oscilloscope in Math 1. Since DPOJET was configured to run automatically, DPOJET will setup to measure TIE and plot the Eye Diagram at Tp2.
- 7. After SDLA Visualizer has finished processing, select **Plot** on the Main Menu to view the Phase, Magnitude, Impulse and Step response at Tp2.
- 8. In the upper right area of the Main Menu, observe the bandwidth setting. It may be necessary to fine-tune this based on the results, as de-embedding can amplify noise. For details on fine-tuning the bandwidth, see *Test Point and Bandwidth Manager*.

Since cables typically have low attenuation, the auto-bandwidth may result in too wide of a bandwidth. If so, go to the Test Point and Bandwidth Manager by pressing on a test point button on the Main Menu. Under **Global Bandwidth Limit** select **Custom**, and set up the desired response.

# Example of embedding a serial data link channel

This example shows how to create and embed a model of a serial data link channel in order to simulate its effects upon the acquired signal from the DUT. In this case, the channel to be modeled consists of a twisted pair of cables, connectors and a printed circuit board.

This example uses Single Input mode.



- 1. The first step is to define the Measurement Circuit model. Use Step 1 and 2 in the above *Example of De-embedding a Cable* as a guide for representing how the waveform was acquired.
- 2. Next, define the Simulation Circuit model.
  - a. Press Tx on the Main Menu once again, which brings up the Tx Configuration Menu, shown below. On the bottom row, select Nominal in the Impedance drop-down list, and enter 50 Ohms in the c1 and c2 text boxes. The Tx source impedance for the embed path should be set the same as for the de-embed path. Press OK.

**NOTE.** The impedance value should match the value that you would like to simulate. This could be the actual impedance of the transmitter, or another impedance value.

SDLA Visualizer - Tx Configuration			
Setup Thevenin Equivalent model for de-er	nbed and embed paths		
		50 b1 ohms 50 b2	Impedance Nominal
© Emphasis	NA Tp11	NA 50 o1 ohms Tp12 50 o2	Impedance Plot Nominal  CK

b. Press Embed on the Main Menu, which brings up the Embed Menu.
 Select the Cascade Tab. Configure blocks to represent the PCB traces, connectors and twisted pair of cables using B1–B8.

🚺 🛛 SDLA Visualizer - De-embed		
Cascade Normalize Convert Resample	$\begin{array}{c} c_{1} \\ c_{2} \\ c_{2} \\ c_{3} \\ c_{4} \\ c_{4} \\ c_{4} \\ c_{4} \\ c_{5} \\ c_{4} \\ c_{5} \\ c_{6} \\$	Save Recall OK

c. Press on the last block of the cascade, which will be labeled **RX Load**. This brings up the *Load Configuration menu*, shown below. Under "Model", select **Nominal**. Enter **50** Ohms for the impedance on both ports. Press **OK**.

4	SDLA Visualizer – Load Block Configuration			
	Select the load model and the applicable file(s) if requir	red		
	Model Nominal	ohm Port 1 v 50 ohm Port 2 v 50	Label Load	ок

- d. Optionally, configure the Rx Block by pressing **Rx** on the Main Menu.
- **3.** Once the Measurement Circuit and Simulation Circuit models have been defined, the desired test points can be enabled. (For more information on test points, see *Understanding Test Points*.)
  - **a.** Press on any test point on the Main Menu, which brings up the Test Point and Bandwidth Menu.
  - b. Turn on, map and label the test points as appropriate. The test points in the Simulation path are: Tp2, Tp3, Tp8, Tp9, Tp11 and Tp12. (If you have configured the Rx Block, those test points are Tp4, Tp5 and Tp10.)

🛃 SDLA Visualizer - Test	Point and Bandwidth	n Configuration						
Tp2 - Map the desire	d test points to Math	to see the processed waveform						
Tp On/Off	Map Tp to I	Math	Label	Filters	Global BW Limit	Delay	H	
Math1	Tp2	•	Tp2	Save	🔘 None	🔘 Keep Delay	Ы	
Math2	ТрЗ	•	ТрЗ	Save	Auto	Remove Delay		
Math3	Tp8	•	Тр8	Save	Custom	Adjust Delay		
Math4	Tp11	•	Tp11	Save				
C Ref3	Tp5				BWI: 25GHz			
🔿 Ref4	Tp4	Scaling Factor		Export filters	for 32-bit scope ?			ок

- c. Press OK.
- **4.** Now, configure the system so that when you press **Apply** on the Main Menu, you will also automatically run DPOJET.
  - a. On the Main Menu, press Config.
  - **b.** On the menu that comes up, shown below, check the box that says **Automatically run Analyze when pressing Apply**. Under Configure Analyze Button, select **Auto Configure**.

Conf	gure Apply Button	1		
	Automatically run	Analyze when pre	ssing Apply	
Conf	gure Analyze Butt	ton		
	Recalculate Use current filters	and recalculate witho	ut changing DPOJET setup	
	Clear and Recalco Use current filters	ulate and clear data and re	calculate, without changing DPOJET setu	ıp
	Auto Configure Use current filters	and automatically set	tup DPOJET parameters	
Wave	eform to use for A	pply and Analyze	•	
	New Acquire new wfms	and apply filters	Use Current Apply filters to current wfms	

### c. Press OK.

- 5. Now, process the models by pressing **Apply** on the Main Menu. This will compute the transfer functions at the configured test points. The resulting waveforms will be visible on the oscilloscope. Since DPOJET was configured to run automatically, DPOJET will setup and open. Observe the eye diagrams of the test points in the DPOJET plot window.
- 6. After SDLA Visualizer has finished processing, select **Plot** on the Main Menu to view the Phase, Magnitude, Impulse and Step response graphs.

# Example of de-embedding a high impedance probe

In this example,

- The user wants to use a high impedance probe to view a test point in an active system with a transmitter channel and receiver, while the probe is loading the system.
- The user also wants to view the same point in the system with the probe deembedded and not loading the system.

This example uses Single Input mode.

First, set up the Measurement Circuit (de-embed) path to include the high impedance probe in the circuit diagram:

1. On the Main Menu, select the **Single Input** radio button. Then, press **De-embed.** 

	SDLA Visualizer		- • ×
	Tektronix	O View O GPIB	Help About
	Measurement Circuit     Tp6     Tp7     Single Input     Dual Input       Tp11     Tp12     Tp1     De-embed     Single Input     Dual Input       Tx     Tp1     De-embed     Tp1     Tp1       Tx     De-embed     Tp1     Tp2       Simulation Circuit     Tp2     C2     Tp3	hit: TED 400 GS/s 8 Gb/s 4 Data 4 Data 4 Clook 5 Clook 5 Save Rec	t Tig Zze L t all Default
l	Press Apply to update filter		

2. This brings up the De-embed Menu. Select the **High Z** radio button. This causes a **Probe** button to appear on the circuit diagram, as well as an arrow that lets you move the location of the probe on the diagram.

🛃 🛛 SDLA Visu	alizer - De-embed	
Cascade Normalize Convert Resample	$ \begin{array}{ c c c c c c c c c c c c c c c c c c c$	Save Recall OK

3. Press the **Probe** button. This brings up the Probe Configuration Menu:

SDLA Visualizer - Probe Configuration				
Define S-parameter files for the Probe and Scope				
Configure Probe/Scope	Probe + Port 1 v ? Ideal v Por Port 2 v Filename	Scope Browse Ideal Ch Ideal	Label Probe	Pict Probe Scope OK

4. In the **Probe** panel, select the correct probe model. Press **Load** to bring up the probe browse menu, and load the 3-port S-parameter file.

- 5. If you'd like to plot the probe S-parameters, under **Plot** on the right side, press **Probe**. Press **OK**.
- 6. On the De-embed Menu, the circuit diagram shows the probe in the circuit, and the test points in that circuit are with the probe loading the circuit. This includes **Tp1**, **Tp6**, and **Tp7**. In other words, all test points in SDLA produce waveforms according to what the circuit diagram shows. If the probe is shown in the circuit, then it is loading the circuit. (To see the circuit and test points *without* the probe loading it, press **Embed Block** on the Main Menu and recreate the same circuit, where there is no probe.)
- 7. For the blocks on each side of the probe connection point, load S-parameter files or create models of RLC and/or transmission line to represent the system that the probe is connected to.
- 8. From the Main Menu, press Tx. The Tx Configuration Menu comes up:



9. Set up the Tx Block to best represent the DUT transmitter source impedance that the system is connected to. Press **OK**.

Next, set up the Simulation Circuit with the same circuit blocks and parameters that were used to set up the Measurement Circuit, but without the probe, following the steps in the diagram below.

- 1. From the Main Menu, press Tx. The Tx Configuration Menu comes up once again (see above).
- 2. Set up the Tx Block as above. Press OK.
- 3. Press Embed in the Main Menu. This brings up the Embed Menu.



4. Configure the cascade the same way as the Measurement Circuit side, but without the probe (note that no probe options are available on the Embed menu). Press **OK**.

The test points in the Simulation Circuit side now represent the system *with no probe loading*. These include **Tp2**, **Tp3**, **Tp8**, **Tp9**, **Tp11**, and **Tp12**. The test points in the measurement circuit remain with the probe loading the circuit.

Now, return to the Main Menu. Open up the Test Point and Bandwidth Manager by pressing on any test point. Turn on and map the desired test points to math waveforms.

Return to the Main Menu. Press **Apply**. This will compute the transfer functions for the enabled test points. The resulting waveforms will be visible on the oscilloscope graticule as math waveforms according to how you mapped them.

# Example of de-embedding significant reflections with dual input waveforms

By using SDLA Visualizer, an accurate de-embedding result may be obtained on a real-time oscilloscope, even when the components have severe gain variations.

The purpose of this example is to show:

- How to set up SDLA Visualizer to de-embed significant reflections coming from an unmatched termination at the end of a pair of 38 inch cables.
- How SDLA test points can be used to represent the generator reference plane with the measurement circuit load. In this case, **Tp1** shows the output of the generator with the measurement system loading it.
- How test points can represent the generator reference plane with the ideal load, thus producing a much cleaner waveform with no observable reflections. In this case, **Tp2** shows the output of the generator terminated by an ideal 50 Ohms.

The acquisitions are averaged in order to reduce the noise from the boost, and a repetitive step function signal is used.

### **Equipment Setup**

In the image below, a step generator is driving a balun to provide a differential step signal. Two 5X attenuators are connected to the two outputs of the balun where the green cables are connected. These ensure minimum reflections from the generator at the reference plane for de-embedding. In order to provide a large reflection for the purpose of this example, the opposite ends of the green cables are each connected to a T and coupler combination.



Below, the outputs of the T couplers are connected to the CH1 and CH2 inputs of the oscilloscope:





In order to provide a set of S-parameters that SDLA Visualizer modeling blocks can use to create real-time waveforms, the 2-port S-parameters for each cable and for each T coupler combination were measured on a VNA.

The T coupler with an open circuit is treated as a single 2-port element, with port 1 where the green cable is connected, and port 2 where the oscilloscope is connected. The open circuit port is simply part of the device characteristic. In this example, the oscilloscope and the generator reference plane impedance are assumed to be ideal 50 Ohms.

### Setting Up SDLA Visualizer

#### First, configure the test points:

- 1. On the SDLA Main Menu, select **Dual Input** mode. Press a Tp button to bring up the **Test Point and Bandwidth Manager.**
- 2. Select all four test points to be on (Math1, Math2, Math3, and Math4).
- 3. Map **Tp1** to Math1 and to Math2. Tp1 is the output of the generator with cables and T and scope loading it. Select **Math1** to be the A line of the differential test point. Select **Math2** to be the B line of the differential test point.
- 4. Map **Tp2** to Math3 and Math4. Tp2 is the output of the generator with ideal 50 ohms loading it. Select **Math3** to be the A line of the differential test point. Select **Math4** to be the B line of the differential test point.

A Visualizer - Test Poi 1 - Map the desired te	int and Bandwidth Config est points to Math to see t	uration the processed waveform	_	_		-	
Tp On/Off	Map Tp to Math	Select Tp Mode	Label	Filters	Global BW Limit:	Delay	
Math1	Tp1: Main 💌	A 💌	Tp1A	Save	None	C Keep Delay	
Math2	Tp1: Main 💌	в	Tp1B	Save	Auto	Remove Delay	
Math3	Tp2: Main 💌	A 🔹	Tp2A	Save	Custom	Adjust Delay	
Math4	Tp2: Main 💌	в	Tp2B	Save	Setup BW		
C Ref3	Tp5						
Ref4	Tp4						ОК
				Export	filters for 32-bit scope		

#### Next, configure the bandwidth limit:

- 1. On the Test Point and Bandwidth Manager, under Global Bandwidth Limit, select Custom (shown above).
- On the resulting Bandwidth Limit Filter Design menu (shown below), set BW GHz to 10.



- 3. Set Stopband GHz attenuation to 12.
- 4. Set Stopband -dB to -80.
- 5. Press Apply. This saves the BW limit filter for the simulation.
- 6. Press Close to return to the Test Point and Bandwidth Manager. Under Delay, select Keep Delay.

### Now, configure the De-embed Block:

1. In the SDLA Main Menu, press **De-embed**. This opens the De-embed Menu shown below.



2. Under Probe, select None.

**3.** In the cascade diagram, press the first cascade block (**B1**) to open up the **Block Configuration Menu** for B1, shown below. Select the **File** tab at the left.

📣 SDLA Visualize	r - Block Configuration			
Select B1 mode	el and the applicable files if required			
Thru File RLC	Model 2-Port	TEKCable_PN174142700_20inch_DUT1_061212.s2p Port1  Port1  Port2	Label B1 Check	
T Line	Scaling Factor	Port1  Port2		Plot

- 4. Under Model, select 2-port.
- **5.** Press the upper **Browse** button load a 2-port S-parameter file for one of the cables.
- **6.** Press the lower **Browse** button to load a 2-port S-parameter file for the other cable.
- 7. Edit the block label to say "cables".

### Next, view the S-parameter plots for the cables:

1. Press the **Plot** button to bring up the cable S-parameter plots that will appear in two windows. One cable is shown below:



2. Press the **Plot TD** button in the S-parameter plot menu to open another window, shown below, containing the time domain plots for each S-parameter:



**3.** Return to the main S-parameter Plot window, shown below. Select the Overlay **Show** radio button. A set of check boxes appears that allow any of the S-parameters to be selected for the overlay plot.



4. At the bottom of the Overlay panel, select **Plot**. This brings up another window, shown below, that contains the overlaid plots. These show magnitude, phase, impulse and step response for the various S-parameters:



5. To view a Smith Chart, return again to the main S-parameter Plot window, and press **Plot Z** on the right. Another window will open, shown below, that contains a Smith chart showing each reflection coefficient to view impedance for one of the green cables:



6. To view a chart of the magnitude of impedance into each port, return to the main S-parameter Plot window, and press the **Plot Z(f)** button on the right. This opens another window, shown below.



### Now, set up the block model for the T and coupler circuits:

1. Now, return to the De-embed Menu and set up the second block in the cascade diagram, **B2**, to represent the T and coupler circuits, as shown below:

🛃 SDLA Visualize	r - Block Configuration							
Select B1 model and the applicable files if required								
	Model	p12_JP_wTermination_100512.s2p	Label					
Thru File	2-Port	Port1	Check					
T Line	Scaling Factor	Port1	Passivity	Plot				
		p34_JP_wTermination_100512.s2p		OK				

- 2. Select the File tab at the left.
- 3. Under Model, select 2-port.
- **4.** Press the upper **Browse** button to load a 2-port S-parameter file for one of the combined T and coupler components.
- **5.** Press the lower **Browse** button to load a 2-port S-parameter file for the other combination of T and coupler.
- 6. Edit the block label to say "T coupler."

#### Next, view the plots for the T and coupler combination:

1. Press the **Plot** button. This brings up the main S-parameter Plot window, shown below. Observe that the S21 of the T coupler has some deep nulls of -30 to -40 dB. De-embedding these will require significant gain in the transfer function. Gain will cause a boost in noise at those frequencies. In this example, the oscilloscope should be put into average mode in order remove the excessive noise.



2. The time domain step response for the T and coupler combination, shown below, shows the reflection characteristic response that will be observed on the waveforms acquired on the oscilloscope:



**3.** The **Overlay** plots for the T and coupler combination are shown below. The step response for S21 is shown in the lower right corner. This general response shape can be seen on the step response edges of the acquired waveform, because this discontinuity in impedance is very close to the input port of the oscilloscope.



1. The reflection coefficients, **s11** and **s22**, and the resulting impedance for the T coupler are shown on the Smith Chart below. Note that the range is over a large percentage of the chart. This makes for a challenging de-embed operation, given the 8-bit resolution of the scope and the noise within the scope. Averaging will help with these issues.



2. The image below shows the Impedance magnitude vs. frequency plot for the T and coupler combination:



#### Now, apply the models:

Go to the SDLA Main Menu and press **Apply**. This will cause SDLA to generate transfer functions based on the S-parameter models of the system. SDLA does this by taking into account all of the S-parameters throughout the system. In other words, cross-coupling, reflections, and transmission terms are all part of the transfer function. These transfer functions represent the filters applied to the acquired waveforms from the oscilloscope to obtain the test point waveforms defined in SDLA.



### Next, view the live waveforms on the oscilloscope display:

Once the filters for each test point transfer function are computed, they are automatically loaded into the oscilloscope math menu, and live waveforms appear on the scope display.

1. The image below shows the **Tp1** waveforms, which are the outputs of the Tx model with the cables and T coupler combination still loading the step generator. The waveforms acquired on the scope are in yellow and Cyan. The waveforms de-embedded to reference plane with cables still loading that point are shown in purple and orange.

The resulting reflections and time delays are correctly represented in the deembedded waveforms. These are the A and B signals that were acquired directly on the oscilloscope at the output from the T couplers.

The reflections due to the open circuit of the coupler can be seen on the acquired waveform at the rising and falling edges where the steps entered the oscilloscope (yellow and cyan). On the de-embedded waveforms (purple and orange), the rising and falling edges have the reflections removed, and the shape of the pulse is as expected at the reference plane into the cables. There is a delay of approximately 4.1 ns through the cable. At 8.2 ns, the round trip time of 8.2 ns, the reflection arrives back at the reference plane input to the green cables. Also, when the pulses (purple and orange) go back to zero, another round trip reflection off the input side to the scope arrives back to the reference plane 8.2 ns later.


2. The image below shows **Tp2**, which in SDLA is at the output of the Tx model with the embed side of the system loading the Tx model. In this example, Tp2 provides the waveforms with the measured cables and components removed, and the step generator reference plane terminated in an ideal 50 ohms. All blocks were left set to default Thru models, and the load was default 50 ohms.

Note that the de-embedded waveforms shown at the Tx reference plane below do not have major reflections; the wave-shape is the expected shape of the step generator. The major reflections on the acquired waveforms were removed when transforming it to the waveform in the simulated circuit at Tp2. The acquired waveforms from Ch1 and Ch2 of the scope are shown in yellow and cyan. These are the waveforms acquired through the cable and T/ coupler combination. The de-embedded waveforms with measurement circuit replaced by ideal 50 ohms are shown as green and red. SDLA Visualizer is capable of showing up to four test point waveforms simultaneously while up to 48 are available to view.



### Finally, view the plots of the test point filters:

Once the **Apply** button is pressed in the SDLA Main Menu and the transfer function filters have been computed, you may view the plots of the test point filters that result. Press the **Plot** button in the Main Menu, as shown below.

Notice that the magnitude vs. frequency plot shows high gain at the frequencies where the T coupler **S21** shown large dips. This is necessary in order to deembed out the effects of the T-coupler, which causes major reflections. This results in the impulse response at the lower left to ring for a long period of time. In addition, the step response at the lower right also shows the ringing over a long period of time. This results in long lengths in numbers of filter coefficients when the sample rate is high.



## Example of removing a DDR reflection with a single input waveform

This example demonstrates how SDLA Visualizer can be used to remove reflections for a low-speed DDR case, using a simplified set of block models.

In this situation, several elements are in place between the probe point and the Rx load resistor: a delay line that can be modeled use T-line model, a package that is modeled using an S-parameter block, and a non-ideal load resistor. Because the probe point is not near the load resistor, and the load resistor is non-ideal, a large reflection can be observed in the waveform acquired at the probe point:



Even though the waveform at the actual probe point is not suitable for any jitter measurements due to the large reflection, the waveform at the load resistor has a regular pulse shape. This example shows how SDLA can be set up to de-embed the reflection from the waveform at the load resistor.

In this example, three blocks in the De-embed cascade diagram will be used to model the delay line, package and load resistor:

- the delay line will be modeled using a T-line model in Block **B2**,
- the package will be modeled using a 4–port S-parameter file in Block B3,
- and the non-ideal load resistor will be modeled as nominal impedance in the Load Block (final block in the cascade).

### First, set up.

1. On the Main Menu, select the **Single Input** radio button.

In the **De-embed block**, select the **Hi Z** probe option. Ideal Hi Z probe is assumed.

<b></b>	SDLA Visualize	r - De-embed		
	Cascade Normalize Convert Resample	Move Tp8 Tp7 Probe None SMA TDT High Z High Z	$\begin{array}{c} & \text{Probe} \\ \text{b1} & 1 & \text{Thru} & 2 \\ \text{B1} & \text{The} & 1 & \text{Thru} & 2 \\ \text{B2} & \text{B3} & \text{B4} & \text{B4} \\ \text{B2} & \text{B3} & \text{A4} & \text{B4} \\ \text{B2} & \text{B3} & \text{A4} & \text{B4} \\ \text{B4} & \text{B4} & \text{L2} \\ \text{B5} & \text{B4} & \text{B4} & \text{L2} \\ \text{B5} & \text{B4} & \text{B4} & \text{B4} \\ \text{B6} & \text{B7} & \text{B7} & \text{B6} \\ \text{B7} & \text{B7} & \text{B8} & \text{Tp7} & \text{I} \\ \text{B8} & \text{Tp7} & \text{I} \\ \text{B8} & \text{Tp7} & \text{I} \\ \text{B8} & \text{B7} & \text{Tp7} & \text{I} \\ \text{B8} & \text{B7} & \text{B7} & \text{I} \\ \text{B8} & \text{B7} & \text{B7} & \text{I} \\ \text{B8} & \text{B7} & \text{B7} & \text{I} \\ \text{B8} & \text{B1} & \text{B1} & \text{B2} \\ \text{B1} & \text{B2} & \text{B2} & \text{B3} \\ \text{B2} & \text{B3} & \text{B4} & \text{B4} & \text{B4} \\ \text{B3} & \text{B4} & \text{B4} & \text{B4} \\ \text{B4} \text{B4} & \text{B4} & \text{B4} & \text{B4} & \text{B4} \\ \text{B4} & \text{B4} & \text{B4} & \text{B4} & \text{B4} \\ \text{B4} & \text{B4} & \text{B4} & \text{B4} & \text{B4} \\ \text{B4} & \text{B4} & \text{B4} & \text{B4} & \text{B4} & \text{B4} \\ \text{B4} & $	Save Recall OK

### Next, model the T-line:

1. Estimate the transmission line delay using the horizontal cursor measurement tools to get the delay for round trip reflection. Then, divide by 2:

Td = 660 ps/2

= 330ps



 Set up a block using a lossless transmission line model with 330ps delay. Select block B2 in the De-embed menu and select the T-line tab. Enter 0.33 in the Delay ns field. Press OK.

🛃 SDLA Visualizer - Block Configura	ation			
T_line Configuration				
Thru File RLC T Line	Z0 ohms 50 Delay ns 0.33 Freq Space MHz 10.0	1   Transmission Line     3   Transmission Line	Label T_line	Plat OK

### Next, model the package:

1. The package is modeled using a 4-port single-ended S-parameter file. Press **B3** on the De-embed cascade diagram and press **Browse** to select an appropriate file. Note that the port assignments are different from the default port assignments: [1 3 2 4] instead of [1 2 3 4].

🚺 SDLA Visualize	r - Block Configuration						
Package Config	e Configuration						
Thru	Model 4-Port Single-ended	pkg.s4p	Label Package				
File		Port1  Port2  Port2	Check				
T Line	Scaling Factor	Port3  Port4	Port assignments				

2. To see if the port assignments are correct, you can press the Check button located on the File tab. You can also check the S-parameter plot to identify insertion loss terms to assign ports properly. Press OK.

### Now, configure the Load Block.

- 1. First, estimate the value of the load resistance in order to enter it into the Load Block. Use the vertical cursor measurements to get a ratio of reflected to incident voltage.
- **2.** Next, compute the resistance:

 $T(\Gamma) = (V2-V1)/V1$ = (1.25-0.75)/0.75 R = Zo(1+T)/(1-T) = 200 Ohms



**3.** Set up the Load Block using 200 Ohms resistance. On the De-embed Block, pressing on the Load Block (the final block). This brings up the Load Configuration Menu, below. Enter this value into Port 1 of the Load Configuration Menu. Press **OK**.

-	SDLA Visualizer – Load Block Configuration		
Select the load model and the applicable file(s) if required			
	Model Nominal	ohm Port 1  50 ohm Port 2  50	Label Load

After all the blocks have been set up in the De-embed menu, return to the Main Menu and press **Apply**. SDLA creates the filter for **Tp7** and turns on the de-embedded waveform on the scope, as shown below.

- White represents the original acquired signal WITH the reflection.
- Purple represents the de-embedded result showing the reflection removed. This is the waveform that the Rx block "sees" at the Rx load resistor. Note that the de-embedded waveform has regular pulse shapes; measurements such as jitter can be performed upon it.



# **GPIB** remote control

### **Using GPIB remote control**

You can use GPIB commands to control SDLA Visualizer remotely, and to monitor the SDLA GPIB activity using the application Main Menu. The GPIB commands listed are separate from, and handled differently than, the DPO70000 Series Oscilloscope GPIB commands. (Note that a TekScope command, *APPLICATION: ACTIVATE SDLA*, is used to start the SDLA Visualizer application.) Press here to see the *set of SDLA Visualizer commands available*.

Analysis results are available by querying the DPOJET application either by using its GPIB command set, or from the oscilloscope front panel. Use the GPIB command interface for the DPOJET application to retrieve measurement results. (Refer to the DPOJET online Help or the PDF document derived from it for DPOJET GPIB control information.)

### GPIB Control Menu

Select **GPIB** on the upper right corner of the SDLA Visualizer Main Menu to enable the GPIB function. Press **View** to monitor command traffic to and from SDLA Visualizer. In the following illustration, GPIB is enabled, and View is selected to show SDLA Visualizer command traffic. Other GPIB command traffic is not shown. Use an application such as the TekVisa OpenChoice Call Monitor to see all GPIB traffic.



The GPIB status includes the following values:

- Sent shows the status sent after executing the last command. May be OK or ERROR.
- Received shows the last value read from the SDLA Visualizer handshaking variable. It is either OK, meaning no command is available, or it shows the received command now being processed.
- Last shows the last instruction executed.

- Cmds contains the count of the commands received since you enabled the GPIB function.
- Idle contains the count of the number of polls of the handshaking variable by the SDLA Visualizer application since completing the last command. GPIB communications is enabled by default.

Using the SDLA Visualizer application's GPIB interface consumes additional compute resources.

### **Handshaking Protocol**

The SDLA Visualizer application handles GPIB communications through its own protocol handshaking.

The requirements for SDLA Visualizer GPIB communications with a controller are as follows:

- 1. Once the SDLA Visualizer application has started, it writes an "OK" status to the SDLA Visualizer handshake variable. This tells the controller application that it may now write a valid SDLA Visualizer command into the "sdla' variable.
- **2.** The GPIB controller polls the handshake variable (variable:value? "sdla") until it detects the OK status.
- **3.** The GPIB controller writes a command string into the SDLA Visualizer handshake variable. For example, sending the command 'variable:value "sdla", "p:apply" writes the string "p:apply" into the variable "sdla".
- 4. The SDLA Visualizer GPIB function polls the handshake variable, reads the command string and interprets it as a command. If the command is bad, it writes an ERROR handshake value to the variable.
- **5.** A good command is parsed and executed. On successful execution, it writes an OK to the handshake variable. When the GPIB controller reads the OK status, it may send a new command string.

### **GPIB** commands

This section lists the commands available for remote control of the SDLA Visualizer application.

NOTE. "sdla" must be in lower-case characters.

### **APPLICATION: ACTIVATE "Serial Data Link Analysis"**

This command instructs the oscilloscope to start the SDLA Visualizer application. It is a set-only parameter. (Note: this is a TekScope command, not an SDLA Visualizer command).

**NOTE.** This command must use the exact syntax below.

### Syntax

APPLICATION: ACTIVATE "Serial Data Link Analysis"

#### Arguments

"Serial Data Link Analysis" which must be just as defined in the syntax and enclosed in double quotes (" ").

#### Returns

NONE

## VARIABLE:VALUE "sdla", "p:exit"

Closes the SDLA Visualizer application. The current state of the application is not saved.

#### Syntax

VARIABLE: VALUE "sdla", "p:exit"

### Arguments

"p:exit" forces the application to close.

### VARIABLE:VALUE? "sdla"

Reads the value of the SDLA Visualizer handshake variable. The returned status must be "OK" before you can send any other commands.

#### Syntax

VARIABLE:VALUE? "sdla"

### Arguments

None

### Returns

OK: The SDLA Visualizer application is running and ready for a command.

ERROR: The SDLA Visualizer application was not able to parse or run the previous command.

## VARIABLE:VALUE "sdla", "p:adapttaps:<value>"

Configure the equalizer adapt taps setting.

### Syntax

VARIABLE: VALUE "sdla", "p:adapttaps:<value>"

### Arguments

"p:adapttaps:<value>" specifies the adapt taps could be any one of auto | from\_current | none.

### Example

variable:value "sdla", "p:adapttaps:auto" sets the adapt tap to be auto.

### VARIABLE:VALUE "sdla", "p:bitrate:<value>"

Sets the bit rate for the source waveform. Determine the native bit rate of the source waveform and use that value.

### Syntax

VARIABLE:VALUE "sdla", "p:bitrate:<value>"

### Arguments

"p:bitrate:<value>" specifies the bit rate of input source waveform. The <value> must be an integer in either engineering notation (6.25e6) or as a regular number (6250000).

### Example

variable:value "sdla", "p:bitrate:6e9" sets the source bit rate to 6 Gb/s.

### VARIABLE:VALUE "sdla", "p:ctletype:<value>"

Configure the equalizer CTLE type.

Syntax

VARIABLE: VALUE "sdla", "p:ctletype: <value>"

Arguments

"p:ctletype <value>" specifies the CTLE type to be any one of Standard | IIR | FIR | None.

### Example

variable:value "sdla", "p:ctletype:none" sets the CTLE type to be none.

### VARIABLE:VALUE "sdla", "p:dfestate:<value>"

Configures the equalizer DFE state

### Syntax

VARIABLE: VALUE "sdla", "p:dfestate:<value>"

### Arguments

```
"p:dfestate:<value>" sets the DFE to be one of state: on | off
```

**Example** variable:value "sdla", "p:dfestate:on" sets the DFE on.

### VARIABLE:VALUE "sdla", "p:RunEQ"

Runs the Equalizer.

### Syntax

VARIABLE: VALUE "sdla", "p:RunEQ"

### Arguments

"p:RunEQ" forces the equalizer to run.

### Example

variable:value "sdla","p:RunEQ"

### VARIABLE:VALUE "sdla", "p:source:<source>"

Sets the input source waveform for the SDLA Visualizer application to operate on.

### Syntax

VARIABLE:VALUE "sdla", "p:source:<source>"

### Arguments

"p:source<source>" specifies the first input source waveform as any one of ch1 | ch2 | ch3 |ch4 | math1 |math2| ref1 | ref2.

### Example

variable:value "sdla", "p:source:ch1" sets the source waveform to be the oscilloscope CH1 input.

### VARIABLE:VALUE "sdla", "p:sourcetype"

Specifies whether to use Single Input mode (one differential signal source) or Dual Input mode (two sources used; usually each input is a leg of a differential signal).

#### Syntax

VARIABLE:VALUE "sdla" "p:sourcetype" <1|2>

#### Arguments

1 sets the source type to Single Input mode.

2 sets the source type to Dual Input mode.

### VARIABLE:VALUE "sdla", "p:recall:<path and file name >"

Loads a setup file from "path and file name". The setup file may be one of the included Standards setup files or a setup file you created with the SDLA Visualizer application interface. The setup file includes configuration of Rx/Tx, the enabled filter blocks and test points, and any custom FIR filters you specified in your custom setup.

### Syntax

VARIABLE:VALUE "sdla", "p:recall:<path and file name >"

#### Arguments

"p:recall:<path and file name >" where <path and file name > specify the path on a mapped drive and a setup file with the .sdl suffix. The path and file name must not contain space characters, but they may contain upper and lower case characters.

#### Example

```
variable:value "sdla", "p:recall:C:\TekApplications
\MyDirectory\mysetup.sdl" recalls the SDLA Visualizer application setup
file named mysetup.sdl.
```

### VARIABLE:VALUE "sdla", "p:source2:<source2>"

Specifies the second source waveform to be processed by the SDLA Visualizer application when using Dual Input mode.

#### Syntax

VARIABLE:VALUE "sdla", "p:source2:<source2>"

#### Arguments

"p:source2<source2>" specifies the second input source waveform as any one of ch1 | ch2 | ch3 |ch4 | math1|math2 | ref1 | ref2.

### VARIABLE:VALUE "sdla", "p:analyze"

Starts the DPOJET application and configures it to display the eye diagrams for the SDLA Visualizer application waveform(s) resulting from the Apply operation.

#### Syntax

VARIABLE:VALUE "sdla", "p:analyze"

#### Arguments

"p:analyze" starts the DPOJET application to display the SDLA Visualizer application waveforms.

### VARIABLE:VALUE "sdla", "p:apply"

Computes the enabled filter blocks and test points and performs equalization if enabled. The result is the same as selecting the front panel Apply button. The Apply computation may take over 60 seconds, depending on input data and sample rate. Make sure that your polling time-out is sufficiently long.

#### Syntax

VARIABLE:VALUE "sdla", "p:apply"

#### Arguments

"p:apply" starts computation of the enabled filters and equalization.

# Index

2 port model block configuration, 52
4 port differential model block configuration, 51
4 port modeling, 14
4 port single-ended model block configuration, 51

# A

Analyze button, 21 Apply button, 21

## В

Block Configuration Menu File Tab 2-port model, 51 4-port differential model, 51 4-port single-ended model, 51 FIR filter model, 51 high Z probe model, 51 transfer function model, 51 RLC network models, 54 RLC Tab, 54 T line Tab, 57

## С

Cascade Tab, 36 Chinese, 4 Clock recovery training sequence functions and FFE/DFE, 87 Config button, 21 Config Tab, 79 Configuring probes circuit configurations, 44 Conventions, 2 Convert Tab, 37 Converting single-ended to mixed mode S-parameters, 37 CTLE loading FIR filter, 84 parameters, 81 second-order, 81

## D

DDR removing reflection, 137 **De-embed Block** possible block configurations, 34 De-embed/Embed Menu Block Configuration Menu, 51 Cascade tab, 36 control buttons, 37 Convert Tab, 37 differences between De-embed and Embed Menus, 36 Normalize Tab, 36 re-normalizing S-parameters to different reference impedances, 40 Resample Tab, 37 De-embedding a cable, 110 De-embedding a high-impedance probe, 118 De-emphasis, 72 Decision feedback equalizer, 78 Directory path, 2 DPOJET configuring how SDLA will work with DPOJET, 97 plots, 60 using DPOJET and SDLA Visualizer together, 16 Dual and Single Input Modes example math expressions, 14 full 4 port modeling, 14 Main Menu diagram differences, 19 saving test points, 27 test point modes, 24

## E

Embed Block possible block configurations, 76 using to close the eye, 8 Embed cascade load block, 59 Embed Menu differences between De-embed and Embed Menus, 36, 59 Embedding a serial data link channel, 115 enable GPIB, 143 Equalizer running the Rx equalizer, 93 Exporting filters for use with 32 bit scope, 30

## F

FFE/DFE training sequence functions and clock recovery, 87 File Types and Locations, 2 FIR filter block configuration, 53 loading custom to set CTLE parameters, 84 reading from file using Tx Emphasis Menu, 72 saving, 2

# G

Global bandwidth limit creating a custom bandwidth limit filter, 32 Global bandwidth limits, 26 **GPIB** Commands APPLICATION: ACTIVATE "Serial Data Link Analysis", 145 SOURCETYPE, 148 VARIABLE: VALUE "sdla", "p:analyze", 149 VARIABLE: VALUE "sdla", "p:apply", 149 VARIABLE: VALUE "sdla", "p:bitrate:", 146 VARIABLE: VALUE "sdla", "p:exit", 145 VARIABLE: VALUE "sdla", "p:recall:", 148 VARIABLE: VALUE "sdla", "p:source:", 148 VARIABLE: VALUE "sdla", "p:source2:", 149 VARIABLE: VALUE? "sdla", 145 GPIB Control Menu, 143 GPIB enable, 143 GPIB handshaking protocol, 144 GPIB View selector, 143

### Η

Handshaking protocol, 144

High Z probe block configuration, 53 probe configuration, 42

## 

IBIS-AMI, 94 Input filters location, 2 Input sources live waveforms, 20 reference waveforms, 20 Installation, 1 Installation path, 2 Interposer circuit configuration, 44

## Μ

Main Menu components, 19 Measurement Circuit Model, 7 moving between applications, 3

## Ν

Normalize Tab, 36

### 0

Online Help Japanese, 4 Korean, 4 Option Key Requirement, 2 Output filters location, 2 saving, 2 Overlay, 60

### Ρ

Plots emphasis, 60 S-parameters 6-port, 4-port, 3-port, 2-port, 1-port, 60 impedance vs. magnitude, 60 overlay, 60 Smith Chart, 60 time domain, 60 test point filter (transfer function plots), 60 troubleshooting S-parameters, 66 Pre-emphasis, 72 Probe and tip selection choosing the correct 3 port S-parameter file, 48 P7520A, P7625, P7630 and P7633 probes, 50 SMA probes, 50

## R

Recalling a reference waveform, 20 Recalling a setup from a file, 22 Reference waveforms recalling a reference waveform, 20 Reinstall the SDLA Visualizer Software, 2 Remote control, 143 Resample Tab, 37 RLC, 54 Running a Test recommended order, 101 **Rx** Block Rx Configuration Menu, 79 second-order CTLE, 81 using to open the eye, 8 **Rx** Configuration Menu AMI mode, 80 Config tab adjusting FFE/DFE, 89 using PCIE3 option in CTLE, 84 using the PCIE3 option in FFE/DFE, 90 Config Tab using clock recovery for FFE/DFE equalization, 86 Taps Tab, 91 Thru mode, 80 TrainSeq Tab, 87 User mode, 79

## S

S-parameters available plots, 60 Load Configuration Menu, 57 using plots for troubleshooting S-parameters, 66 Save recall file location, 3 Saving a setup to a file, 21 Simulation Circuit Model, 8 Smith Chart, 60 Software Compatibility, 1 Software reinstallation, 2

# T

T line block configuration, 57 T Line Tab, 57 Test Point and Bandwidth Manager Global bandwidth limit, 26 Test Point Filter Delay Slider, 27 Test points crosstalk and reflection handling, 13 how test point filters are applied, 13 plotting test points, 25 table of test point descriptions, 11 Test Point Filter Delay Slider, 27 test point modes, 24 Text Conventions, 2 Thevenin Equivalent Voltage, 69 **Training Sequence functions** in FFE/DFE Equalization and clock recovery, 87 Transfer function model block configuration, 52 Transmission line block configuration, 57 T Line Tab, 57 **Troubleshooting S-parameters** bad phase response, 66 bad step response, 66 bad VNA measurements, 66 verifying mixed mode vs. single-ended mode, 66 viewing DUT with mismatched differential pairs, 66 Tx Configuration Menu, 70 Tx Emphasis Menu adding de-emphasis, 72 adding pre-emphasis, 72 plots, 71

reading from FIR filter file, 72

## U

Updates Software, 1

## V

View selector, 143

## W

Waveforms example file location, 2 selecting new or acquired, 98