# cādence<sup>®</sup>

# SystemSI – Serial Link Analysis Tutorial

Product Version 16.6 July 2014 © 2014 Cadence Design Systems, Inc. All rights reserved.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522. All other trademarks are the property of their respective holders.

**Restricted Permission:** This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

- 1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
- 2. The publication may not be modified in any way.
- 3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
- 4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

**Restricted Rights:** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

# **Table of Contents**

1	Single Ch	annel Analysis	7
	1.1 Ove	rview	7
	1.2 Star	ting the Single Channel Template	8
	1.2.1	Template Blocks	9
	1.2.2	New Workspace	9
	1.3 Cha	nnel Components	10
	1.3.1	TX_PRIMARY	10
	1.3.1.1	Connection Tab	11
	1.3.1.2	Stimulus Tab	13
	1.3.1.3	Jitter & Noise Tab	14
	1.3.1.4	Power Supply Tab	
	1.3.2	RX_PRIMARY	
	1.3.2.1	Connection Tab	
	1.3.2.2	Jitter & Noise Tab	
	1.3.2.3	Power Supply Tab	
	1.3.3	Pkg1	
	1.3.3.1	Connection Tab	
	1.3.3.2	Layout Extraction Tab	
	1.3.4	Pkg2	
	1.3.5	PCB	
	1.3.5.1	Block Name	
	1.3.5.2	Connection List	
	1.3.5.3	SPICE File List	
	1.3.6	AMI	
	1.3.6.1	Transmitter AMI	
	1.3.6.2	Receiver AMI	
	1.3.6.3 1.4 Ana	AMI Options lysis Options	
	1.4 Alla	Simulator	
	1.4.1.1	Circuit Simulator	
	1.4.1.1	Circuit Simulator Options/Channel Simulator Controls	
	1.4.1.2	Windows 32 Bit AMI DLL Support in a Windows 64 Bit SystemSI	
	Installat		
	1.4.1.4	Simulation Name	32
	1.4.2	Analysis Setup	
	1.4.2.1	General Channel Simulation Parameters	
	1.4.2.2	Simulation Configuration	
	1.4.2.3	Xtalk	
	1.4.2.4	Eye Distribution Methods	
	1.4.2.5	Statistical Eye Contours	
	1.4.2.6	IBIS Model Selection	
	1.5 The	First Simulation	
	1.5.1	Setting up Probe Point	35
	1.5.2	Begin Running the Simulation	
	1.5.3	Simulation Directory	
	1.5.4	Simulation Results	39
	1.5.5	Run the Simulation Again	40
	1.6 Incr	easing Data Rate	40
	1.7 Enal	ble AMI	
	1.7.1	Coeffout Parameter	
	1.7.2	Enabling AMI DFE @ RX_PRIMARY	44

2	Cross	talk Channel Analysis	46
	2.1	Overview	46
	2.2	Starting the Crosstalk Channel Template	46
	2.2.1	Channel Blocks	48
	2.3	Examining Each Channel's Component	
	2.3.1	Transmitters	48
	2.3.		
	2.3.		
	2.3.		
	2.3.		
	2.3.2	Receivers	
	2.3.		
	2.3.	2.2 Jitter & Noise Tab	51
	2.3.		
	2.3.3	Pkg1	
	2.3.	6	
	2.3.		
	2.3.4	Pkg2	
	2.3.5	PCB	
	2.3.		
	2.3.		
	2.3.6	AMI	
	2.4	Analysis Options	
	2.4.1	Circuit Simulator Options	
	2.4.2	Xtalk	
	2.4.3	Terminate Unconnected Nodes	
	2.5	Running Simulation	55
	2.5.1	AC Sweep	
	2.5.2	First Simulation (Default Settings)	58
	2.5.	2.1 Change Parameters	58
	2.5.	2.2 Pause before Simulation	59
	2.5.	2.3 General Options	59
	2.5.3	Simulation Results	60
	2.5.4	Enable AMI DFE	61
	2.5.5	Turn on Statistical Crosstalk	62
3	Sigrity	y AMI Models	63
	3.1	Model Design	63
	3.2	Model Configuration	64
	3.2.1	AMICDR Configuration	64
	3.2.	F	
	3.2.	1.2 AMICDR User Parameters	64
	3.2.2	AMICTWF Configuration	64
	3.2.	1	
	3.2.		
	3.2.3	AMICTWFADAPT Configuration	
	3.2.	- · · · · · · · · · · · · · · · · · · ·	
	3.2.		
	3.2.4	AMIFFE Configuration	
	3.2.	1	
	3.2.	1 05	
	3.2.		
	3.2.		
	3.2.5	AMIDFE2 Configuration	
	3.2.	5.1 AMIDFE2 Description	68

	3.2.5.2	2 AMIDFE2 LMS Algorithm	68
	3.2.5.3	3 DFE Coefficient Dynamic Adaptation Example	68
	3.2.5.4	4 AMIDFE2 User Parameters	69
	3.2.6	AMIDFENL Configuration	69
	3.2.6.1	AMIDFENL Description	69
	3.2.6.2		
4	Advance	ed Capabilities	71
	4.1 IB	IS Transmitter and Receiver	
	4.1.1	Add IBIS Transmitter and Receiver	71
	4.1.2	Load an IBIS File	
		veep Manager	74
	4.2.1	Launch Single Channel Complex Template	75
	4.2.1.1	Block Models	76
	4.2.2	Explore Sweep Manager	76
	4.2.2.1	I Sweep Mode	77
	4.2.2.2	2 Sweep Type	77
	4.2.2.3	3 Sweep List	
	4.2.2.4	4 Total Iterations	
	4.2.2.5		
	4.2.3	Running a Single Sweep	
	4.2.3.1		
	4.2.3.2	2 View the Results	
	4.2.4	Run Multiple Sweeps	
	4.2.5	Other Sweep Features	
	4.2.5.1	10	
	4.2.5.2		
	4.3 S-I	Parameter Extraction	
	4.3.1	Launch Single Channel Complex Template	
	4.3.2	S-Parameter Extraction	
	4.3.2.1	8	
	4.3.2.2		
	4.3.3	S-Parameter View	
		Parameter Wrapping and BBS Integration	
	4.4.1	Add S Parameter Block	
	4.4.2	Load S Parameter File	
	4.4.3	Extract the BBS Model	
	4.4.4	Load the BBS Model	
	4.4.5	Switch the Models	
		ock Sensitivity	
	4.5.1	Normalized Jitter and Noise	
	4.5.2	Launch Single Channel Complex Template	
	4.5.3	Set up Simulation Parameters	
	4.5.4	Start Block Sensitivity	
	4.5.5	View and Analyze the Results	
	4.5.5.1	5	
	4.5.5.2		
		sult Browser and 2D Curve Presentation	
	4.6.1	Result Browser	
	4.6.2	Compliance Curve Window	
	4.6.3	Curve Pane Context Menu	
		to Archive SSI Project	
_		egration with Allegro Signal Explorer	
5	-	ince Kits	
	5.1 Co	ompliance Workflows	126

5.1.1 5.1.2 5.1.3	comprante entens	127
5.2	SFP+ Compliance	
5.3	HDMI Compliance	134
5.4	PCIe Compliance	134
5.5	10GBASE-KR Compliance	136
5.6	USB 3.0 Compliance	137
5.7	MIPI Compliance	138

# A APPENDIX: BATCH MODE SUPPORT 140

# 1 Single Channel Analysis

This chapter describes how to use a template to perform a Single Channel Analysis. You will learn how to:

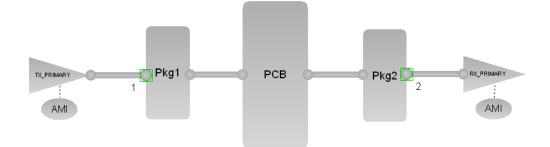
- Assign models to components
- Run the single channel simulations
- Set up the simulation parameters
- Set up the simulation options

What-if scenarios illustrate how results change when there are changes in the data rate and the inclusion of equalization via AMI modeling.

# 1.1 Overview

The single channel template contains:

- One transmitter
- One receiver
- Printed-circuit board (PCB)
- Two packages as shown below



As shown above, two AMI blocks in lighter grey connect to the transmitter and receiver, which are disabled by default.

The single channel template has different meanings for different designs.

#### Example

The PCB block could mean a backplane or an add-in module.

The Pkg2 block could be a flip chip or an SiP design or even a connector.

You can define the blocks according to your applications.

You can modify the template; for example, adding additional blocks which are part of your actual channel.

# **1.2** Starting the Single Channel Template

- 1. Launch SystemSI.
- 2. In the **File** menu, click **New**;

or, in the **Main** toolbar, click the **New** button

The Select Module dialog opens.

Se	ect Module ×
	Module Name
	Parallel Bus Analysis
	Serial Link Analysis
	Testbench
	OK Cancel

- 3. Select Serial Link Analysis.
- 4. Click OK.

The **New Workplace** dialog for a single channel template opens. You can now create a new workspace.

<ul> <li>Create a blank project</li> <li>Create by template T</li> </ul>	emplate Path: C:\Cadence\SPB_16.6\ASI\B	ase\SpeedXP\Library\template\
Name	Path	Description
measurement_xtalk	C:\Cadence\SPB_16.6\ASI\Base\S	peed
repeater_simple	C:\Cadence\SPB_16.6\ASI\Base\S	peed
single_channel_complex	C:\Cadence\SPB_16.6\ASI\Base\S	peed
single_channel_simple	C:\Cadence\SPB_16.6\ASI\Base\S	peed
sla_simple_em	C:\Cadence\SPB_16.6\ASI\Base\S	peed
vtalk channel simple	C+\Cadance\SDR 16 6\AST\Race\S	haar
lame:		
ocation:		

5. Select Create by template.

NOTE!	You can also click 🛄	to browse to the folder where templates are
	located.	

The paths for the templates are automatically generated during SystemSI installation. You cannot modify these paths.

<INSTALL\_DIR>\SpeedXP\Library\template\SystemSI\Serial Link Analysis\single\_channel\_simple

<INSTALL\_DIR>\SpeedXP\Library\template\SystemSI\Serial Link Analysis\single\_channel\_complex

<INSTALL\_DIR>\SpeedXP\Library\template\SystemSI\Serial Link Analysis\xtalk\_channel

 $<\!INSTALL_DIR >\!\!\!SpeedXP \ Library \ template \ SystemSI \ Serial \ Link \ Analysis \ measurement \ xtalk$ 

<ul> <li>Create a blar</li> <li>Create by ter</li> </ul>		mplate Path:	C:\Cadence\SPB_16.6\ASI\Base\Spec	edXP\Library\templa	ateV
Name		Path		Description	
measurement_xt repeater_simple	alk	-	ce\SPB_16.6\ASI\Base\Speed ce\SPB_16.6\ASI\Base\Speed		
single_channel_c	omplex		ce\SPB_16.6\ASI\Base\Speed		
single_channel_s		C:\Caden	ce\SPB_16.6\ASI\Base\Speed ce\SPB_16.6\ASI\Base\Speed		
	mole	CilCaden			•
Name: sc exa	mple 1				

- 6. Enter a name for the new workspace, such as **sc\_example1**.
- 7. Enter or select a location.
- 8. Click **OK**.

A folder with the same workspace name will be created in the location. The folder contains the workspace file **sc\_example1.ssix** and all models for the transmitter, receiver and other channel blocks. You'll use them to create new workspaces with all component models, connectivity and settings.

**NOTE!** Do not change the contents of any template folders.

# 1.2.1 Template Blocks

The single channel template has the following blocks:

TX\_PRIMARY: A primary transmitter.

**RX\_PRIMARY:** A primary receiver.

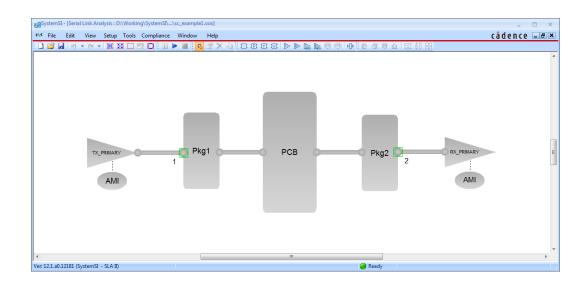
Pkg1 and Pkg2: Two packages.

**PCB:** A printed-circuit board.

AMI: Two AMI models.

## 1.2.2 New Workspace

The newly-created single channel workspace **sc\_example1.ssix** looks like the following figure. You can click to highlight any block or connection in this interface.



**NOTE!** The AMI blocks are enabled by default. Disable them first for the following steps of setting.

# 1.3 Channel Components

This section describes each block in the single channel workspace and examines the properties.

The properties of each block in the Single Channel Template have been set up. You can skip this section. Your simulation setup procedure is not affected.

# 1.3.1 TX\_PRIMARY

Double-click the **TX\_PRIMARY** block to open the **Property** pane.

tk Name: TX_	PRIMARY		File Name: D:\Working\SystemSI\sc_example1\tx_bh Sub-circuit Name: nmos_diff_tx
Conn. Port	Connect To Block Nam	e Conn. Port	.subckt nmos_diff_tx pos neg pwr in ngnd + nmos_imp=25
tx_pkg	Pkg1	DIE	+ tb_c_comp=Ip + tb_c_comp=Ip + tb_c_comp=Ip + tb_c_scale=1 ************************************
	111		Launch MCP Header Editor Edit Sub-circuit Definiti

The **Property** pane of the **TX\_PRIMARY** block contains four tabs:

- Connection
- Stimulus
- Jitter & Noise
- Power Supply

Connection Stimulus Jitter & Noise Power Supply

#### 1.3.1.1 Connection Tab

The **Connection** tab contains these parts:

- Block Name
- Connection List
- File Name and Sub-circuit Name
- SPICE Netlist File Content

#### 1.3.1.1.1 Block Name

Change the Block Name in the Block Name window.

Block Name: TX\_PRIMARY

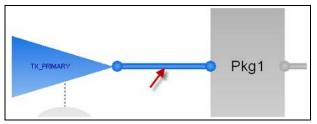
#### 1.3.1.1.2

#### **Connection List**

The **Connection List** displays connections between blocks. The **TX\_PRIMARTY** block has a connection (**tx\_pkg**) which connects to the **Pkg1** block at the **DIE** side.

Conn. Port	Connect To	Block Name	Conn. Port
tx_pkg		Pkg1	DIE

Click in the field to highlight the connection.



You can define and manage connections through the **Connection Definition** tab.

Click the **button** to open the **Connection Definition** interface, as shown in the following snapshot.

and a second				Block Name:	Pkg1	
nn. Port:	tx_pkg	-		Conn. Port:	DIE	
nName / CktNod	deName / NetName	X		PinName / CktNo	odeName / NetName	
1 / pos / po 2 / neg / ne				1 / posin 2 / negin		
3 / ngnd / q				3 / ngnd /		
Unconnect	ed Signal Net(s)			Unconnec	ted Signal Net(s)	
Unconnected Power Net(s)						
Unconnecte	ed Ground Net(s)		Unconnected Ground Net(s)			
7			Auto Connect	Auto net prope	arty overwrite	

#### 1.3.1.1.3

#### File Name and Sub-circuit Name

This section specifies the SPICE netlist file name and the sub-circuit model associated with the block.

File Name: D:\working\SystemSI\sc_example1\tx_bhvr.	sp	Sub-circuit Name:	nmos_diff_tx	-
---	----	-------------------	--------------	---

By default, the Single Channel Template has only one sub-circuit model specified for each block. You can choose your SPICE netlist file. Your file might contain several sub-circuit models.

#### 1.3.1.1.4 SPICE Netlist File Content

This content of the SPICE netlist file includes sub-circuit calls and an MCP section. The MCP section is used to maintain the connections between blocks.

.subckt tx_kr posineg pwr in ngnd	
+ tx_rt=50	
+tx_c_comp=1p	
* max swing	
* + tx_scale=1.27	
* typ swing	
* + tx_scale=1.06 * min swing	=
+ tx scale=0.84	
+ tx_lev=1000m	
****	*
* BEHAVIORAL CIRCUIT MODEL FOR 10GBASE-KR TRANSMITTER	
* for Cadence's SystemSI - Serial Link Analysis	
*****	*
* MODEL NOTES	
*	
* This is a Tx circuit model for 10GBASE-KR compliance testing, based on	
* the specification.	
* Intended for use with algorithmic model amiffe 10Gkr.ami/amiffe.dll.	
* This DLL is part of Cadence's SystemSI product.	
*	
*****	*
* MODEL PARAMETERS	
*	
* This model takes the following parameters:	
*	
* tx_rt > pullup termination	
* tx_c_comp > parasitic die capacitance * tx_scale > scale factor for swing, leave set to 1	
<pre>cx_scale &gt; scale ractor for swing, leave set to 1</pre>	-
▲ III	
Launch MCP Header Editor Edit Sub-circuit Definit	ion

It is recommended that you should not edit the MCP section. Make sure the sub-circuit nodes are called properly so that the connections are correct.

Click Edit Sub-circuit Definition to edit the SPICE netlist file.

To edit or modify the MCP header, select the Launch MCP Header Editor.

NOTE!	Do not edit the MCP section between * [MCP Begin] and * [MCP End].
NOIE:	SystemSI needs this section to manage the channel connections.

#### 1.3.1.2 Stimulus Tab

Data Rate:	3.125	Gbps	
Data Pattern:	Random	•	
Leading Bits:	leading_bits	.txt	
Delay:	0	ns	
🗹 Data Coding:	8b10b	•	
Rise/Fall Time			
Rise Time:	20	ps	
Fall Time:	20	ps	
Restore Defaults	1		

#### • Data Rate (Gbps)

Specifies the nominal data rate that the system will operate at. For example, PCI Express 2.0 operates at 5 Gbps.

The default value is 3.125 Gbps, which is based on the XAUI standard.

Data Pattern

Following stimulus types are available:

- Random (Default)
- **PRBS** (up to 100)
- User Defined (See User Defined Bit Pattern)
- Sinusoidal Waveform
- Sawtooth
- Clock
- Leading bits

One of the ways to change phase alignment during crosstalk simulation. Can also be used before the beginning of a bit stream for training pattern.

• Delay (ns)

Another way to change phase alignment in crosstalk simulation. Enter a delay in nanoseconds. Delay can be positive or negative; referred to as global zero time. Default value is 0 ns.

• Data Coding

Place statistical bounds on the rate of SignalTransitions, allows for easier clock recovery in the receiver, and for DC balance. Disabled by default.

Available coding types:

- 8b10b
- 64b66b (Default)
- 64b67b
- 128b130b

#### • Rise/Fall time (ps)

Specifies the rise and fall time of the Driver Signal. Disabled by default.

#### • Restore Defaults

Resets the field values to the original values that are displayed on the first launch of the tool. For regular templates, all fields, except **Data Rate**, are restored to their default values. In case of compliance templates, all fields are restored to their original values.

#### 1.3.1.2.1 User Defined Bit Pattern

You can create your own bit patterns to use with SystemSI. This functionality allows you to point to a text file containing the desired bit pattern. For example:

#### 0101111000110100..

The two periods at the end are required for the pattern to be repeated over and over until the desired number of bits has been reached. If the two periods are not included, just a short bit stream will be run. No post-processed outputs (ex. eye contour) will be generated.

An example of a bit pattern text file is provided as **bit\_pattern.txt** in the installation directory:

<INSTALL\_DIR>\SpeedXP\Library\template\SystemSI\

#### 1.3.1.3 Jitter & Noise Tab

The Jitter & Noise tab has two sections, Jitter and Noise, and a Restore Defaults button.

Jitter				Noise		
Periodic				Periodic		
Frequency:	2e+008	Hz		Frequency:	2e+008	Hz
Amplitude:	0.1	JUI		Amplitude:	10	mV peak
Frequency Offset:	100	ppm		Transition:	10	mV RMS
Transition Rj:	1	%UI RMS				
Transition Dj:	1	%UI peak				
DCD:	1	%UI				
Note: These parameters are	e incorporated in	to the stimulu	is hit st	tream applied to the Tx.		
noter mese parameters an	e men por dece m	no are sumaio	a alt a	a can applica to the TXI		
Restore Defaults						

**Restore Defaults**: For regular templates, selecting this button resets all field values to the product defaults. In case of compliance templates, all fields are restored to the value saved for that particular compliance.

#### 1.3.1.3.1

## Jitter

#### Periodic

This is one of the principle ways to test Jitter Tolerance. Specify the frequency of the sinusoid jitter source in Hz and the amplitude in UI.

#### • Frequency Offset (ppm)

Specifies the deviation from the nominal data rate in parts-per-million or ppm. Default value is 100 ppm. If the Bit Rate is 10 Gb/s then the actual rate can be 10 Gb/s +/- 1e6.

#### • Transition Rj

Applied to each logic transition of the transmitter's incoming bit stream (i.e. stimulus signal) in a Gaussian distribution, out to 8 sigma.

#### • Transition Dj

Applied to each logic transition of the transmitter's incoming bit stream (i.e. stimulus signal) in a rectangular window of equal probability.

#### • DCD (%)

Type of Dj. Describes the deviation in duty cycle value from the ideal value. Can also be modeled as an asymmetry between rise and fall time at the transmitter. Default value is 0% of the bit time.

#### 1.3.1.3.2

#### Noise

- **Periodic** Model the Noise that is usually introduced through the reference clock on the PLL. Modeled as a sinusoid. Specifies the frequency of the sinusoid Jitter source in Hz and the amplitude in % of input voltage swing.
- **Transition** (**mV**) Type of Dn and it's applied at each transmitter edge. Default value is 1% of input voltage swing.

**NOTE!** All noise elements are disabled by default.

#### 1.3.1.4 Power Supply Tab

✓ Ideal Power Supply: 1

• Ideal Power Supply – Ideal voltage source that is used at the transmitter. Default value is 1 V.

**Ideal Power Supply** should be defined according to the following rules, or else, the warning or error message will appear.

For ideal power simulation (Ideal Power is checked in the Analysis Options window),

- If the .sp file assigned to the Tx or Rx does not have its own power supply defined in the .sp file, **Ideal Power Supply** should be enabled in the **Property** GUI.
- If the .sp file assigned to the Tx or Rx has its own power supply defined in the .sp file,
   Ideal Power Supply should not be enabled in the Property GUI.

For non-ideal power simulation (**Ideal Power** is not checked in the **Analysis Options** window), both the Tx and Rx blocks should get the power supply from the VRM block.

# 1.3.2 RX\_PRIMARY

Double-click on the **RX\_PRIMARY** block to open the **Property** pane. The **Property** pane contains three tabs: **Connection**, **Jitter & Noise** and **Power Supply**.

ock Name: RX_F	RIMARY			File Name: D:\Working\SystemSI\sc_ex Sub-circuit Name: nmos_diff_r
Conn. Port	Connect To	Block Name	Conn. Port	.subckt nmos_diff_rx pos neg pwr ngnd rxnode + rx_rt=50
rx_pkg		Pkg2	DIE	+ rx_c_comp=1p ***********************************
	111			Launch MCP Header Editor Edit Sub-circuit Definition

#### 1.3.2.1 **Connection Tab**

The **Connection** tab contains these parts:

- Block Name •
- Connection List •
- File Name and Sub-circuit Name •
- SPICE Netlist File Content •

#### 1.3.2.1.1 **Block Name**

Change the **Block Name** in this field.

Block Name: RX\_PRIMARY

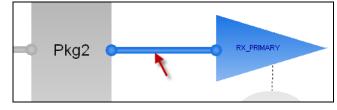
#### 1.3.2.1.2

#### **Connection List**

This pane displays the connections between the blocks.

Conn. Port	Connect To	Block Name	Conn. Port
rx_pkg	••	Pkg2	DIE

The connection listed above shows the **rx\_pkg** connects to the **Pkg2** block at the **DIE** side.



You can define or manage the connections in the Connection Definition tab.

To open the **Connection Definition** tab, click on the **button** or double-click on the highlighted connection. The Connection Definition tab opens. The connection names and connection nodes are displayed. Any broken connections are indicated.

#### 1.3.2.1.3 **File Name and Sub-circuit Name**

This section specifies the SPICE netlist file name and the sub-circuit model associated with the block.

File Name: D	D:\Working\SystemSI\sc_example1\rx_bhvr.sp		Sub-circuit Name:	nmos_diff_rx	-	
--------------	--	--	-------------------	--------------	---	--

By default, the Single Channel Template has only one sub-circuit model specified for each block. You can choose your SPICE netlist file. Your file might contain several sub-circuit models.

#### 1.3.2.1.4 SPICE Netlist File Content

Please refer to Section 1.3.2.1.4 SPICE Netlist File Content for details.

#### 1.3.2.2 Jitter & Noise Tab

The Jitter & Noise tab has two sections: Jitter and Noise.

Jitter			Noise		
Random (Rj):	1	%UI RMS	Random (Rn):	1	mV RMS
Deterministic (Dj):	1	%UI peak	Deterministic (Dn):	1	mV peak
Note: These parameters a	are post-processe	ed into the eye	distribution associated wit	h the Rx.	
Restore Defaults					

#### • Random Jitter (Rj) (%)

Jitter has not been bounded. Random jitter is described by a Gaussian probability distribution, characterized by its standard deviation (RMS) value. This type of jitter is caused by thermal noise or other random noise effects in the system. Default value is 1% of the bit time.

• Deterministic Jitter (Dj) (%)

Jitter with a non-Gaussian probability density function. Jitter is always bounded in amplitude and with specific causes. Default value is 1% of the bit time.

• Random Noise (Rn) (mV)

Caused by random fluctuations in signal voltage. Default value is 1 mV.

• Deterministic Noise (Dn) (mV)

Comes from power supply. Can have many sources such as capacitive and inductive coupling. Default value is 1 mV.

**Rj**, **Dj**, **Rn** and **Dn** are all post-processed jitter and noise. They show up in the Bathtub curve, which is a cumulative distribution function. Other types of deterministic jitter are added to the Transmitter Bit Stream, such as:

- periodic jitter
- DCD

• Duty Cycle Distortion.

#### Restore Defaults

For regular templates, Selecting this button resets all field values to the product defaults.in case of compliance templates, all fields are restored to the value saved for that particular compliance.

#### 1.3.2.3 Power Supply Tab

✓ Ideal Power Supply:	1	V

• **Power Supply** – Ideal voltage source that is used at the transmitter. Default value is 1 V. Please refer to *Section 1.3.1.4 Power Supply Tab* for details.

### 1.3.3 Pkg1

Double-click the **Pkg1** block to open the **Property** pane.

#### The **Property** pane contains two tabs: **Connection** and **Layout Extraction**.

ock Name: Pkg1				File Name: D:\Working\SystemSI\sc Sub-circuit Name: txpkg
Conn. Port DIE BGA	Connect To	Block Name TX_PRIMARY PCB	Conn. Port tx_pkg UI	.subckt txpkg posin negin posout negout ngnd * [MCP Begin] * [Connection] DIE * [Connection Type] * [Power Nets] * [Ground Nets] * 3 ngnd gnd * [Signal Nets] * 1 posin pos * 2 negin neg * [Connection] BGA
				* [Connection Type] * [Power Nets] * [Ground Nets] * 3 ngnd gnd * [Signal Nets] * 1 posout pos Launch MCP Header Editor Edit Sub-circuit Definition

#### 1.3.3.1 Connection Tab

The **Connection** tab contains these parts:

- Block Name
- Connection List
- SPICE Netlist File

#### 1.3.3.1.1

#### Block Name

Block Name: pkg1

Change the **Block Name** in this field.

#### 1.3.3.1.2 Connection List

This pane displays the connections between the blocks.

Conn. Port	Connect To	Block Name	Conn. Port
DIE		TX_PRIMARY	tx_pkg
BGA		PCB	U1

The **Pkg1** block has two connections:

- From **Pkg1** to **TX\_PRIMARY**, through the **DIE** connection.
- From **Pkg1** to **PCB**, through the **BGA** connection.

#### 1.3.3.1.3 SPICE Netlist File

File Name: \sc_example1\tx_p.kg.s Sub-circuit Name: txpkg 🗸				
.subckt txpkg posin negin posout negout ngnd				
* [MCP Begin] * [Connection ] DIE * [Connection Type] Do Not Ed * [Power Nets] * Ground Nets] * I posin pos * [Connection] BGA * [Connectio	used			
жижносконсконсконского жите и политического жите и политического жите и политического жите и политического жите				
**************************************	Here, you can add your own spice subckt that can include s-parameter models and/or			
*S0100.5041234 hgnd *S11234 ngnd mname=s_model_p1 *.model s_model_p1 s tstonefile="./channe *.ends sp4	el_bga.			

#### 1.3.3.2 Layout Extraction Tab

File Name:	]
Extraction Engine:	💌 📖 🗆 Launch
Command-line Switches:	

### 1.3.4 Pkg2

The difference between **Pkg1** and **Pkg2** is in the connectivity and the model name. Both use the same package model.

Double-click on the **Pkg2** block to open the **Property** pane. The **Property** pane opens and displays the Pkg2 tabs.

The single channel template used here has dummy models for **Pkg1** and **Pkg2** blocks. So there is a short circuit between **PCB** and **TX\_PRIMARY** blocks, and **PCB** and **RX\_PRIMARY** blocks.

# 1.3.5 PCB

Double-click on the PCB Block to open the Property pane:

Conn. Port					File Name:         D:\ASI166\1664\SLA\zz\chain.sp         Sub-circuit Name:         channel           S Parameter File:         D:\ASI166\1664\SLA\zz\chain.sp          Sub-circuit Name:         channel
	Connect To		Conn. Port		S Parameter File: D:\ASI166\1664\SLA\zz\xaui_ref.s4p View S Paramete
U1		Pkg1	BGA	Edit Layout Linkage	.subckt channel posin negin posout negout ngnd
U2	••	Pkg2	BGA	Edit Layout Linkage	R Dates on which
					* [MCP Begin] * [Connection] U1
					* [Connection Type]
					* [Power Nets] * [Ground Nets]
					* 3 ngnd gnd
					* [Signal Nets]
					* 1 posin pos * 2 negin neg
					* [Connection] U2
					* [Connection Type]
					* [Power Nets] * [Ground Nets]
					· [Ground Nets]

The **Connection** tab contains these parts.

- Block Name
- Connection List
- SPICE Netlist File

#### 1.3.5.1 Block Name

Change the **Block Name** in this field.

Block Name:	PCB

#### 1.3.5.2 Connection List

The **Connection** pane shows the connections between the blocks.

Connection	Connect To	Block	Block Connection
U1	••	Pkg1	BGA
U2	••	Pkg2	BGA

The **PCB** block has two connections:

- From **PCB** to **Pkg1**, through the **BGA** connection.
- From **PCB** to **Pkg2**, through the **BGA** connection.

#### 1.3.5.3 SPICE File List

The model used by the **PCB** block is a 4-port s-parameter file. You can replace the file with:

- Another S-parameter File
- A W-element model
- Other Circuit Elements

File Name: D:\ASI166\1664\SLA\zz\chan.sp Sub-circuit Name: channel
S Parameter File: D:\ASI166\1664\SLA\zz\xaui_ref.s4p View S Parameter
.subckt channel posin negin posout negout ngnd  * [MCP Begin] * [Connection] U1 * [Connection Type] * [Power Nets] * [Ground Nets] * 3 ngnd gnd * [Signal Nets] * 1 posin pos * 2 negin neg * [Connection] U2 * [Connection] U2 * [Connection Type]
* [Power Nets]     * [Ground Nets]      Extract BBS Model Load BBS Model Launch MCP Header Editor Edit Sub-circuit Definition

#### • View S parameter

Select this button to review the s-parameters for a block in the s-parameter viewer window.

#### Extract BBS Model

Select this to launch Broadband Spice and generate a BBS model for the S Parameter or Touchstone file

#### Load BBS Model

For subcircuits with an S Parameter model, use this button to load the BBS model (generated by Broadband spice for the S Parameter or Touchstone file).

#### • Launch MCP Header Editor

Select this to invoke the MCP Header Editor. If the Touchstone file or the BNP files do not have the MCP header information, you need to manually add it in MCP Header Editor.

#### • Edit sub-circuit Definition

Select this to open the .sp file in sub-circuit Definition Editor for editing.

### 1.3.6 AMI

AMI stands for Algorithmic Modeling Interface. It is designed to model advanced Serializer/Deserializer (SERDES) devices. All models contain complex signal processing routines that are compiled into separate executables (DLL files) that are called by the channel engine. AMI contains three functions:

- AMI\_Init
- AMI\_GetWave
- AMI\_Close
- SystemSI Serial Link Analysis first characterizes the channel by means of an impulse or step response, and then calls the AMI DLL files to apply equalization to the channel.

#### **Related Topic**

• **R**efer to: <u>http://www.vhdl.org/pub/ibis/birds/</u>

### 1.3.6.1 Transmitter AMI

To open the **Property** pane, double-click on the **AMI** block connected to the transmitter.

Property	
Enable AMI Parameter File: C:\Cadence\SPB_16.6\ASI\Base\SpeedXP\Library\template\SystemSI\SerialLink\amiwin\amiffe.ami Reset AMI dll File: C:\Cadence\SPB_16.6\ASI\Base\SpeedXP\Library\template\SystemSI\SerialLink\amiwin\amiffe.dll Reset	AMI Parameter:     Reload AMI File       ( amiffe     ( fwd 3 )       ( pre 1 )     ( coeffout nil )       ( UserTapsFile nil )       )
AMI	
	OK Capcel (pply

The **AMI** property pane contains two sections:

- AMI parameter and DLL file locations
- AMI parameter list

The AMI parameter list contains two tabs:

- Model Specific
- Reserved Parameters

#### 1.3.6.1.1

#### AMI Parameter and DLL File Locations

Enable	
AMI Parameter File:	
C:\Program Files\Sigrity\SpeedXP 12.0\Library\template\SystemSI\SerialLink\amiwin\amiffe.ami	Reset
AMI dll File:	
C:\Program Files\Sigrity\SpeedXP 12.0\Library\template\SystemSI\SerialLink\amiwin\amiffe.dll	Reset

- **AMI parameter file** Contains all AMI parameters such as **Forward** tab, **Precursor** tab and tap **Coefficient**, as well as usage definitions of those parameters.
- **AMI dll file** Library file represents the AMI model in an executable format and is called by the channel engine.

NOTE!	When installing the Channel Analysis tool, a template directory is created with all the template files, as well as a new folder called <b>amiwin</b> . That folder contains all the AMI models.
NOIE:	amiwin. That folder contains all the AMI models.

AMI Parameter:	Reload AMI File
(amiffe (fwd <u>3</u> ) (pre <u>1</u> ) (coeffout <u>nil</u> ) (UserTapsFile <u>nil</u> ))	
Reserved Parameters Model Spec	cific

1.3.6.1.3

#### **AMI Parameter List – Reserved Parameters**

AMI Parameter:	Reload AMI File
( amiffe ( Max_Init_Aggressors ( Init_Returns_Impulse ( GetWave_Exists Fa)	<u>25</u> ) <u>True</u> )
Reserved Parameters Model S	ipecific

Generally, you can edit the AMI parameters from the AMI Parameter Editor as shown below.

AMI Parameter Editor	×
3	
Usage: In	
Type: Integer Format: Range 31128	
Default: 3	
Description: Number of FFE taps	
	OK Cancel

Some reserved parameters are not editable or supported in SystemSI.

• If you try to edit the following reserved parameters Max\_Init\_Aggressors, Init\_Returns\_Impulse, or GetWave\_Exists, the values are not editable in the AMI Parameter Editor dialog box.

A	MI Parameter Editor	х
	25	
	Usage: Info Type: Integer Format: Default: 25 Description: Number of aggressors is actually unlimited.	

If you try to edit TrainOn, the following message will be issued.

AMI Parameter: Reload A	MI File
( amictwf_pcieg3_bc ( Ignore_Bits 40000 ) ( Max_Init_Aggressors 25 ) ( Init_Returns_Imput ( GetWave_Exists T ( TrainOn 0 ) ( Train <u>C:\Progr</u> ) ) TrainOn is not editable. To turn back-channel functionality on or off, please go to the menu "Setup->AMI Options->Back-Channel".	e Pr
ОК	
	•
Reserved Parameters Model Specific	

• The reserved parameters Tx\_Jitter and Rx\_Clock\_PDF are not supported by SystemSI, as post BIRD 123 they are obsolete.

If an AMI file has any of these parameters, a message will be issued when it is loaded to SystemSI.



#### 1.3.6.2 Receiver AMI

To open the **Property** pane, double-click on the **AMI** block connected to the receiver.

Property	, ,
Chable AMI Parameter File: C:\Cadence\SPB_16.6\ASI\Base\SpeedXP\Library\template\SystemSI\SerialLink\amiwin\amidfe.ami Reset AMI dll File: C:\Cadence\SPB_16.6\ASI\Base\SpeedXP\Library\template\SystemSI\SerialLink\amiwin\amidfe2.dll Reset	AMI Parameter:       Reload AMI File         ( amidfe       ( bwd 5 )         ( res 64 )       ( foffset 0 )         ( foffset 0 )       ( meas_delay 50e-09 )         ( coeffout dfe.txt )       )
AMI	
	OK Cancel Apply

The **AMI** property pane contains two sections:

- AMI parameter and DLL file locations
- AMI parameter list

The AMI parameter list contains two tabs:

- Model Specific
- Reserved Parameters

#### 1.3.6.2.1

#### AMI parameter and DLL File Locations

Enable	
AMI Parameter File:	
C:\Program Files\Sigrity\SpeedXP 12.0\Library\template\SystemSI\SerialLink\amiwin\amidfe.ami	Reset
AMI dll File:	
C:\Program Files\Sigrity\SpeedXP 12.0\Library\template\SystemSI\SerialLink\amiwin\amidfe2.dll	Reset

#### 1.3.6.2.2

#### AMI Parameter List – Model Specific

AMI Parameter:	Reload AMI File					
( amidfe						
(bwd <u>5</u> )						
( res <u>64</u> )	(res <mark>64</mark> )					
(foffset <mark>0</mark> )						
( meas_delay <u>50e-09</u> )						
(coeffout dfe.txt)						
) .						
Reserved Parameters Mod	el Specific					



#### AMI Parameter List – Reserved Parameters

AMI Parameter:	Reload AMI File
(amidfe (Ignore_Bits <u>4</u> (Init_Returns_I (GetWave_Exis )	mpulse False)
<b>Reserved Parameters</b>	Model Specific

Please refer to Section 1.3.6.1.3 AMI Parameter List - Reserved Parameters for details.

#### 1.3.6.3 AMI Options

You can choose Setup > AMI Options to check the AMI options.

SystemSI - [Serial Link Analysis : D:\Sigrity\Channel_Test\pcie3\pcie3.ssix]				
₩ File Edit View	Setup Tools Compliance Window	Help		
🗋 💕 🛃 🤟 🔻 (* 🔻		s 🖉 🗙 🐚 🖽 🖽 🖽 🗹 🔽 🐚		
Workflow: SystemSI	Terminate Unconnected Nodes			
Serial Link Analysis Pause before Simulation				
ourier anne renergoto	Hide Channel Simulator Messages			
SFP+ Compliance	Sweep Mode			
HDMI Compliance	Probe Point			
	AMI Options	Jitter & Noise of Tx and Rx Blocks: On		
PCIe 3 Compliance	~	Back-Channel: Off		

- If any AMI files has any Jitter and Noise parameters (as listed below) as the Reserved Parameters, AMI Options > Jitter & Noise of Tx and Rx Blocks can be turned on or off. Otherwise, it is turned on and grayed out.
  - Tx\_Dj
  - Tx\_DCD
  - Tx\_Rj
  - Tx\_Sj
  - Tx\_Sj\_frequency
  - Rx\_Rj
  - Rx\_Dj
  - Rx\_Sj
  - Rx\_DCD
  - Rx\_Clock\_Recovery\_Mean
  - Rx\_Clock\_Recovery\_Rj
  - Rx\_Clock\_Recovery\_Dj
  - Rx\_Clock\_Recovery\_Sj
  - Rx\_Clock\_Recovery\_DCD
  - Rx\_Noise

- If any AMI files has the Reserved Parameter TrainOn, AMI Options > Back-Channel can be turned on or off. Otherwise, it is turned off and grayed out.
- When loading an AMI file, if the AMI has any Jitter and Noise parameters (as listed above) as Reserved Parameters, and AMI Options > Jitter & Noise of Tx and Rx Blocks is turned on, a dialog will be issued asking a question.

SystemSI	
?	This AMI file 'amiffe_tx_jitter.ami' contains Reserved_Parameters for jitter or noise injection. Similar items are additionally enabled in the Tx and Rx blocks. Would you like to turn those items off in the Tx and Rx blocks to avoid double-counting? Note: You can also turn those items on or off from the menu 'Setup->AMI Options->Jitter & Noise of Tx and Rx Blocks'.
	<u>Y</u> es <u>N</u> o

If you click the **Yes** button, AMI Options > Jitter & Noise of Tx and Rx Blocks will be turned off automatically.

# 1.4 Analysis Options

To open the Analysis Options interface, start in the Setup menu and click Analysis Options....

Setup		Tools	Compliance	Window		
	Analysis Options					
	Terminate Unconnected Nodes					
	Pause before Simulation					
	Hide Channel Simulator Messages			Messages		
	Sva	/eep Mo	de			
	Probe Point					
	A٨	/I Optio	ns	•		

The Analysis Options pane opens.

Analysis Options	
Simulator Circuit Simulator O HSPICE SPDSIM Characterization Duratiof 30 r Vmea	Analysis Setup Ignore Time: 5000 ns # of Bits: 200000 Bit Sampling Rate: 32
Circuit Simulator Options Channel Sin	Slow     Fast/Slow     Slow/Fast     Slow/Fast     Wrbare scale (eye width)     Wrbare scale (eye bainth)
Channel Simulator OS: Windows 64 Bit	Image: Statistical LBERs:       -12
Restore Defaults	III OK Cancel App

The Analysis Options interface has two parts:

Simulator

It includes Circuit Simulator, Circuit Simulator Options/Channel Simulator Controls, Channel Simulator OS, and Simulation Name.

Analysis Setup

It includes **Ignore Time**, **# of Bits**, **Bit Sampling Rate**, **BER Floor**, **Simulation Configuration**, **XTalk**, **Eye Distribution**, **BER\_Eyes**, and **IBIS Model Selection**.

### 1.4.1 Simulator

#### 1.4.1.1 Circuit Simulator

-Circuit Simula	tor		
○ HSPICE			
SPDSIM     Characteriz     Duration:		Vmeas:	

You can specify the circuit simulator that will be used in the Time-Domain Simulation to characterize the channel via the step response.

There are two simulators available:

Circuit Simula	tor
SPDSIM	

- Specify the path for HSPICE executable file in the **HSPICE** text field.
- SPDSIM is a built-in circuit simulator. It is a SPEED2000 simulator.

#### For the **Characterization** part:

- **Duration** refers to the duration of the Characterization run with the specified Circuit Simulator. The Characterization should be run long enough to allow any reflections to settle out, and the waveforms reach their steady state.
- $V_{meas}$  refers to the voltage threshold at which delay is measured from the Characterization. This information is included in the Channel Report as Delay. If  $V_{meas}$  value is not explicitly called out in an IBIS file associated with the Tx block,  $V_{meas}$  is taken as the midpoint of the voltage swing seen in the Characterization waveform.

#### 1.4.1.2 Circuit Simulator Options/Channel Simulator Controls

#### 1.4.1.2.1 Circuit Simulator Options

Circuit Simulator Options	Channel Simulator Controls
* Add global .option and .includ * They'll be used for time domai *.option delmax=1p	

Add the global **.option** and **.include** commands. These commands can be used in the Time Domain characterization.

For HSPICE simulation accurate characterization usually requires the .option delmax command. It sets the maximum allowable step size of the time steps taken during Transient analysis.

.option delmax=1p

or

.option delmax=2p.

.option delmax

For this exercise, you can set

.option delmax=2p.

**NOTE!** Do not run the simulation using HSPICE without specifying the **delmax** option. Setting this option increases the simulation time.

#### 1.4.1.2.2

#### **Channel Simulator Controls**

Circuit Simulator Options	Channel Simulator Controls	
* Add global controls for ti useblkfit 2000	he Channel Simulator.	

This field enables expert users to pass specific controls to the Channel Simulator. These controls will be reflected in the simulation results.

Following are some of the options which can be input in the Channel Simulator Controls field.

Format: <command\_name> <value(s)>

• AMI Model Controls

These options address deficiencies in AMI models.

– useblkflt

Format: useblkflt <number>

**Description:** Controls the getwave size. The getwave blocks will be made of <number> bits.

- ignoreamiclk

Format: ignoreamiclk

Description: Ignores the clock vector returned by the AMI model.

NOTE:You should avoid using these controls.<br/>Use them only if the AMI model is not IBIS compliant and not robust.

Cheap Probing

probealleyes

Format: probealleyes

**Description:** Normally the eye contour is ISI only eye contour at the Rx output. You can also output ISI only eye contours at the following additional points:

- o Tx input
- o Tx output (if the Tx has an AMI model and the model uses getwave)
- Rx input

The files will be in standard tab delimited ASCII format and will be named as:

- eyectr\_in.txt (Rx input)
- eyectr\_tx.txt (Tx input and output only if Tx AMI model with getwave is available)

- o eyectr\_tx\_in.txt
- User Supplied Step Responses (from 13.0 only)

The channel characterization can be overwritten with a user supplied step response file.

The syntax is more complex.

- Format:

```
impfile (<tx_id>
<path_to_user_supplied_rx_step_response_file> (type step))
```

#### **Description:**

<tx\_id> is the name of the Tx block. It should be the same name as that appears in the **command.txt** file.

<user\_supplied\_rx\_step\_response\_file> is a two-column step response
file. The first column is time, and the second column is the step response.

Waveforms

By default, 1000 bits of waveforms are output to **waveform.txt**. This output is controlled by the directive wavecnt.

You can modify the number of bits by passing these as a parameter. The command syntax in that case is wavecnt <#\_bits> instead of just wavecnt.

In addition to the Rx output, you can also continue on the theme of cheap probing by modifying wavecnt to output waveforms at Rx input, Tx input and Tx output by specifying these as parameters. The command is:

```
wavecnt (tx)(txin) (rxin).
```

The corresponding waveforms are available in the ASCII tab delimited files **chan2\_tx.txt**, **chan2\_tx\_in.txt**, and **chan2\_in.txt**, respectively.

Tx input and output waveforms will be available only if Tx has a getwave type of AMI model.

• Tx Bits

To save Tx bits as output, use the command output\_txbits.

The Tx bits are stored in a two-column (time and voltage) ASCII file, *srcbit\_<tx\_id>.txt*.

• Changing ISI Only Eye Contour

By default, the eye contour is ISI only eye (as opposed to BER eye).

You can modify the eye contour to a BER eye by using the following directive:

- Format: eyectr\_nber 1e-12

**Description:** The eye contour will now be 1e-12 nBER eye.

- Format: eyectr\_ber 1e-12

**Description:** The eye contour will now be 1e-12 BER eye.

- Format: eyectr\_jnber 1e-12

**Description:** The eye contour will now be 1e-12 BER eye.

#### 1.4.1.3 Windows 32 Bit AMI DLL Support in a Windows 64 Bit SystemSI Installation

This feature is for 64 bit Windows SystemSI - Serial Link Analysis installations.

When running channel simulations, it is necessary that the channel simulator (spdut.exe) is compiled for the same Operating System (OS) as the algorithmic models (AMI DLLs) that are being used. As it is fairly common for SerDes suppliers to distribute win32 AMI DLLs, and fairly common for you to install win64 software on their computers, this feature has been added to enable compatibility in these scenarios.

#### You can find a folder **win32** under the

<INSTALL\_DIR>\SpeedXP\Library\template\SystemSI\SerialLink\amiwin folder.

- The amiwin folder contains the 64 bit AMI parameter files and DLL files
- The win32 folder contains the 32 bit Windows AMI DLLs and channel simulator

ary 🕨 template 🕨 SystemSI 🕨 S	SerialLink ▶ amiwin ▶ win32		
		-	
Jame	Date modified	Туре	Size
🗟 amicdr2.dll	10/23/2012 7:49 AM	Application extens	23 KB
🚳 amictwf.dll	10/23/2012 7:49 AM	Application extens	4,005 KB
🚳 amictwfadapt.dll	10/23/2012 7:49 AM	Application extens	4,530 KB
🚳 amidfe2.dll	10/23/2012 7:49 AM	Application extens	32 KB
🚳 amidfenl.dll	10/23/2012 7:49 AM	Application extens	3,895 KB
🚳 amiffe.dll	10/23/2012 7:49 AM	Application extens	4,558 KB
🚳 amirx.dll	10/23/2012 7:49 AM	Application extens	4,575 KB
spdut.exe	10/23/2012 7:50 AM	Application	2,560 KB

A new option, **Channel Simulator OS**, is added to the **Analysis Options** window for Windows 64 bit SystemSI – Serial Link Analysis.

Circuit Simulator		
HSPICE		
SPDSIM		
Characterization	n	
Duration: 30	ns Vmeas:	
Add global .optio	n and include commands her	re.
Add global .optio They'll be used fo	n and .include commands her or time domain characterization	re.
Add global .optio They'll be used fo	n and .include commands her or time domain characterization	re.
Add global .optio They'll be used fo	n and .include commands her or time domain characterization	re.
Add global .optio	n and .include commands her or time domain characterization	re.
Add global .optio They'll be used fo .option delmax=1	n and .include commands her or time domain characterization	re.
Add global .optio They'll be used fo option delmax=1	n and .include commands her or time domain characterizati P	re.

It has two options:

- Windows 64 Bit
- Windows 32 Bit

The default is **Windows 64 Bit**. If the AMI models you are using are 32 bit DLLs, you should select **Windows 32 Bit**.

If your DLLs do not match the selected Channel Simulator OS, the channel simulation aborts, and following message is displayed.

SystemSI	
?	The channel simulation is aborted. Do you want to open the spdut.log file?
	Yes No

Click Yes to view the spdut.log file. It provides information about the nature of the error.

	SystemSI installation has both, win64 and win32, versions of the AMI models.
NOTE!	Based on the value specified in the Channel Simulator OS field, appropriate
	version of the DLLs and channel simulator are used.

#### 1.4.1.4 Simulation Name

Define Simulation Result Name.

-Simulation Name Automatic 
 Custom

- By default, the **Automatic** option is selected. In this case, the result folder names are automatically defined according to the simulation times.
- If you select the **Custom option**, you need to specify the simulation name before the start of the simulation.

When you click the **Play** button , the **Simulation Name** window pops up. Enter the name of result folder.

Simulation Name		×
Please assign a simulation name:	OK Cancel	

## 1.4.2 Analysis Setup

1.4.2.1	General Channel Sin	nulation Parameters
---------	---------------------	---------------------

Ignore Time: 5000	ns	# of Bits:	200000	Bit Sampling Rate:	32	BER Floor:	1e-20
-------------------	----	------------	--------	--------------------	----	------------	-------

#### • Ignore Time (ns)

Specifies the initial time to be ignored from the waveform, so that the data is not corrupted with the startup time transients.

Default value is 5000 ns. You can use a lower value such as 100 ns if you do not use adaptive equalizers like adaptive DFE.

#### • Number of Bits

Specifies how many bits to simulate. The default value is 200,000.

For BER computation you need to simulate at least 100,000 bits. If you are doing crosstalk simulation, you may need to simulate more bits like 200,000.

#### • Bit Sampling Rate

Similar to time step in circuit simulation, which is used by the channel simulation engine. The larger the number is, the longer the simulation time is. Default value is 32 samples/bit. The default is sufficient for most cases.

#### BER Floor

Specifies the minimum Bit Error Rate or **BER** to be used in the simulation. The value can go up to 1e-20. Default value is 1e-20.

#### 1.4.2.2 Simulation Configuration

Simulation Configuration			
🗹 Ideal Power			
-Corner			
Fast			
🗹 Тур			
Slow			
Fast/Slow			
Slow/Fast			

- The **Corner** settings are for the IBIS block and the VRM block. It will be available for selection only when the design has an IBIS block or a VRM block.
- The **Ideal Power** setup is for the VRM block. It will be checked and grayed out if the project does not have any VRM block.

#### 1.4.2.3 Xtalk

– 🔜 XTalk——	
🖲 Odd	OEven
ORandom	O Statistical

**Xtalk** is not available in Single Channel Analysis. For the details of this part, please refer to *Section 2.4.2 Xtalk* in Crosstalk Channel Analysis.

#### 1.4.2.4 Eye Distribution Methods

You can select an eye distribution method from this dialog. The default **Eye Distribution** is **Time Domain Waveform (Channel Simulator)**.

e end n
Eye Distribution
Method
neulou
<ul> <li>Time Domain Waveform (Channel Simulator)</li> </ul>
O Statistical
O Statistical

Select **Statistical** to enable the eye distribution to be calculated statistically.

Eye Distribution
Eye Distribution
r Method
Healou -
<ul> <li>Time Domain Waveform (Channel Simulator)</li> </ul>
O She Ke Ke al
<ul> <li>Statistical</li> </ul>
Sector Contract of

If you select Statistical, in the next step BER\_Eye generation is automatically selected by default.

#### 1.4.2.5 Statistical Eye Contours

You can enable **BER\_Eye generation** to generate statistical eye contours. If enabled, **BER\_Eyes** are generated and displayed. Choose a criterion from:

- Time scale (eye width)
- Voltage scale (eye height)
- Both time and voltage

BER_Eyes
BER_Eye generation
Criteria
Time scale (eye width)
Voltage scale (eye height)
Both time and voltage
LBERs: -12

By default, **Both time and voltage** is checked when **BER\_Eye generation** is enabled.

After you select a criterion, the BER eye will be generated based on the criterion.

If you place the cursor in the **LBERs** field, an indication box appears. The LBER values are displayed.

LBERs: -12							
	LBER should b a single LBER,						
	Several LBERs	, ex6, -9,-12					
	Range of LBER	Rs, ex12 to -1	.5				
All LBERs should be negative integer.							
		ОК	Cancel				

**NOTE!** All LBERs should be negative integers. Set the LBER values within the -3 to -20 range.

### 1.4.2.6 IBIS Model Selection

If any AMI model is included, it will be listed in the **IBIS Model Selection** part, as the example shown below.

IBIS Model Selection				
Tx2 <b>Rx2</b>				
Signal Name	IBIS File	Component	Receive IO Model	Status
ami_rx_p	test_ibis.ibs	test_serdes	test_ami_rx1	Signal
ami_rx_n	test_ibis.ibs	test_serdes	test_ami_rx1 🛛 🔻	Signal
			test_ami_rx1	1
			test_ami_rx2	ļ
•		111		

# 1.5 The First Simulation

Any new channel needs to be characterized before running a channel simulation. Characterization of the channel means finding the step or impulse response. Run either HSPICE or SPDSIM simulation to characterize the channel.

The **standard\_step.sp** file is installed in the main programs directory under

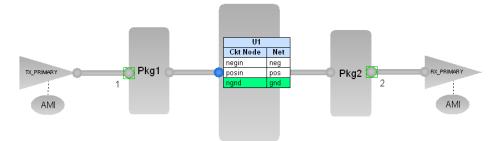
<INSTALL\_DIR>\SpeedXP\Library\template\SystemSI\Serial Link Analysis

This file is a standard stimulus sub-circuit used by the tool to generate the step response. All the characterization information for the channel is stored in the Workspace directory.

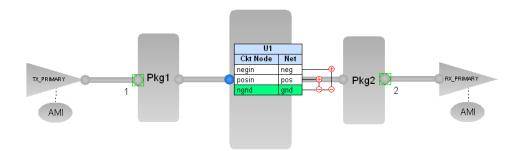
NOTE!Do not edit the standard\_step.sp file unless you are an<br/>advanced user and want to set up a different stimulus.

# 1.5.1 Setting up Probe Point

1. Click the connection point to open the **Ckt Node** menu.

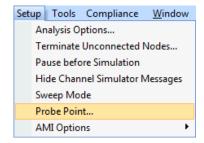


2. To set Probe Point for the simulation, click two Ckt Nodes in the menu.



- The two selected nodes are connected.
- The node clicked first is defined as positive, and the second is negative.
- 3. To view and edit the defined Probe Points, select

Setup> Probe Point....



The Probe Point window opens below.

lock Name	MCP Connection	Positive Ckt Node	Negative Ckt N		
PCB	U1	posin	ngnd		
PCB	U1	negin	ngnd		

4. To show or delete the defined Probe Points, right-click and select from the pop-up menu.

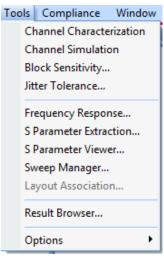
ock Name MCP Connection	Positive Ckt Node	Negative Ck	t N	
IB <b>U1</b>	posin	ngnd		
28 UI	negin		Show All Highlighted Items Show All Lide All Highlighted Items Hide All Delete All Highlighted Items Delete All	

5. After the simulation, the results of all the defined Probe Points can be viewed from the **2D Curve (Eye Contour)** window, **3D Eye Density** window and **Report View** window.

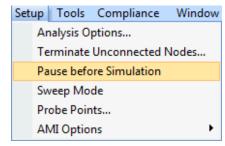
Image: Control       Image: Contro       Image: Control       Image:	💄 3D Eye Density 💶 🗙	2D Curve (Eye Contour)	×	Report View	×
For Contour For Contou		🛶 🗰 🔹 🗶 🖷 🗂 🕅 🔿		Report	Channel R 🔺
<ul> <li>Tesult (Leydensity, CdC [posin,ngnd], bt ] result (Leydensity, PCG [posin,ngnd], bt ] result (Leyd</li></ul>	📄 📕 🜍 🌲 🛣 🛅 😇		<none> ▼ 👸</none>	result\1\report_PCB[posin,n	Mon Jul 16 10:26:32
Oreskill levedensky. DCE[oosin,ngnd].tt • reskill levedensky. DCE[oosin,ngnd].tt • Teskill levedensky. DCE[oo		result) 1) eventr. cur	1/-H 0.0	resulting operations and	General:
-0.1 -0.2 -0.3 -0.4 -0.2 -0.3 -0.4 -0.4 -0.2 -0.3 -0.4 -0.4 -0.2 -0.3 -0.4 -0.4 -0.2 -0.3 -0.4 -0.4 -0.2 -0.3 -0.4 -0.1 -0.2 -0.3 -0.4 -0.2 -0.3 -0.4 -0.2 -0.3 -0.4 -0.2 -0.4 -0.2 -0.3 -0.4 -0.2 -0.4 -0.2 -0.3 -0.4 -0.2 -0.4 -0.2 -0.3 -0.4 -0.2 -0.4 -0.4 -0.4 -0.2 -0.4 -0.4 -0.4 -0.4 -0.4 -0.4 -0.4 -0.4	<ul> <li>result\1\eyedensity_PCB[posin,ngnd].txt</li> <li>result\1\eyedensity_PCB[regin,ngnd].txt</li> <li>30 Bathtub</li> <li>result\1\bathtub_3d.txt</li> <li>result\1\bathtub_3d.txt</li> <li>result\1\bathtub_3d.txt</li> </ul>	result\1\eyectr_PCB[posin,ngnd].cur	0.4		Data Rate Number of B Measurement Channel Cod Primary Dri Data Patter Number of A Characteriz
UI - 0 45			-0.2 -0.3 -0.4		Random Jitt
EVe Heldi			_		Eye Measurements: Eye height

# 1.5.2 Begin Running the Simulation

- 1. Click the **Start Simulation** button **b** to start a simulation.
- 2. If the channel has been characterized before, the characterization is skipped and the channel simulator is invoked directly.
- 3. On the Tools menu, click **Channel Characterization** to characterize the channel with only running the simulation.



4. On the **Setup** menu, click **Pause before Simulation**.



5. If you have enabled **Pause before Simulation** from the **Setup** menu, a message will prompt you whenever a new simulation is about to be run.

### 1.5.3 Simulation Directory

Each simulation creates a unique directory (1, 2, 3 ... etc) with all the waveform files under: \sc\_example1\result.

By default, these directories are moved to the **history** folder when opening an existing workspace file. However, you can make the following setup by choosing **Tools** > **Options** > **Edit Options...**, and clicking **Result** under **Simulation**.

Options		×
Simulation © General Result	Change the 'Result' options in Serial Link	
Advanced AMI Options           AMI Builder            General	Perform the following operation when openning an existing workspace file	
	Move the the previous simulation results under the "result" folder to the "history" folder	
	O Keep the the previous simulation results under the "result" folder	
	O Delete the the previous simulation results under the "result" folder	
	Default Apply OK Cano	:el

The first one is checked by default.

The next illustration shows some of the waveforms generated by the channel engine. This illustration is the default display. To show the waveforms, select

Tile

or

Cascade

On the Windows menu, click To Default to arrange all open windows to the default display.

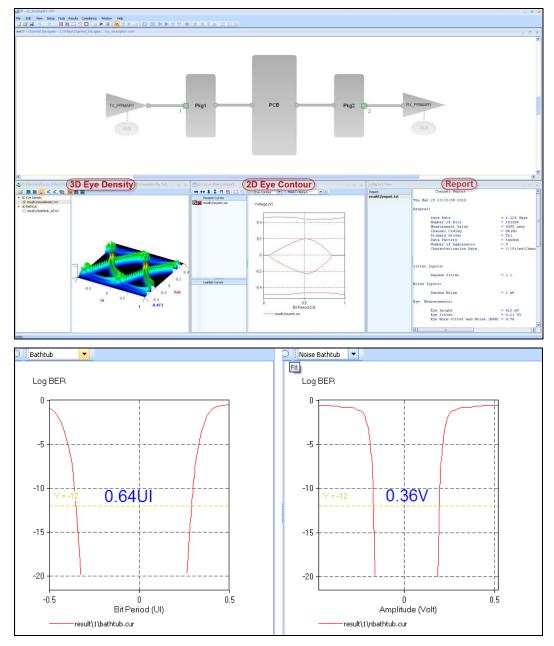
Basically, the .ssix, .html and .txt windows should be put together at the top, and other curve windows should be put together at the bottom.

The waveforms in the **2D Curve** window are the most useful ones. The eye contour and bathtubs are very good measures of the quality of any channel.

- **Bathtub Curves** Provide the eye opening (in % UI) at a specific BER.
- Noise Bathtub Curves Provides the eye height (in Volt) at a specific BER.

Running the channel simulation at TX data rate of 3.125 Gbps produces waveforms with an eye opening of 0.64 UI at BER 1e-12 and an eye height of 360 mV at BER 1e-12.





### 1.5.4 Simulation Results

The simulation results present the following waveforms:

- 3D
  - Eye Density (Default)
  - Bathtub
- 2D

- Eye Contour (Default)
- Bathtub
- Noise Bathtub
- Ramp Response
- Impulse Response
- Modified Impulse Response
- Channel report (Default)

# 1.5.5 Run the Simulation Again

If you click the **Start Simulation** button  $\blacktriangleright$  to run the simulation again without making any changes to the channel components, the characterization of the channel is skipped. Only the channel simulation is run.

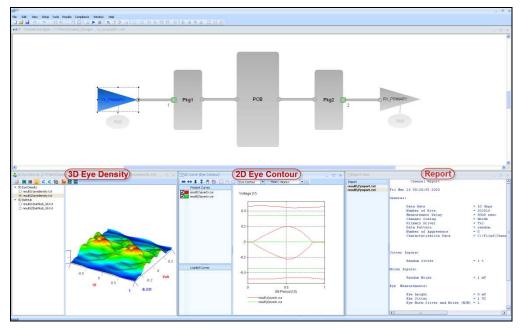
Changes to the channel could mean:

- Changing the parameters for **Pkg1**, **PCB** or **Pkg2** blocks, by modifying the connections, or using a different sub-circuit model.
- Changing **Tx** or **Rx** parameters, such as drive level or **c\_comp** (for Tx).
- Adding additional components to the channel.
- Deleting any component from the channel.

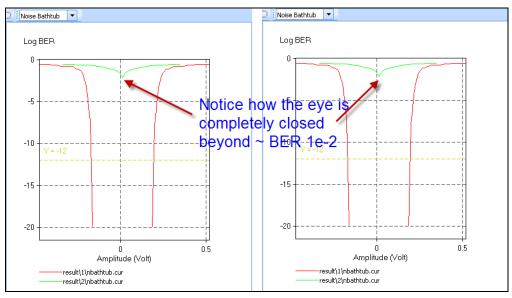
# 1.6 Increasing Data Rate

This section describes how to run the simulation with an increased TX data rate, which would close the eye much more.

- 1. Enable the **AMI** model (a standard **FFE** model) at the transmitter. Notice how the eye improves.
- 2. Change the **TX** data rate to 10 Gbps
- 3. Rerun the simulation. The results are as shown below:



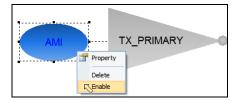
- 4. Notice how the eye (green curve) is completely closed.
- 5. Check the bathtub curves.



# 1.7

# Enable AMI

- 1. Right-click on the **AMI** block.
- 2. Click **Enable** to enable the **AMI** block at the transmitter.



3. Double-click on the AMI block to view its properties. Two files have already been loaded by default:

✓ Enable	
AMI Parameter File:	
C:\Program Files\Sigrity\SpeedXP 12.0\Library\template\SystemSI\SerialLink\amiwin\amiffe.ami	 Reset
AMI dll File:	
$\label{eq:c:Program Files} ignity \ \ \ \ \ \ \ \ \ \ \ \ \ \ \ \ \ \ \$	 Reset

The AMI model assigned to the TX\_PRIMARY block is either:

- AMIFFE
- Feed Forward Equalizer

The parameters are displayed in the AMI parameter window.

AMI Parameter:	Reload AMI File
(amiffe (fwd <u>3</u> ) (pre <u>1</u> ) (coeffout <u>nil</u> ) (UserTapsFile <u>nil</u> )	)
Reserved Parameters Model	Specific

4. Click the default **fwd** parameter to open the **AMI Parameter Editor**.

AMI Parameter Editor	×
3	
Usage: In Type: Integer	
Format: Range 3 1 128 Default: 3	
Description: Number of FFE taps	
	OK Cancel

5. Change the value in the AMI parameter window. The new value means the number of forward tabs has been increased to 4.

AMI Parameter:		Reload AMI File			
( amiffe					
(fwd <u>4</u> )					
(pre <u>1</u> )					
(coeffout <u>nil</u> )					
( UserTapsFile <u>nil</u> )					
)					
Reserved Parameters	Model Spe	rific			

6. Rerun the simulation.

# 1.7.1 Coeffout Parameter

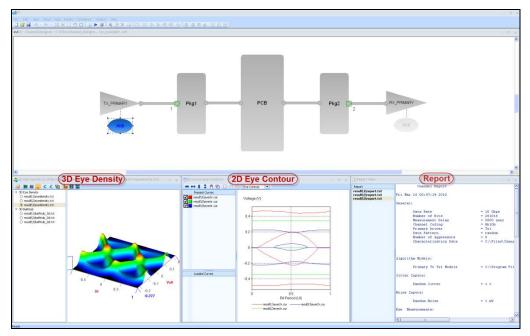
You can put a value for the **coeffout** parameter.

#### Example

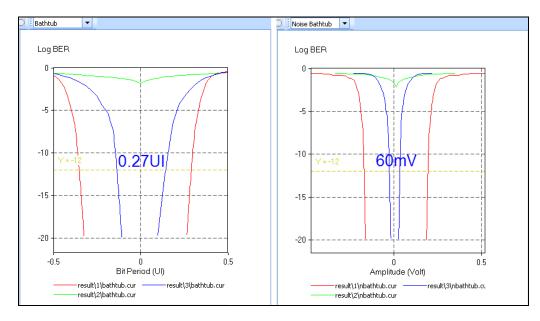
coeffout = coeff.txt.

The FFE coefficients are then output to the file **coeff.txt**. These are the optimized coefficients for this particular channel computed by the **amiffe** model. Based on hardware implementation, the coefficients could be programmed into the real hardware for optimal performance.

The Channel simulation results for data rate = 10 Gbps + TX FFE are shown in the next illustration.



The closed eye is now open (blue curve). The next illustration shows the 2D Bathtub curves for the data rate = 10Gbps + TXFFE. Although the used FFE model does open the eye, the eye width at BER 1e-12 is only 0.27 UI compared to 0.64 UI for data rate of 3.125 Gbps. The eye height is only 60 mV compare to 360 mV.



# 1.7.2 Enabling AMI DFE @ RX\_PRIMARY

- 1. Right-click on the AMI block.
- 2. Click **Enable** to enable the **AMI** block at the receiver.

Enable	
AMI Parameter File:	
$\label{eq:cadence} C:\Cadence\SPB_16.6\ASI\Base\Speed\P\library\template\SystemSI\SerialLink\amiwin\amidfe.ami} amidfe.ami$	Reset
AMI dli File:	
$\label{eq:cond} C:\Cadence\SPB_16.6\ASI\Base\Speed\P\library\template\SystemSI\SerialLink\amiwin\amidfe2.dl$	Reset

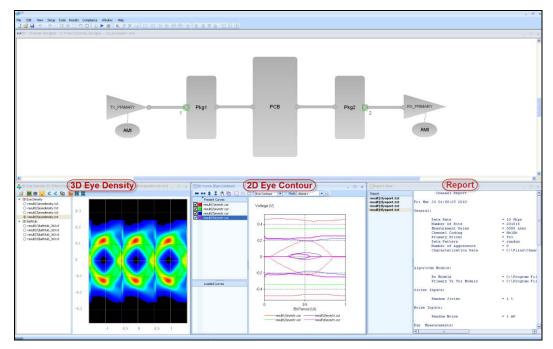
The default AMI model used at the receiver is amidfe2 or Decision Feedback Equalizer.

3. Use the default parameters.

AMI Parameter: Reload AMI					
( amidfe	<u>9</u> )				
Reserved Parameters Model Spec	ific				

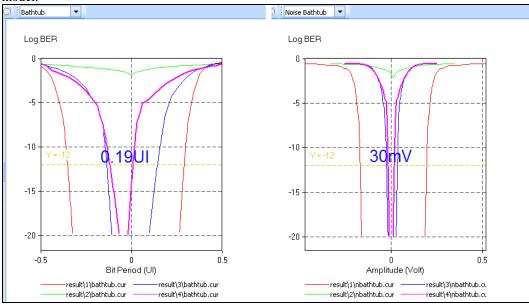
The Channel simulation results for data rate = 10 Gbps + TX FFE + RX DFE are shown in the next illustrations.

As you can see the eye (pink curve) does not open more. It actually closes more than a simulation using **TX FFE** alone.



The 2D Bathtub curves (pink curves).for the data rate = 10 Gbps + TX FFE + RX DFE are shown in the next illustration.

This particular channel using the TX FFE model yields better results than with the RX DFE model.



# 2 Crosstalk Channel Analysis

This chapter describes how to perform a Crosstalk Channel Analysis. You will learn how to use an existing Crosstalk Channel template to demonstrate Crosstalk Channel Analysis. The chapter focuses on how to:

- Assign Models to Components
- Set up the Simulation Parameters
- Set up the Simulation Options
- Run Crosstalk Channel Simulations.

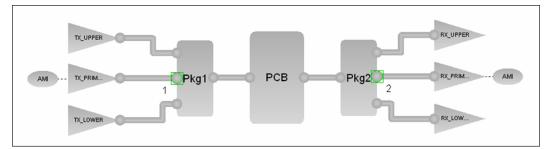
What-if scenarios are used to see how the channel behaves to changes in the data rate as well as the inclusion of Crosstalk and Equalization via **AMI** models.

# 2.1 Overview

The Crosstalk Channel Template consists of three channels. Each channel contains:

- One Transmitter
- One Receiver
- A Printed-Circuit Board(PCB)
- Two Packages

The next illustration shows two **AMI** blocks connected to the primary transmitter and primary receiver. You can add additional blocks and aggressors to modify the template.



# 2.2 Starting the Crosstalk Channel Template

Use the **New Workspace** dialog to create a new workspace. Do not change the contents of any of the template folders. These folders contain the component models, connectivity and settings used to create a new workspace directory.

- 1. Launch SystemSI.
- 2. In the **File** menu. Click **New**.

Alternatively, you can select the **New** button **D**. The **Select Module** dialog opens.

Se	ect Module ×
	Module Name
	Parallel Bus Analysis
	Serial Link Analysis
	Testbench
	OK Cancel

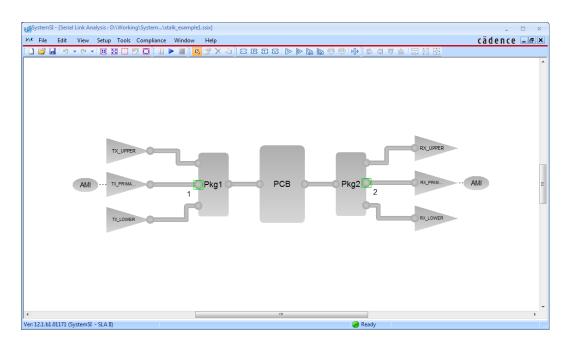
- 3. Select Serial Link Analysis.
- 4. Click OK.

The New Workspace dialog for Crosstalk Channel appears.

New Workspace		×
<ul> <li>Create a blank project</li> <li>Create by template Term</li> </ul>	nplate Path: C:\Cadence\SPB_16.6\ASI\Base\Spee	dXP\Library\template\{
Name	Path	Description
single_channel_complex single_channel_simple sla_simple_em xtalk_channel_simple xtalk_channel_sparam	C:\Cadence\SPB_16.6\ASI\Base\Speed C:\Cadence\SPB_16.6\ASI\Base\Speed C:\Cadence\SPB_16.6\ASI\Base\Speed C:\Cadence\SPB_16.6\ASI\Base\Speed C:\Cadence\SPB_16.6\ASI\Base\Speed	
Name:         xtalk_example1           Location:         D:\Working\System!	JI	
		OK Cancel

- 5. Click Create by template.
- 6. Select **xtalk\_channel\_sparam**.
- 7. Enter a name for the template.
- 8. Enter a location for the template.
- 9. Click **OK**. A directory for the new template is created in the location you entered.

The following illustration shows the newly-created Crosstalk Channel Template workspace **xtalk\_example1.ssix**.



# 2.2.1 Channel Blocks

You can set up one or more block defaults for a new workspace. Do not attempt this if you are not an advanced user. You must pay careful attention to the connections between the blocks.

The Crosstalk Channel Template contains the following blocks for each of the channels:

- A transmitter
- A receiver
- Two packages
- A printed-circuit board (PCB)
- Two AMI models (only for the primary channel)

**NOTE:** The AMI blocks are enabled by default. Disable them first for the following steps of setting.

# 2.3 Examining Each Channel's Component

This section describes each block in the Crosstalk Channel workspace and the properties for each block. The focus is on the parameters and settings that are different from those in the Single Channel Template.

All of the properties for each block are already set in the Crosstalk Channel Template.

**IMPORTANT!** Do not change any block properties in the template.

You can skip this section of the Tutorial, if you wish. This information in this section does not affect the simulation setup procedure.

### 2.3.1 Transmitters

Double-click on a transmitter block to view its properties.

Conn. Port tx_pkg	Connect To	Block Name Pkg1	Conn. Port tx_primary	.subckt nmos_diff_tx pos neg pwr in ngnd + nmos_imp=25 + tx_t=50 + tx_c_comp=1p + tx_scale=1
				* BEHAVIORAL CIRCUIT MODEL FOR SIGRITY's SystemSI - Serial Link Analysis * * MODEL PARAMETERS * * MoDEL PARAMETERS * * * * * * * * * * * * * * * * * * *
				Launch MCP Header Editor

The template has three transmitter blocks and each block has four tabs.

Connection Stimulus Jitter & Noise Power Supply

### 2.3.1.1 Connection Tab

This tab shows information about the connections between the transmitter's block and other blocks.

# 2.3.1.1.1 TX\_PRIMARY

Connection	Connect To	Block	Block Connection
tx_pkg	••	Pkg1	tx_primary

TX\_UPPER

### 2.3.1.1.2

Connection	Connect To Block	Block Connection
tx_pkg	Pkg1	tx_xtalk1

#### 2.3.1.1.3

### TX\_LOWER

Connection	Connect To Bloo	k Block Connection
tx_pkg	Pkg	1 tx_xtalk2

The primary difference between the three connections is in the block connection name. The names are for the connections at **Pkg1**.

### 2.3.1.2 Stimulus Tab

Data Rate:	5	Gbps
Data Pattern:	PRBS	Poly: 7
Leading Bits:		
Delay:	0	ns
Data Coding:	8b10b 🔻	
Rise/Fall Time		
Rise Time:	20	ps
Fall Time:	20	ps

Please refer to Section 1.3.1.2 Stimulus Tab for details.

### 2.3.1.3 Jitter & Noise Tab

Jitter			1	Noise		
Periodic				Periodic		
Frequency:	2e+008	Hz		Frequency:	2e+008	Hz
Amplitude:	0.1	]UI		Amplitude:	10	mV peak
Frequency Offset:	100	ppm		Transition:	10	mV RMS
Transition Rj:	1	%UI RMS				
Transition Dj:	1	%UI peak				
DCD:	1	%UI				

Please refer to Section 1.3.1.3 Jitter & Noise Tab for details.

### 2.3.1.4 Power Supply Tab



• **Power Supply** – Ideal voltage source that is used at the transmitter. Default value is 1 V. Please refer to *Section 1.3.1.4 Power Supply Tab* for details.

### 2.3.2 Receivers

Double-click on a receiver block to view its properties.

Property				_ C ×
Block Name: RX_U	JPPER			File Name:       D:\ASI166\1664\\$LA\xtlkex1\rx_bhvr.sp        Sub-circuit Name:       nmos_diff_rx
Conn. Port	Connect To	Block Name	Conn. Port	.subckt nmos_diff_rx pos neg pwr ngnd rxnode +rx rt=50
rx_pkg		Pkg2	rx_xtalk1	<pre>TX_1(=30 +TX_c_comp=1p +T</pre>
Connection Jitter	& Noise   Power	Supply		
				OK Cancel Apply

The property pane contains three tabs.

Connection Jitter & Noise Power Supply

### 2.3.2.1 Connection Tab

This tab shows the information about the connections between the receiver blocks and other blocks.

### 2.3.2.1.1 RX\_PRIMARY

Connection	Connect To	Block	Block Connection
rx_pkg		Pkg2	rx_primary

### 2.3.2.1.2

RX\_UPPER

Connection	Connect To	Block	Block Connection
rx_pkg		Pkg2	rx_xtalk1

### 2.3.2.1.3 RX\_LOWER

Connection	Connect To	Block	Block Connection
rx_pkg		Pkg2	rx_xtalk2

The main difference between the three connections is in the block connections name. The names are for the connections at **Pkg2**.

#### 2.3.2.2 Jitter & Noise Tab

Jitter	Noise
▼ Random (Rj): 1 %	Random (Rn): 1 mV
Deterministic (Dj): 1 %	Deterministic (Dn): 1 mV
Note: These parameters are post-processed into t	the eye distribution associated with the Rx.

Please refer to Section 1.3.2.2 Jitter & Noise Tab for details.

### 2.3.2.3 Power Supply Tab

✓ Ideal Power Supply: 1

• **Power Supply** – Ideal voltage source that is used at the receiver. Default value is 1 V. Please refer to *Section 1.3.1.4 Power Supply Tab* for details.

### 2.3.3 Pkg1

Double-click on the Pkg1 block to view its properties.

Conn. Port	Connect To	Block Name	Conn. Port	.subckt txpkg
x_primary		TX_PRIMARY	tx_pkg	+ xtalk1_pos xtalk1_neg xtalk1_posout xtalk1_negout + prim_pos prim_neg prim_posout prim_negout
x_xtalk1		TX_UPPER	tx_pkg	+ xtalk2_pos xtalk2_neg xtalk2_posout xtalk2_negout
x_xtalk2		TX_LOWER	tx_pkg	+ ngnd
ocb	••••	PCB	tx_pkg	* [MCP Begin] * [Connection] tx_xtalk1 * [Connection Type]
				* [Power Nets] * [Ground Nets]
				* 3 ngnd gnd
				* [Signal Nets] * 1 xtalk1_pos xtk1pos
				* 2 xtalk1_neg xtk1neg
				* [Connection] tx_primary * [Connection Type]
				* [Power Nets]
				* [Ground Nets] * 3 ngnd gnd
	111		•	Launch MCP Header Editor Edit Sub-circuit Definitio

The **Property** pane has two main sections:

- Connection
- File and Content

### 2.3.3.1 Connection

Connection	Connect To	Block	Block Connection	
:x_primary	••	TX_PRIMARY	tx_pkg	[Edit Layout Linkage]
:x_xtalk1		TX_UPPER	tx_pkg	[Edit Layout Linkage]
tx_xtalk2		TX_LOWER	tx_pkg	[Edit Layout Linkage ]
pcb		PCB	tx pkg	[Edit Layout Linkage]

The Connection window shows the connections between the Pkg1 block and the other blocks.

### 2.3.3.2 File and content

File Name: \Examples\xtalk_example1\txpkg_12port.sp Sub-circuit Name: txp	kg 🔽
* [Ground Nets]	^
* 7 ngnd gnd * [Signal Nets]	
* 1 xtalk1_posout xtk1pos	
* 2 xtalk1_negout xtk1neg	
* 3 prim_posout_chanpos	
* 4 prim_negout channeg * 5 xtalk2 posout xtk2pos	
* 6 xtalk2 negout xtk2neg	
* [MCP End]	
x + xtalk1 pos xtalk1 neg xtalk1 posout xtalk1 negout	
+ prim_pos prim_neg prim_posout prim_negout	
+ xtalk2_pos xtalk2_neg xtalk2_posout xtalk2_negout	
+ ngnd	
+ pkg_12port	=
.include "./pkg_12port.sp"	
.ends txpkg	
	<u> </u>
	Edit Sub-circuit Definition

The **Crosstalk Channel** Template already provides a SPICE Netlist file, which includes another SPICE file with a sub-circuit model of a 12 port s-parameter file. You can edit this model.

**NOTE!** Make sure to assign the nodes properly to maintain the connections with other blocks.

### 2.3.4 Pkg2

Refer to the **Pkg1** section. The difference between **Pkg1** and **Pkg2** is in the connectivity and the model name. Both use the same package model.

### 2.3.5 PCB

Double-click the **PCB** block to view its properties.

k Name: PCB				File Name: D:\ASI166\1664\\$LA\xtlkex1\chan_12port.sp Sub-circuit Name: channel
Conn. Port	Connect To	Block Name	Conn. Port	S Parameter File: D:\ASI166\1664\SLA\xtlkex1\xaui_ref.s4p View S Parameter File:
tx_pkg		Pkg1	pcb	( subckt channel
rx_pkg		Pkg2	PC8	[] + xtalk1_posout xtalk1_negout xtalk1_posout2 xtalk1_negout2 + prim_posout prim_posout2 prim_peoput2 + xtalk2_posout xtalk2_negout xtalk2_negout2 + ngnd * [McP Begin] * [Connection] tx_pkg * [Connection] Type] * [Connection] Type] * [Forund Ntets] * 7ngnd gnd * [Signal Ntets] * 1 xtalk1_posout xtk1pos * 2 xtalk1_posout xtk1pos * 2 xtalk1_posout xtk1pos * 2 xtalk1_posout xtk1pos * 4 prim_peoput txk1pos * 5 xtalk2_posout xtk2pos * 6 xtalk2_negout xtk2pos * 6 xtalk2_negout xtk2pos * 6 xtalk2_negout xtk2pos
				* [Connection Type] * [Power Nets] * [Ground Nets] * 7 ngnd gnd
	111			Extract BBS Model Load BBS Model Load BBS Model

The **Property** pane has two main sections:

- Connection
- File and Content

### 2.3.5.1 Connection

Connection	Connect To	Block	Block Connection
tx_pkg		Pkg1	pcb
rx_pkg		Pkg2	PCB

The connection window shows the connections between the blocks.

### 2.3.5.2 File and content

File Name: D:\ASI166\1664\SLA\xtlk	exitchan_12port.sp	Sub-circuit Name:	channel	
S Parameter File: D:\ASI 166\1664\SL	.A\xtlkex1\xaui_ref.s4p			View S Parameter
.subckt channel + xtalk1_posout xtalk1_negout xtalk + prim_posout prim_negout prim_pos + xtalk2_posout xtalk2_negout xtalk + ngnd	sout2 prim_negout2			<b>^</b>
* [MCP Begin] * [Connection] tx_pkg * [Connection Type] * [Power Nets] * [Ground Nets] * 7 ngnd gnd * [Signal Nets]				≡
* 1 xtalk1_posout xtk1pos * 2 xtalk1_negout xtk1neg * 3 prim_posout chanpos * 4 prim_negout channeg * 5 xtalk2_posout xtk2pos * 6 xtalk2_posout xtk2pos				
* 6 xtalk2_negout xtk2neg * [Connection] rx_pkg * [Connection Type] * [Power Nets] * [Ground Nets] * 7 nand and				
7 right grid				-

### 2.3.6 AMI

The AMI model (@ RX\_PRIMARY) in gray are disabled by default.

When installing the Channel Analysis tool, a template directory is created with all the template files, as well as a new folder called **amiwin**. The folder contains all the **AMI** models.

# 2.4 Analysis Options

This section covers the Analysis Options in the Setup menu.

On the Setup menu, click Analysis Options....

Set	цр	Tools	Compliance	Window
	Ar	nalysis O	ptions	
	Te	rminate	Unconnected N	Vodes
	Pa	use befo	re Simulation	
	Sw	veep Mo	de	
	Pr	obe Poin	its	
	A	MI Optio	ns	•

The Analysis Options interface opens.

Analysis Options					
Simulator	Analysis Setup Ignore Time: 5000 ns	# of Bits: 200000 Bit Samplir	ng Rate: 32	BER Floor: 1e-	16
	Simulation Configuration	Eye Distribution	IBIS Model Selectio		
SPDSIM	✓ Ideal Power	Method	No IBIS Tx/Rx		
Characterization	Corner	Time Domain Waveform	Signal Name	IBIS File	Component
	Fast V Typ	O Statistical			
Circuit Simulator Options Channel Simulator Contr	Slow	BER_Eyes			
* Add global .option and .include commands here. * Thev'll be used for time domain characterization.	Fast/Slow	BER_Eye generation			
*.option delmax=1p		Criteria Time scale (eye width)			
	- ✓ XTalk	Voltage scale (eye height)			
Channel Simulator OS: Windows 64 Bit	Odd OEven	Both time and voltage			
Simulation Name	ORandom OStatistical				
Automatic O Custom		LBERs: -12			►
•	11				
				OK Can	Apply

# 2.4.1 Circuit Simulator Options

Add the **global .option** and **.include commands**. These can be used in the Time Domain characterization.

Circuit Simulator Options Channel Simulator Controls	;
* Add global .option and .include commands here. * They'll be used for time domain characterization. *.option delmax=1p	

HSPICE simulations usually require one of these options for accurate characterization.

The Delmax option sets the maximum allowable step size of the time steps taken during transient analysis in HSPICE.

NOTE!	This option increases the simulation time but provides more accurate result, especially when using HSPICE. The option in this exercise is commented out. SPDSIM is used as the circuit simulator since it does not require 1 ps time step. It is faster than HSPICE without sacrificing accuracy.
-------	---

### 2.4.2 Xtalk

Several types of Crosstalk can be selected:

- Odd Crosstalk Channels switch opposite to the primary channel as 180° out-of-phase. The Odd Crosstalk mode is selected by default.
- **Even** Crosstalk Channels switch in-phase with the primary channel.

Random - Crosstalk Channels switch randomly with respect to the primary channel.

**Statistical** – Primary channel is simulated. The eye density is obtained. Based on the pulse response of the Crosstalk Channels, all of the Interferences of the Crosstalk Channels are statistically added. This should exhaustively add all possible interferences.

### 2.4.3 Terminate Unconnected Nodes

If there are some unused nodes in the block, SystemSI will automatically terminate the unused nodes.

By default, the termination value for Signal node is 500hm, for Power node is 1e+0080hm and for Ground node is 00hm.

To change the termination value, follow these steps:

1. Select

Setup > Terminate Unconnected Nodes....

Setup	Tools	Compliance	Window	
A	nalysis Op	otions		
Te	erminate	Unconnected I	Nodes	
Pa	ause befo	re Simulation		
Sv	Sweep Mode			
Probe Points				
A	MI Option	ns	•	

2. The Termination Impedance Definition window pops up.

Termination Impedance De	finition	×
Unconnected Signal Node:	50	Ohm
Unconnected Power Node:	1e+008	Ohm
Unconnected Ground Node:	0	Ohm
Defa	ult OK	Cancel

**NOTE!** SystemSI will add termination for unused nodes in the blocks except Transmitters and Receivers blocks.

# 2.5 Running Simulation

This section describes how to run several simulations:

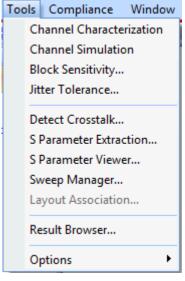
- AC Sweep
- First Simulation (Default settings)
- Enabling AMI DFE @ RX PRIMARY

• Turning on Statistical crosstalk

### 2.5.1 AC Sweep

Before running any transient or channel simulation, run **AC sweep**. This detects the coupling percentage between the primary channel and all other channels.

1. On the Tools menu, click Detect Crosstalk....



- 2. Select all transmitters.
- 3. Click Calculate.

)etect Crosstalk			_ 0
Victim Rx: RX_PRIMARY	Coupling Threshold:	% Calculate	
Include in Channel Sim	Transmitter	Coupling Factor (%)	Frequency Response
	TX_PRIMARY		·····
	TX_UPPER		····
	TX_LOWER		····
Maximum Frequency: 1	GHz # of Frequency Poin	ts: 128 OK	Cancel Apply

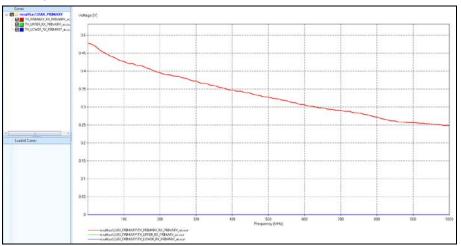
All coupling is calculated for all channels with respect to the Victim Rx.

The **Primary** channel is identified as the one with the most power transferred from **TX** to **RX**.

tim Rx: RX_PRIMARY	Coupling Threshold	d: 0 % Calculati	e
nclude in Channel Sim	Transmitter	Coupling Factor (%)	Frequency Response
<b>V</b>	TX_PRIMARY	Primary	····
	TX_UPPER	0.013	····
<b>V</b>	TX_LOWER	0.0086	···· ···

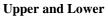
The default maximum frequency is 1 GHz. You can change the value to a higher frequency. Notice the impact on the coupling factor.

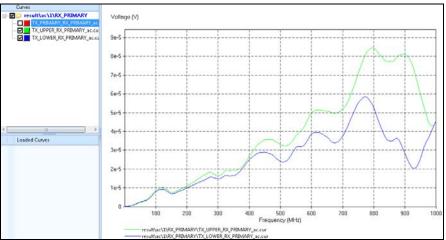
The next two figures represent the **AC sweep** results. The **AC sweep** results show the voltage at receivers.





Primary





# 2.5.2 First Simulation (Default Settings)

Click the **Start Simulation** button  $\triangleright$  to start a simulation. The purpose of the **Start Simulation** button is to characterize the channel. If the channel has not been characterized before, the button invokes the channel engine.

Any new channel needs to be characterized first before running channel simulation. Characterization of the channel means finding the step or impulse response. Run either an HSPICE or SPDSIM simulation to characterize the channel.

The **standard\_step.sp** is installed in the main programs directory. This file is a standard stimulus sub-circuit used by the tool to generate the step response.

<INSTALL\_DIR>\SpeedXP\Library\template\SystemSI\Serial Link Analysis

 IMPORTANT!
 Do not edit the standard\_step.sp file unless you are an advanced user and want to set up a different stimulus.

All the characterization information for the channel is stored in the Workspace directory.

### 2.5.2.1 Change Parameters

If you click  $\triangleright$  to run the simulation again without making any changes to the channel components, the characterization of the channel is skipped. Only the channel simulation is run. Changes to the channel include:

- Changing the parameters for **Pkg1**, **PCB** or **Pkg2** blocks, by modifying the connections, or using a different sub-circuit model.
- Changing **Tx** or **Rx** parameters, such as drive level or **c\_comp** (for **Tx**).
- Adding additional components to the channel.
- Deleting any component from the channel.
- 1. Select the **Tools** menu.
- 2. To characterize the channel only without running the channel simulation, click

### Channel Characterization.

Tools	Compliance	Window		
C	hannel Characte	rization		
C	hannel Simulatio	on		
BI	ock Sensitivity	·		
Jit	tter Tolerance			
D	etect Crosstalk			
S	S Parameter Extraction			
S	Parameter Viewe	er		
Sv	weep Manager			
La	ayout Associatio	n		
Re	esult Browser			
0	ptions	•		

### 2.5.2.2 Pause before Simulation

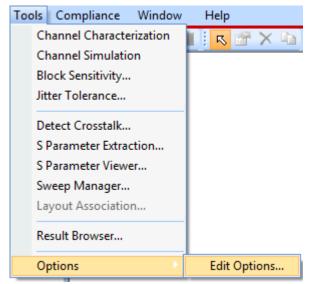
The **Pause before Simulation** option prompts you with a message whenever a new simulation is about to be run.

- 1. Select the **Setup** menu.
- 2. Click Pause before Simulation.

Setup Tools		Tools	Compliance	Window			
	Analysis Options						
	Te	rminate	Unconnected I	Nodes			
	Pause before Simulation						
	Hide Channel Simulator Messages						
	Sweep Mode						
	Probe Point						
	٨N	AMI Options					

### 2.5.2.3 General Options

Select Tools > Options > Edit options....



The **Options** window opens.

Options		×
Simulation (Seneral Result	Change the 'General' options in Serial Link	
Advanced AMI Options           AMI Builder <td< th=""><th>Automation</th><th></th></td<>	Automation	
	Address of notification mail sent when simulation is complete:	
	Multiple CPU usage	
	Maximum number of CPU to use in the simulation:	
	Messages and Windows	-
	<ul> <li>✓ Show Warning Messages</li> <li>✓ Show Channel Simulation Messages</li> </ul>	
	Default Apply OK Can	cel

The General sheet contains three sections:

- Automation you can put email address into the blank dialog box, and then you can get notification email when simulation is complete
- Multiple CPU usage you can set the maximum number of CPU to use in the simulation
- Messages and Windows -- Select the options to display or hide the simulation messages .

### 2.5.3 Simulation Results

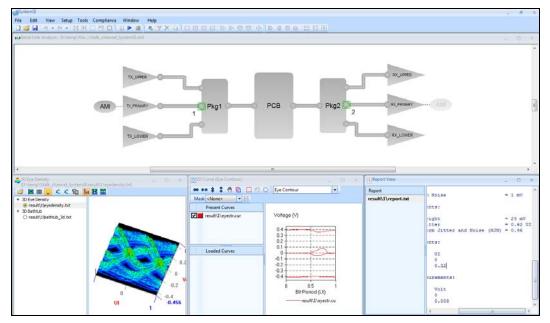
The simulation results consist of the following waveforms:

- 3D
  - Eye Density (Default)
  - Bathtub
- 2D
  - Eye Contour (Default)
  - Bathtub
  - Noise Bathtub
  - Ramp Response
  - Impulse Response
  - Modified Impulse Response
  - Rx Waveform
- Channel report (Default)

Each simulation creates a unique directory (1, 2, 3 ...etc.). All the waveform files under \**xtalk\_example1**\**result**.

These directories are moved to the history folder when SystemSI is closed.

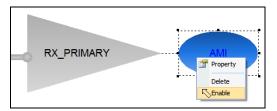
The following figure shows the Channel simulation results for Data rate = 5 Gbps. Running the channel simulation at a Data rate of 5 Gbps produces an eye contour that is not open very well.



# 2.5.4 Enable AMI DFE

Enable the **AMI** model at the receiver. That **AMI** model is a standard **DFE** or **Decision Feedback Equalizer** model.

- 1. Run the simulation again.
- 2. Right-click on the **AMI** block.
- 3. Select **Enable** to enable the **AMI** block at the primary receiver.



The **Property** pane for the **AMI** block is already loaded with an **AMIDFE** model:

Carable	
AMI Parameter File:	
$\label{eq:cadence} C:\Cadence\SPB\_16.6\ASI\Base\Speed\XP\library\template\SystemSI\Serial\Link\amiwin\amidfe\ami = 100000000000000000000000000000000000$	Reset
AMI dll File:	
C:\Cadence\SPB_16.6\ASI\Base\SpeedXP\library\template\SystemSI\SerialLink\amiwin\amidfe2.dll	Reset

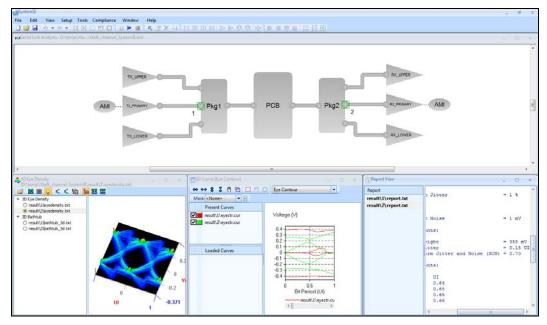
4. Use the default parameters shown in the example below:

AMI Parameter:	Reload AMI File
(amidfe (bwd <u>5</u> ) (res <u>64</u> ) (foffset <u>0</u> ) (meas_delay <u>50e-09</u> ) (coeffout <u>dfe.txt</u> )	
Reserved Parameters Model Specific	

- 5. Rerun the simulation.
- 6. Observe the results.

The following illustration shows the Channel simulation results for a Data rate = 5 Gbps + RX DFE.

7. Notice the eye contour opens up nicely.



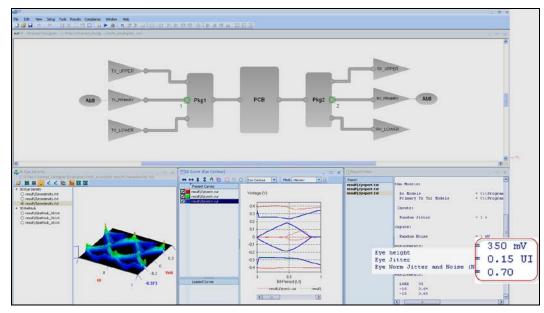
### 2.5.5 Turn on Statistical Crosstalk

The section describes how to turn on **Statistical** crosstalk.

- 1. Choose Setup > Analysis Options....
- 2. Select Statistical.

– 🗹 XTalk	
Odd	OEven
Random	<ul> <li>Statistical</li> </ul>



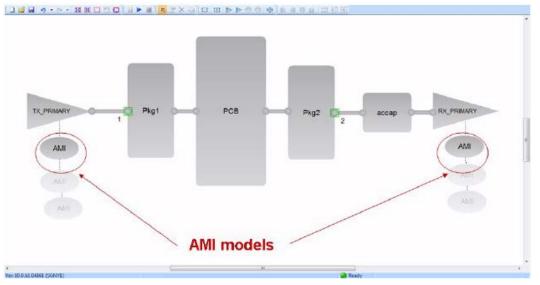


# 3 Sigrity AMI Models

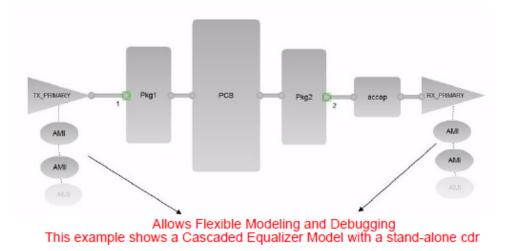
The Sigrity SystemSI – Serial Link Analysis includes several SERDES equalizer behavior models. All of these models use the Algorithmic Model Interface (AMI). This chapter describes using the Sigrity AMI models.

# 3.1 Model Design

AMI Models can be added to Tx or Rx.



AMI Models can also be cascaded. This allows flexible modeling and debugging. The example shows a Cascaded Equalizer Model with a stand-along **CDR** (Clock & Data Recovery).



# 3.2 Model Configuration

- **AMICDR** Recovers the Reference Clock signal and aligns it to the middle of the eye.
- Typically at Rx.
- **AMICTWF** Used for Continuous Time Linear Equalization.
- **AMICTWFADAPT** Adaptive version of AMICTWF.
- **AMIFFE** Feed Forward Equalization. Pre-de-emphasis; typically at Tx.
- **AMIDFE2** Decision Feedback Equalization.
- **AMIDFENL** Advanced. Non-linear DFE.

# 3.2.1 AMICDR Configuration

The algorithmic model is **amicdr2.dll**.

### 3.2.1.1 AMICDR Description

AMICDR is generally applied at the Rx side. CDR:

- 1. Takes in raw data waveforms.
- 2. Recovers the reference clock signal.
- 3. Aligns it to the middle of the eye.

### 3.2.1.2 AMICDR User Parameters

Parameter	Туре	Values	Comment
res	integer	64 32 128	Resolution for recovery.
cdr_off	integer	001	Turn CDR functionality ON or OFF.
foffset	integer	0 -7000 7000	Frequency offsetin PPM.

# 3.2.2 AMICTWF Configuration

The algorithmic model is **amictwf.dll**.

### 3.2.2.1 AMICTWF Description

CTWF stands for Continuous Time Waveform.

It is used for Continuous Time Linear Equalization.

It typically applies high-pass frequency filtering to the incoming signal at the Rx side.

### 3.2.2.2 AMICTWF User Parameters

Parameter	Туре	Values	Comment
profile	string	profile.txt	Transfer function table in either fre- quency or time domain format.
gain	float	1 0.5 2	Manual gain adjustment.

# 3.2.3 AMICTWFADAPT Configuration

The algorithmic model is **amictwfadapt.dll**.

### 3.2.3.1 AMICTWFADAPT Description

AMICTWFADAPT is the adaptive version of AMICTWF. It includes an adaptive high-pass filtering algorithm. It has optional integrated CDR and DFE functionality.

### 3.2.3.2 AMICTWFADAPT User Parameters

Parameter	Туре	Values	Comment
dbl	float	6 6 15	Initial dB loss of filter.
fO	range	1e9 0.5e9 3e9	Cutoff frequency.
adapt_on	integer	101	Adaptation ON/OFF switch.
adapt_cyc_latency_transitions	integer	256	Adaptation cycle in UI.
adapt_freq_factor	integer	128	The quantized tap coefficients are out- put to a text file qffecoeff.txt.
cdr	string	cdr dfe off	Integrated CDR, CDR together with DFE or neither one enabled.
cdr_off	integer	001	Turns off CDR correction for cdr and dfe settings in previous parameter.
magphout	string	mph.txt	Magnitude / phase plot of CRLE filter.
rel_err	float	1e-4	Relative error to limit filter size.

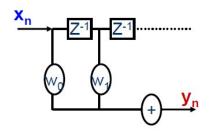
# 3.2.4 AMIFFE Configuration

Feed Forward Equalization is represented mathematically as:

yn = Σ wi\*xi

xn - input

Yn - output

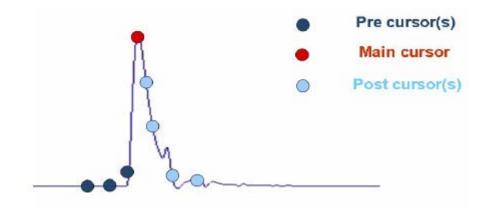


### 3.2.4.1 AMIFFE Description

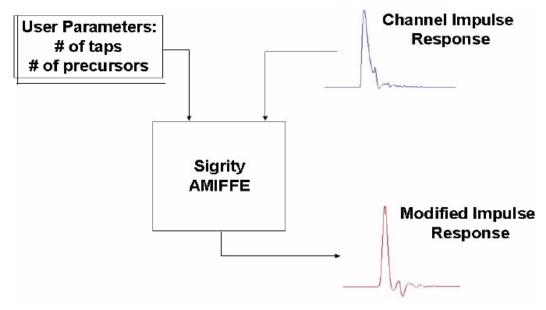
AMIFFE is generally applied at the Tx side. It can be applied to the Rx. They are mathematically equivalent. The filter is specified by:

- Number of Taps
- **Tap Coefficients** Set of weighing factors (wi).
- **Tap Spacing** Delay between taps. AMIFFE assumes Tap Spacing is one bit at a time.

3.2.4.2 AMIFFE Tap Terminology



3.2.4.3 AMIFFE Automatically Optimizes the Tap Coefficients



### 3.2.4.4 AMIFFE.AMI User Parameters

Parameter	Туре	Values	Comment
fwd	Integer	2 1 128	Forward Taps. Includes main cursor.
pre	Integer	105	Precursor taps.
coeffout	Integer	ffcoeff.txt	The tap coefficients are output to the text file ffco- eff.txt.
UserTapsFile	String	nil	User-supplied Tap Coefficient from a file. By default it is set to <b>nil</b> meaning no user-supplied Tap Coeffi- cients. If the user supplies the coefficients no Opti- mization is performed.
lffe	String	0.5 0.1 1.0	Normalized Tap Limits. Main driver typically = 1.0.
qffe	Float	6 1 10	Tap Resolution to number of decimal places.
csum	Integer	001	Setting = 1 forces sum of normalized Tap Coeffi- cients equal to 1.
qcoeffout	String	qffcoeff.txt	The quantized Tap Coefficients are output to the text file <b>qffcoeff.txt</b> .
offset	Float	0 -0.5 1.5	Tweaks the automatic error minimization algorithm for Tap Coefficient synthesis.
OptimizerPulse	Integer	001	Tweaks the FFE coefficient generation algorithm. Setting this parameter may improve the coefficients.
refine_coeff	Integer	001	Setting = 1 gives an additional stage of non-linear Tap Optimization. Can improve over standar MMSE algorithm. Can use user-supplied starting coeffi- cients.
lffe_abs	Integer	1	Normalized Tap absolute values in milli-amps.
qffe_dec	Integer	1	Tap Resolution in decimal units.

The AMI parameter "lffe" is an important parameter for the "amiffe.dll" algorithmic model. This parameter sets both the number of taps (it will override the "fwd" parameter if it exists), and also the limits on those taps.

The AMI parameter "csum" can be set to make the sum of all taps=1. Otherwise the main tap will be "1" and the other taps will take values no greater than their associated 'lffe' limit.

If you do not want a precursor tap, set the "lffe" parameter in the following manner:

( <val\_less\_than\_1>,,,,,,)

### 3.2.5 AMIDFE2 Configuration

DFE stands for Decision Feedback Equalizer. It removes the Inter-symbol Interface (ISI) by adding corrections to the input based on previous decisions. Decision Feedback Equalization is represented mathematically as:

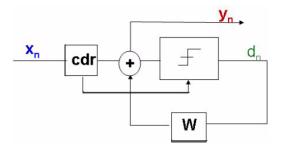
 $y_n = X_n + \Sigma w_i * d_i$ 

yn - output

xn - input

di - previous ith decision

wi - ith tap weight



#### 3.2.5.1 AMIDFE2 Description

The **Tap weights** are determined adaptively by the equalizer and cannot be set by the user. This is also known as **blind equalization**. During the simulation an **ignore\_time** setting of 5000ns is recommend-ed. This ensures that the adaption algorithm has enough time to settle.

DFE has an integrated Clock and Data Recovery module (**CDR**). This DFE does not correct precursor ISI. The best results are obtained when an **FFE** with precursor correction is applied at the **Tx** side. DFE is applied to the **Rx** device.

### 3.2.5.2 AMIDFE2 LMS Algorithm

The Least Mean Square (LMS) adaptive algorithm looks like this:

Wi(n + 1) = Wi(n) + U \* E \* dm(n)

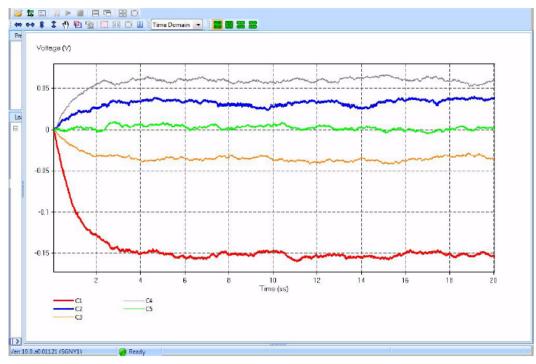
Wi - ith coefficient

di – ith symbol

E - error

u - proportionality factor



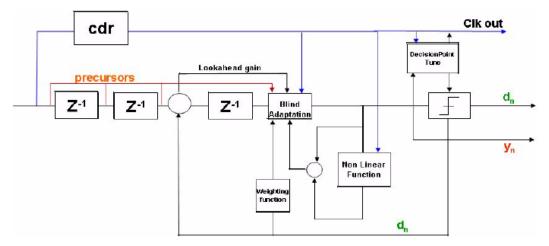


### 3.2.5.4 AMIDFE2 User Parameters

Parameter	Туре	Values <sup>1</sup>	Comment
bwd	Integer	5 1 64	Number of backward Taps. Default is 5.
res	Integer	64 32 128	Bit Resolution for the integrated CDR. Default is 64, meaning UI/64.

1- If values are numbers then the format is typical min max.

# 3.2.6 AMIDFENL Configuration



### 3.2.6.1 AMIDFENL Description

The DFE coefficients by blind adaptation are:

- Standard linear feedback error minimization algorithm.
- Optional non-linear erro minimization algorithm.
- Optional tuning of decision point.

Optionally include precursor.Optional adaptive gain amplifier means the algorithm decides on amplification based upon the corrected waveform.

AMIDFENL includes:

- Optional tuning of decision point.
- Optionally include precursor.

### 3.2.6.2 AMIDFENL User Parameters

Parameter	Туре	Values <sup>1</sup>	Comment	
bwd	Integer	5 1 64	Number of backward Taps. Default is 5.	
nbwd	Integer	1 0 64	Number of special non-linear backward Taps. Default is 1.	
pre	Integer	0010	Number of precursor Taps. Default = 0. Not as effective as dedicated pre-emphasis in transmitter.	
lookahead	Integer	001	Turn on adaptive gain amplifier. Default is 0.	
Vc_min	Float	0.1 0.1 0.6	Normalized gain for adaptive gain amplifier. Valid only if <b>lookahead</b> is set to 1. Default value is 0.4. Higher values add significant power consumption. Try to use as low a value as possible.	
Tc_adjust	Integer	0 - 1 1	Tune the decision point. 0 is inactive1/+1 changes the direction of tuning. Some coding patterns may require -1.	
res	Integer	64 32 128	Bit Resolution for the integrated CDR. Default is 64 meaning UI/64.	

1- If values are numbers then the format is typical min max.

# 4 Advanced Capabilities

This chapter covers following advanced capabilities in SystemSI – Serial Link Analysis (SSI–SLA).

- IBIS Transmitter and Receiver
- Sweep Manager
- S-Parameter Extraction
- Jitter Tolerance
- Block Sensitivity
- Result Browser and 2D Curve Presentation
- Auto Archive SSI Project
- Integration with Allegro Signal Explorer

Use these capabilities to further investigate channel performance and optimize the design.

# 4.1 IBIS Transmitter and Receiver

### 4.1.1 Add IBIS Transmitter and Receiver

- 1. Click the Add IBIS Transmitter icon is or Add IBIS Receiver icon is.
- 2. Click in the layout view window.

The **Tx1** or **Rx1** block is added.

SystemSI - [Serial Link Ar	alysis : D:\12.1\simp	ple_channel\simple_ch	annel.ssix]	
*√¢ File Edit View	Setup Tools Co	ompliance Window	Help	
🗋 💕 🛃 🤌 🔻 (° -	- 🐹 🖾 🛄 🗹		rs 🚰 🗙 🐚	日12112  >  >
Workflow: SystemSI	×			Add IBIS Transmitter
Serial Link Analysis	≽			
SFP+ Compliance	×			
HDMI Compliance	*			
PCIe 3 Compliance	*			Tx1
10GBASE-KR Compliance	e 😞			······
Channel Setun				
		-		
SystemSI - [Serial Link A	nalysis : D:\12.1\sim	nple_channel\simple_c	hannel.ssix]	
₩ File Edit View	Setup Tools C	Compliance Window	v Help	
🗋 💕 🛃 🧳 🔻 (*	- 🐹 🖸 🛄	2 🖸 11 🕨 🔳	r 🖉 🗙 🕻	1 日日日日   ● ● ● ● ● ● ● ● = = = = = = = = = = =
Workflow: SystemSI	×			Add IBIS Receiver
Serial Link Analysis	×			
SFP+ Compliance	×			BB.

### 4.1.2 Load an IBIS File

PCIe 3 Compliance 10GBASE-KR Compliance

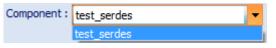
1. Double-click the IBIS Tx or IBIS Rx block. The **Property** pane opens.

					OnDie Parasitics		r
Connection	Connect To	Block	Block Connection	File Name:		Sub-circuit Name:	
		111					Load IBIS

2. Click the Load IBIS... button. The Load IBIS window opens.

					Component :		
Pin Mapping [	Diff Pin						
Pin	Pulldown	Pullup	GND Clamp	Power Clamp	Signal Name	Model Name	
Filter						ОК	Cancel

- 3. Click the button to load an IBIS file.
- 4. Choose from the **Component** drop-down list to select a component for the IBIS Tx block or IBIS Rx block.

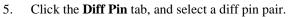


The **Pin Mapping** tab is shown.

The references, including Pullup, Pulldown and Gnd/Pwr rails for the Bus Signals, are identified in the Pin Mapping.

In the example below, Serial Link Analysis will assume Ideal Power and Ground for all simulations.

in Mapping	Diff Pin						
in	Pulldown	Pullup	GND Clamp	Power Clamp	Signal Name	Model Name	
					ami_tx_p	test_ami_tx1	
					ami_tx_n	test_ami_tx1	
					ami_rx_p	test_ami_rx1	
					ami_rx_n	test_ami_rx1	
					vdd	POWER	
i					VSS	GND	



Pin         Inverting Pin         vdiff         tdelay_typ         tdelay_min         tdelay_max           I         2         0.01Y         NA         NA         NA         NA           I         2         0.01Y         NA         NA         NA         NA           Agorithmic Model:         Image: State S	es	_serdes	les	-		>
Algorithmic Model:     OS     AMI Model       Model Name     OS     AMI Model       Model Name     OS     AMI Model       Itest_ami_tx1     Windows_VS_32     v ami_ffe_ibis_ami       test_ami_tx2     Windows_VS_32     ami_ffe_ibis_wint.dll						
Algorithmic Model: Model Name OS AMI Model AMI File AMI File test_ami_tx1 Windows_VS_32 ▼ ami_ffe_ibis_wint.dll ami_ffe_ibis.ami test_ami_tx2 Windows_VS_32 ami_ffe_ibis_wint.dll ami_ffe_ibis.ami						
Model Name     OS     AMI Model     AMI File       test_ami_tx1     Windows_VS_32 <ul> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_ami</li> <li>ami_ff</li></ul>						
Model Name     OS     AMI Model     AMI File       test_ami_bx1     Windows_VS_32 <ul> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_ami</li> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_ami</li> <li>a</li></ul>						
Model Name     OS     AMI Model     AMI File       test_ami_bx1     Windows_VS_32 <ul> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_ami</li> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_ami</li> <li>a</li></ul>						
Model Name     OS     AMI Model     AMI File       test_ami_bx1     Windows_VS_32 <ul> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_ami</li> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_ami</li> <li>a</li></ul>						
Model Name     OS     AMI Model     AMI File       test_ami_bx1     Windows_VS_32 <ul> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_ami</li> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_ami</li> <li>a</li></ul>						
test_ami_tx1     Windows_VS_32 <ul> <li>ami_ffe_ibis_wint.dll</li> <li>ami_ffe_ibis_ami</li> </ul> ami_ffe_ibis_wint.dll     ami_ffe_ibis.ami           Linux_gcc4.1.2_32 Solaris_gcc3.3_32         ami_ffe_ibis_wint.dll         ami_ffe_ibis.ami						
test_ami_tx2 Windows_V5_32 ami_ffe_jbis_wint.dl ami_ffe_jbis.ami Linux_gcc4.1.2_32 Solaris_gcc3.3_32						
Linux_gcc4.1.2_32 Solaris_gcc3.3_32						
Solaris_gcc3.3_32						
				_	_	_

If the IO models have any AMI models, please select an OS that matches the OS of the Channel Simulator.

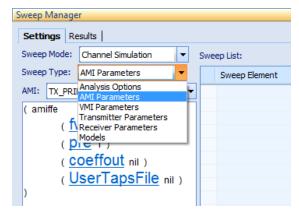
If the IO model is selected for the simulation, the selected AMI model will be used. The .sp file for the selected IBIS component will be automatically created and assigned to the IBIS block.

Rx OnDie Parasitics Package Parasitics	
File Name: D:\12.1\simple_channel_complex\result\test_ibis_Rx2.:	-
.subckt test_serdes_Rx2 3 4 5 6 rxnode	
* [MCP Begin] * [Connection] Rx_Out * [Connection Type] * [Power Nets] * 5 5 vdd * [Ground Nets] * 6 6 vss * [Signal Nets] * 3 3 ami_rx_p * 4 4 ami_rx_n * 4 4 ami_rx_n * [Connection] Repeater_Connection * [Connection Type] * [Power Nets]	11
[ one real	
Load IBIS.	

# 4.2 Sweep Manager

The exercise uses one of the single channel templates. The types of sweeps are explained and in different case examples.

The Sweep Manager supports six types of sweeps.



- Analysis Options Sweeps parameters defined in Analysis Options.
- **AMI Parameters** Sweeps AMI model parameters such as the number of forward tabs and coefficients.
- VMI Parameters Vendor Model Interface. Companion capability that can be used together with standard IBIS-AMI models to enable additional automation from advanced SERDES IP suppliers.
- Transmitter Parameters Sweeps transmitter parameters like data rate and number of bits.
- **Receiver Parameters** Sweeps receiver parameters like random jitter and deterministic jitter.

If IBIS transmitter and receiver are added, the Transmit IO Model and Receive IO Model are be added to the Sweep Manager for sweeping.

Settings Re													
occording 105	suits												
Sweep Mode:	Channel Simulation	1-	3	weep List:						Total Iterations: 0		F	Run Sweeps
Sweep Type:	Transmitter Parameter			Sweep Element	Sweep Type	Min	Max	Step	Value I	Iteration			
Transmitter:	Tx2		•						11.0000100				
Property		Value											
Periodic Jitter		5	2										
	Frequency (Hz)	60	m										
	Amplitude (mV)	10 0											
Transition Jitte Transition Jitte		0											
Transition Nois		10	1										
DCD (%)	- (***)	0											
Transmit IO M	odel	test_ami_tx											
L	a parameter item to ad		•	elect some sweep items and right click on t			25		P	Click the checkbox to select			
Analysis Optio	ns Sweep Manager	Property											
weep Manager	Gassocheronausflech	Property											
weep Manager Settings Res	Gassocheronausflech	Property	5	veep List:		_			_	Total Iterations: 0			Run Sweep
Settings Res	ulta		s	veep List: Sweep Bonent	Sweep Туре	Min	Max	Step	Value 1			(	Run Sweep
Settings Res Sweep Mode: Sweep Type:	ults   Channel Smulation	•	s		Sweep Туре	Min	Max	Step	Value 1			[	Run Sweeps
Settings Res Sweep Mode: Sweep Type: Receiver: Property	ults   Channel Smulation Rx2	•	_[		Sweep Туре	Min	Max	Step	Value 1			(	Run Sweeps
Settings Res Sweep Mode: Sweep Type: Receiver: Property Random Jitter	ults   Channel Simulation Theorem For anothers Rx2 (R) (%)	Value 1	_[		Sweep Type	Me	Max	Step	Value 1			[	Run Sweeps
Settings Res Sweep Mode: Sweep Mode: Receiver: Property Random Jitter Deterministic J	ults   Channel Smulation Receiver Parameters Rx2 (R) (%) tter (D) (%)	Value 1 0	_[		Sweep Type	Mn	Max	Step	Value 1			[	Run Sweeps
Settings Res Sweep Mode: Sweep Mode: Sweep Type: Receiver: Property Randon Jitter Deterministo J Rendom Noise	ults   Channel Smulation Rx2 (R) (%) (tter (D) (%) (mr) (mV)	Value 1 0	_[		Sweep Type	Mn	Max	Step	Value 1			[	Run Sweeps
Settings Res Sweep Mode: Sweep Mode: Sweep Type: Receiver: Property Random Jitter Deterministic J Random Noise Deterministic J	ults   Channel Smulation Second Planameters Rx2 (R) (%) (R) (%) (R) (P) (R) (P)	Value · · · · · · · · · · · · · · · · · · ·	_[		Sweep Type	Min	Max	Step	Value 1			(	Run Sweeps
Settings Res Sweep Mode: Sweep Mode: Sweep Type: Receiver: Property Randon Jitter Deterministo J Rendom Noise	ults   Channel Smulation Second Planameters Rx2 (R) (%) (R) (%) (R) (P) (R) (P)	Value 1 0	_[		Sweep Type	Min	Max	Step	Value 1			[	Run Sweeps
Settings Res Sweep Mode: Sweep Mode: Sweep Type: Receiver: Property Random Jitter Deterministic J Random Noise Deterministic J	ults   Channel Smulation Second Planameters Rx2 (R) (%) (R) (%) (R) (P) (R) (P)	Value · · · · · · · · · · · · · · · · · · ·	_[		Бикер Туре	Min	Max	Step	Value 1			[	Run Sweeps
Settings Res Sweep Mode: Sweep Mode: Sweep Type: Receiver: Property Random Jitter Deterministic J Random Noise Deterministic J	ults   Channel Smulation Second Planameters Rx2 (R) (%) (R) (%) (R) (P) (R) (P)	Value · · · · · · · · · · · · · · · · · · ·	_[		Sweep Type	Mm	Max	Step	Volue 1			[	Run Sweeps
Verty Manager Settings Res Sweep Mode: Sweep Type: Receiver: Property Random Note: Deterministic J Random Note: Deterministic J Receive 10 Mod	ults   Channel Smulation Second Planameters Rx2 (R) (%) (R) (%) (R) (P) (R) (P)	Value Value I O Lest_omi_rx			11	Min	Max	Step	Value 1		t or unselect on illeratio	( ħ	Run Sweeps
Verty Manager Settings Res Sweep Mode: Sweep Type: Receiver: Property Random Note: Deterministic J Random Note: Deterministic J Receive 10 Mod	white   Channel Smulation Scracher Planaters Ika2 (R) (%) (R) (%) (R) (%) (R) (%) (R) (m) (s) (R) (m) (s) (R) (M) (s) (R) (M) (s) (R) (M) (s) (R) (R) (R) (R) (R) (R) (R) (R) (R) (R) (R) (R) (R)	Value Value I O Lest_omi_rx		Sweep Benent	11	Min	Max	Step	Value 1	Iteration	t or unselect an iteration	( 1.	Run Sweeps
Verty Manager Settings Res Sweep Mode: Sweep Type: Receiver: Property Random Note: Deterministic J Random Note: Deterministic J Receive 10 Mod	white   Channel Smulation Scracher Planaters Ika2 (R) (%) (R) (%) (R) (%) (R) (%) (R) (m) (s) (R) (m) (s) (R) (M) (s) (R) (M) (s) (R) (M) (s) (R) (R) (R) (R) (R) (R) (R) (R) (R) (R) (R) (R) (R)	Value Value I O Lest_omi_rx		Sweep Benent	11	Min	Max	Step	Value 1	Iteration	t or unselect an iteratio	n.	Run Sweeps

• Models - Sweeps SPICE Netlist files such as .Inc models or other parameters.

## 4.2.1 Launch Single Channel Complex Template

- 1. Launch SystemSI.
- 2. In the **File** menu, click **New**;
  - or

in the Main toolbar, click the New button.

The **Select Module** dialog opens.

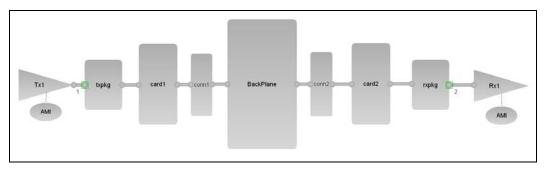
Se	elect Module	×
	Module Name	
	Parallel Bus Analysis	
	Serial Link Analysis	
		OK Cancel

- 3. Select Serial Link Analysis.
- 4. Click OK.

The New Workspace dialog (Single Channel Template) opens.

New Workspace		×
Create a blank project Create by template Tem	plate Path: C:\Cadence\SPB_16.6\ASI\Base\Spee	dXP\Library\template\}
Name	Path	Description
measurement_xtalk repeater_simple	C:\Cadence\SPB_16.6\ASI\Base\Speed C:\Cadence\SPB_16.6\ASI\Base\Speed	=
single_channel_complex single_channel_simple	C:\Cadence\SPB_16.6\ASI\Base\Speed C:\Cadence\SPB_16.6\ASI\Base\Speed	
sla_simple_em	C:\Cadence\SPB_16.6\ASI\Base\Speed	
Name: single_complex Location: D:\Working\SystemS		
		OK Cancel

- 5. Select **Create by template**.
- 6. Enter a name for the new workspace, such as **simple\_complex**.
- 7. Enter or select a location.
- 8. Click **OK.** The workplace appears.



The single channel complex template contains the following blocks:

- A primary transmitter (**Tx1**)
- A primary receiver (**Rx1**)
- Two packages (**txpkg** and **rxpkg**)
- Two add-in cards (card1 and card2)
- Two connectors (conn1 and conn2)
- A backplane (**BackPlane**)
- Two **AMI** models

	You can either double-click on a component in the canvas or select a
NOTE!	component and click the <b>Property</b> button and the <b>Select</b> toolbar to view the properties.

### 4.2.1.1 Block Models

Each block contains the following models.

Component	Circuit model
Тх	nmos output driver behavior model.
Rx	simple input behavior model.
txpkg, rxpkg	S-parameter package model (s4p).
card1, card2	Approximate 3-inch daughter card; w-element model.
conn1, conn2	VHDM distributed circuit connector model.
BackPlane	24-inch XAUI type channel (s4p).

# 4.2.2 Explore Sweep Manager

• Start in the **Tools** menu.

Тоо	ls Compliance	Window						
Frequency Response								
	S Parameter Extrac	tion						
	S Parameter Viewe	r						
	Characterize Chan	nel						
	Channel Simulatio	n						
	Block Sensitivity							
	Jitter Tolerance							
	Sweep Manager							
	Model Builder							
	Result Browser							
	Options	•						

• Click Sweep Manager....

The Sweep Manager interface opens. The Sweep Manager interface has three sections:

- Parameter
- Sweep List
- Iteration

Sweep Mai	nager									_ 🗆 ×
Setting	s Results									
Sweep	Channel Simulatior	Ŧ	Swe	ep List:				Total Iterat	ions: 0	Run Sweeps
Sweep	Analysis Options	•		Sweep Element				Iteration		
Paramete	er:									
Propert	У		-							
Number										
Bit Sam	pling Rate	_	_							
			-							
		_								
			-							
	111	Þ	•	111	)		•			
Double cl	ick on a parameter it to 'Sweep List'.	em	Sele	ct some sweep item roup or delete.	s and right cli	ick on the list to		Click the che an iteration		elect or unselect
	to phoop block		angi					an test delori		
								ок	Cancel	Apply

#### 4.2.2.1 Sweep Mode

The two options supported are:

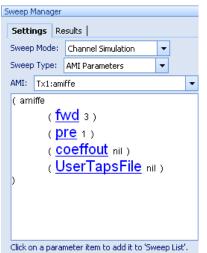
- Channel Simulation
- Frequency Response

### 4.2.2.2 Sweep Type

By default, Analysis Options is selected as the sweep type.

Based on the Sweep Type selected, different values are populated in the first list box. You can either click or double-click on a parameter to add to the **Sweep List**.

In the example shown, the sweep name appears in the Sweep List if you click on fwd.



#### 4.2.2.3 Sweep List

To choose parameter values, select the Sweep item from the Sweep List and go to the next section. The **Sweep List** pane contains these items:

- Sweep Element- Identify the Sweep Type and parameter or model selected.
- **Sweep Type** Choose from the drop-down menu.
- **Step Count** Number of different values for each parameter.
- Min, Max, Step, and Value List Related element values.

Sweep Element	Sweep Type	Min	Max	Step	Value List	Step Count
AMIParameterSweep>Tx1>amiffe>fwd	Parameter					0
ModelFileSweep>conn1	File					0
TransmitterParameterSweep>Global	Parameter					0

#### Sweep Type

The default **Sweep Type** is dependent on the parameter chosen. In this example, **Parameter** is the default.

Sweep Element	Sweep Type	Min Max	Step	Value List	Step Count
AMIParameterSweep>Tx1>amiffe>fwd	Parameter 🔻				0
ModelFileSweep>conn1	Parameter				0
TransmitterParameterSweep>Global	File				0
	Output File				

#### 4.2.2.4 Total Iterations

**Total Iterations** are the total number of simulations to be preformed. In this example, 3 Iterations will be performed.

						Total Iterati	ions: 3	
Sweep Type		<b>.</b>						
				value List			Tx1>amiffe>f	
Parameter	1	3	1		3	🗹 1	1	
						2	2	
						✓ 3	3	
111					<b>)</b>			
st to group, un	group	or delete.				Click the che	eckbox to select or	unselect an iteration.

1. Right-click the spreadsheet of **Total Iterations**.

Two options Export Settings... and Import Settings... are available in the pop-up menu list.

Iteration	Тx	1>amiffe>
✓ 1	1	
<ul> <li>✓ 2</li> <li>✓ 3</li> </ul>	23	Select All Deselect All Select All Highlighted Items Deselect All Highlighted Items Export Settings
		Import Settings
Click the che iteration.	eckb	ox to select or unselect an

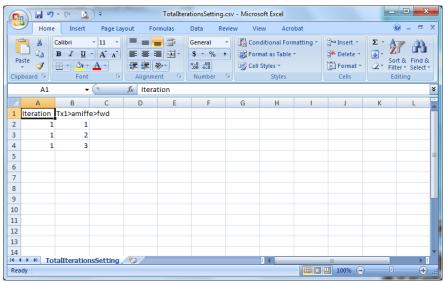
They are used to export and import the Iteration settings in the csv file.

2. Click **Export Settings...**.

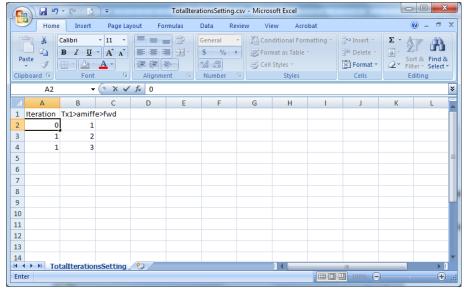
The Export settings window opens.

Export settings							×
🕞 🕤 - 🕌 «	Working 🕨 Syste	mSI ► single_comple	ex ▶ result	▼ \$	Search result		P
Organize 🔻	New folder						0
★ Favorites ■ Desktop ↓ Downloace ③ Recent Pla ⊘ My Site ○ Libraries ○ Documen	eces E	<u>.</u> *	No items m	Date modified atch your search.	Туре	2	ize
Document     Music     Pictures     Videos							
Computer	<b>▼</b>						•
File <u>n</u> ar Save as <u>t</u> y	me: TotalIterations						•
Alide Folders				E	Save	Cance	

- 3. Click **Save** to save a csv file listing all the possible iterations.
- 4. Open the saved csv file.



- The number 1 in the Iteration column indicates this iteration is enabled in the Total Iterations spreadsheet
- The number 0 in the Iteration column indicates this iteration is disabled in the Total Iterations spreadsheet
- 5. Change the **Iteration** value of the first row to **0** as follows, and save the file.



- 6.
- Click Import Settings... in the Sweep Manager window.

Total Iterat	ions:	3	Run Sweeps
Iteration	Tx1	1>amiffe>	
<b>V</b> 1	1		
✓ 2	2	Select All	
<b>V</b> 3	3		
		Deselect All	
		Select All Highlighted Items	
		Deselect All Highlighted Items	
		Export Settings	
		Import Settings	
Click the ch	eckbo	ox to select or unselect an iteration.	

The Import settings window opens.

MaImport settings			×
💮 🌍 🗸 📙 « Worl	king → SystemSI → single_complex → result	👻 🍫 Search	n result 🔎
Organize 🔻 New	folder		:= • 🔟 🔞
🔆 Favorites	A Name	Date modified Typ	oe Size
<ul> <li>Desktop</li> <li>Downloads</li> <li>Recent Places</li> <li>My Site</li> <li>Libraries</li> <li>Documents</li> <li>Music</li> <li>Pictures</li> <li>Videos</li> </ul>	I TotallterationsSetting.csv	3/25/2013 10:17 AM Mic	rosoft Office E 1 K
P Computer	• •	m CSV Files	(* cov)
	File name:	CSV Files	

- 7.
- Choose the saved csv file, and click **Open**.

The settings of iterations in the csv file are shown in the Total Iterations spreadsheet.

Total Iterat	ions: 2	Run Sweeps
Iteration	Tx1>amiffe>	
1	1	
<b>V</b> 2	2	
<b>V</b> 3	3	
Click the ch	eckbox to select or	unselect an iteration.

The first row of **Iteration** is disabled.

#### 4.2.2.5 Results

1. After all simulations are done, click the **Results** tab to display the results.

weep Manager					_ 🗆 X
Settings Resu	lts				
Current Hist	tory		Ex	port	how Result
Iteration	Folder	Eye Height (mV)	Eye Jitter (UI)	Eye NJN	
Click 'Show Resul	It' button to show the	results of the check	ed iterations, or dou	ıble dick on ar	iteration to
show the results					
			ОК	Cancel	Apply

2. To view curves such as Eye contour and Bathtub curves, highlight one or more iterations and click Show Result...

## 4.2.3 Running a Single Sweep

Set up the Simulation Sweep Options and run the sweep using the example in *Section 4.3.1 Launch Single Channel Complex Template*.

### 4.2.3.1 Analysis Options

When the Sweep Manager is open, the Sweep Mode option is automatically selected.

Seti	цρ	Tools	Compliance	Window		
	Ar	nalysis O	ptions			
	Te	rminate	Unconnected I	Nodes		
	Pa	use befo	ore Simulation			
	Hide Channel Simulator Messages					
•	Sv	veep Mo	de			
	Pr	obe Poin	it			
	A	MI Optio	ns	•		

Make sure the Sweep Manager interface is open. Click OK to start simulation.

SP	×
1	This will require 3 simulations. Continue?
	OK Cancel

#### 4.2.3.2 View the Results

Click the **Results** tab after the simulations finish. The **Results** pane displays:

• Measured Eye Height (mV)

- Eye Jitter (UI)
- File Location(Folder path)
- Eye NJN
- List of all the Iterations
- Tx1

Current H	listory			Expor	t Show Result
Iteration	Folder	Eye Height (mV)	Eye Jitter (UI)	Eye NJN	Tx1>amiffe>fwd
✓ 1	result\1	0	1	1	1
✓ 2	result\1	0	1	1	2
✓ 3	result\1	67	0.53	0.92	3
Click 'Show Re he results.	sult' button to sho	w the results of the d	hecked iterations, o	or double click	c on an iteration to show

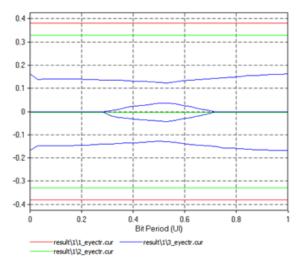
Two settings for the **fwd** parameter do not produce any eye opening.

- 1. Select one or more iterations.
- 2. Click Show Result... to view the curves. The Show Simulation Result dialog appears.

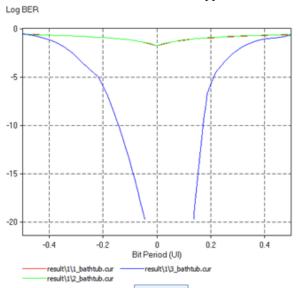
Show Simulation Result
Option
Show in the same windows
Select Result
Eye Density
Eye Contour
✓ Bathtub
☑ Noise Bathtub
Ramp Response
✓ Impulse Respons
Modified Impulse Response
Rx Waveform
Report
Select All Deselect All
OK Cancel

3. Click **OK**. The **Eye Contour** results appear.





The **Bathtub** cureves for all the iterations appear.



4. Click the **Export** button in the **Sweep Manager** interface to export the results. The **Save As** dialog opens.

Save As						? ×
Save in:	🚞 result		•	G 🦻	<del>ب</del> 🕫	
My Recent Documents	iac iemp					
Desktop						
My Documents						
My Computer						
	File name:	test_results			•	<u>S</u> ave
My Network	Save as type:	TXT Files (*.txt)			-	Cancel

# 4.2.4 Run Multiple Sweeps

This section describes how to sweep multiple parameters and across categories to show the sweep capability. The parameters and the values are to be set:

fwd of amiffe: 2, 3

Data Rate of Tx1 (Gbps): 5, 8

**DCD** of Tx1 (%): 0, 1

There are a total of 8 combinations.

weep List:							Total Iterat	ions: 8		
Sweep Element	Sweep Type	Min	Max	Step	Value List	Step Count	Iteration	Tx1>amiffe>f	Tx1>Data Ra	Tx1>DCD (%)
AMIParameterSweep>Tx1>amiffe>fwd	Parameter	2	3	1		2	✓ 1	2	5	0
TransmitterParameterSweep>Tx1>Da	Parameter				5, 8	2	2	3	5	0
TransmitterParameterSweep>Tx1>D	Parameter				0, 1	2	🖌 3	2	8	0
							✓ 4	3	8	0
							✓ 5	2	5	1
							6	3	5	1
							7	2	8	1
							8 🖌	3	8	1
Gelect some sweep items and right click on the li	st to group, un	group or	delete.				Click the ch	eckbox to select or	unselect an iterati	on.

After the simulations are run, the results appear in the **Results** pane. The results from the first sweep are now found under the **History** tab.

Iteration	Folder	Eye Height (mV)	Eye Jitter (UI)	Eye NJN	Tx1>amiffe>fwd	Tx1>Data Rate (Gbps)	Tx1>DCD (%)
✓ 1	result\1	211	0.34	0.85	2	5	0
2	result\1	275	0.31	0.78	3	5	0
✓ 3	result\1	0	1	1	2	8	0
✓ 4	result\1	125	0.40	0.85	3	8	0
5	result\1	216	0.31	0.85	2	5	1
6	result\1	275	0.28	0.77	3	5	1
7	result\1	0	1	1	2	8	1
✓ 8	result\1	128	0.40	0.85	3	8	1

# 4.2.5 Other Sweep Features

The Sweep Manager can group parameters and help select Sweep Models.

#### 4.2.5.1 Grouping Parameters

You can group two or more parameters to reduce the number of iterations.

For example, the **Data Rate** and **DCD** parameters can be grouped together as shown in the following examples.

- 1. Select the two rows (for Data Rate and DCD).
- 2. Right-click,Click on Group.

Sweep Element	Sweep Type	Min	Max	Step	Value List	Ste
AMIParameterSweep>Tx1>amiffe>fwd	Parameter	2	3	1		2
TransmitterParameterSweep>Tx1>Da	Parameter				5, 8	2
TransmitterParameterSweep>Tx1>D	Group				0, 1	2
	Delete					

#### The number of **Total Iterations** drops from 8 to 4.

Sweep Element	Sweep Type	Min	Max	Step	Value List	Ste
AMIParameterSweep>Tx1>amiffe>fwd	Parameter	2	3	1		2
⊑ group1						2
TransmitterParameterSweep>Tx1>Da	Parameter				5, 8	2
TransmitterParameterSweep>Tx1>D	Parameter				0, 1	2

#### Originally, the iterations look like this:

Iteration	fwd	Data Rate (Gbps)	DCD (%)
1	2	5	0
2	3	5	0
3	2	8	0
4	3	8	0
5	2	5	1
6	3	5	1
7	2	8	1
8	3	8	1

After grouping the **Data Rate** and **DCD**, the new iterations are:

Iteration	fwd	Data Rate (Gbps)	DCD (%)
1	2	5	0
2	3	5	0
3	2	8	1
4	3	8	1

When combining two or more parameters in a group:

The first value for each parameter in the group constitutes one combination.

The second value for each parameter constitutes the second iteration.

.....

**NOTE!** Within a group, the value in the **Step Count** column of each parameter must be identical. Or else, the **Total Iterations** of the group will be **0**.

#### 4.2.5.2 Model Sweep

The Sweep Manager allows you to perform different types of Model Sweep:

- .Inc sweep Sweep multiple circuit models if the original model is called within a .include statement (such as conn1 and conn2 blocks).
- Model File sweepSweep different parameters defined in the .sp files (such as c\_comp for the Tx1 block).

or spice sub-circuit files for one or more blocks, and sweep them.

Model File

When using the **Model File** sweep, make sure that the connectivity inside each .sp file is the same to maintain everything between \*[MCP Begin] and \*[MCP End].

Click the **New** button to add model files.

veep Mode:	Channel Simulation	Sweep List:	ť	1 🗶 1	Total Iterat	ions: 2	
veep Type:	Models 💌	Sweep Element	Sweep Type	Min Max	Iteration	BackPlane	
Model File	O.Inc OParameter	ModelFileSweep>BackPlane	File		<b>V</b> 1	chan_xaui4_short.sp	
Block	Model file	D:\temp\Signle_channel_complex_SystemSI\chan_xaui			2	chan_xaui4.sp	
BackPlane	D:\temp\Signle_channel_com	D:\temp\Signle_channel_complex_SystemSI\chan_xaui					
conn 1	D:\temp\Signle_channel_com						
:onn2	D:\temp\Signle_channel_com						
ard2	D:\temp\Signle_channel_com						
xpkg	D:\temp\Signle_channel_com						
xpkg	D:\temp\Signle_channel_com						
ard 1	D:\temp\Signle_channel_com						
Tx1	D:\temp\Signle_channel_com						
lx1	D:\temp\Signle_channel_com						
(	111	▲					
uble click on	a component to add it to 'Sweep List'.	Select some sweep items and right click on the list to group, ungrou	p or delete.		Click the ch	eckbox to select or unselect an	iteration.

#### .Inc Sweep

When using **.Inc** sweep make sure that the sub-circuit name inside **.cir** file or other circuit files all have the same name.

Click the **New** button to add circuit files.

weep Mode: Channel S	Smulation -		Sweep List:					2 % 1 1	Total Steral	tions: 2	
weep Type: Models			Sweep Element	<b>Sweep Type</b>	Mn	Max	Step	Value List	Iteration	conn1>hsd5d	
Model File ( .Inc (	Parameter		ModelIncludeFileSweep>corn1>./hsd5de.cir(File: ./conn	File					1 🕅	tx_dummy	
lock: com1									SE 2	tx_bhvr.sp	
Include Command	Circuit File	Line	D:\temp\Signle_channel_complex_SystemST\tx_bhvr.sp								
"/had5de.cr	-\comLsp	24	2								

**Parameter Sweep** 

Weep Mode: C	Dannel Simulation	-		Sweep List:						Total Iteral	tions: 3	
weep Type: N	fodels	•		Sweep Element	Sweep Type	Min	Max	Step	Value List	Iteration	Tx1>bx_scale	
Model File	.inc @Paramete	r		ModeParameterSweep+Tx1>tx_scale(File: D: (temp(Sign)	Parameter	0	1	0.5		1	0	
locic Tx1										17 2 17 3	0.5	
Parameter	Value	Crouit File	Line							(M) 2		
nmos_imp	25	D:\temp\Signle	1									
tx_rt	50	D:\temp\Signle	1							_		
tx_c_comp	lp	D:\temp\Signle	1									
tx_scale	1	D:\temp\Signle	1									
				C. [								

# 4.3 S-Parameter Extraction

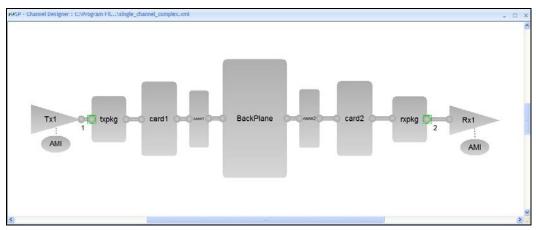
# 4.3.1 Launch Single Channel Complex Template

This exercise uses the same workspace as section *Sweep Manager*.

Click **Open** in the **File** menu to open the **single\_channel\_complex** Template.

Open						? X
Look in:	🚞 single_channe	el_complex	•	G 💋	• 🗈 🏷	
My Recent Documents	history	_complex				
Desktop						
My Documents						
My Computer						
	File name:	single_channel_complex			-	<u>O</u> pen
My Network	Files of type:	SP Files (* xml)			-	Cancel

The **single\_channel\_complex** template workplace is shown below.



The Single Channel Complex template contains the following blocks:

- A Primary Transmitter (**Tx1**)
- A Primary Receiver (**Rx1**)
- Two Packages (**txpkg** and **rxpkg**)
- Two Add-in Cards (card1 and card2)
- Two Connectors (conn1 and conn2)
- A Backplane (**BackPlane**)
- Two AMI Models

## 4.3.2 S-Parameter Extraction

SystemSI - Serial Link Analysis supports **S-Parameter Extraction** for both single-ended nets and differential nets.

Select

Tools > S Parameter Extraction....

Tool	s	Compliance	Window					
	Fre	quency Respo	nse					
	S P	arameter Extra	ction					
	S Parameter Viewer							
Characterize Channel								
Channel Simulation								
Block Sensitivity								
	Jitt	er Tolerance						
:	Sw	eep Manager						
	Mo	del Builder						
	Res	sult Browser						
(	Ор	tions	×					

The S Parameter Extraction window opens.

Iomponent	Circuit	Ground: ngnd	-		# of Frequency Points:	1000 AFS
BackPlane	channel	Ckt Node	Net		Parameter File Name:	S_para
conn1	conn1	posin	pos			
conn2	conn2	negin	neg		Parameter File Format:	bnp 🔽
card2	daughter2	posout	pos			
txpkg	txpkg	negout	neg			
rxpkg	rxpkg	negouc	ney	>>		
card1	daughter1			~		
(			•		Extract	

#### 4.3.2.1 Single-ended Mode

1. Click the **Single-ended Mode** tab.

Component	Circuit	Ground: ngnd	-		# of Frequency Poir	nts: 1000 🗸 AFS
BackPlane	channel	Ckt Node	Net		Parameter File Name	
conn1	conn1	posin	pos			
conn2	conn2	negin	neg		Parameter File Form	at: bnp 💌
card2	daughter2	posout	pos			
txpkg	txpkg	negout	neg			
rxpkg	rxpkg	negou		>>		
card1	daughter 1		l			
(			•		Extract	

2. Select the **BackPlane** component to extract S Parameter.

Differential Mode	Single-ended Mode
Component	Circuit
BackPlane	channel
conn1	conn1
conn2	conn2
card2	daughter2
txpkg	txpkg
rxpkg	rxpkg
card1	daughter 1

3. Set up ports.

3.1 Click **posin** to define positive node.

Ground: ngnd	-
Ckt Node	Net
🕀 posin	pos
negin	neg
posout	pos
negout	neg

3.2 Choose **ngnd** as Ground node.

¢	Ground:	ngnd	•					
	Ckt Noo	ngnd						
	posin							
	negin							

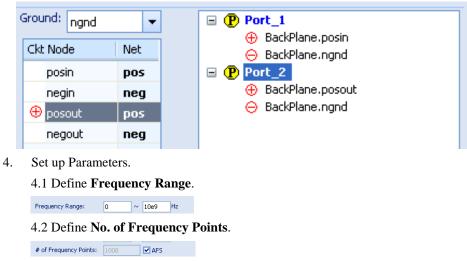
3.3 Click the  $\geq$  button.

Settings for **Port\_1** are completed as the following window shows.

Ground: ngnd	-	Port_1     Pastolese secie
Ckt Node	Net	<ul> <li>BackPlane.posin</li> <li>BackPlane.ngnd</li> </ul>
🕀 posin	pos	
negin	neg	
posout	pos	
negout	neg	

3.4 Repeat Step 3.1 to 3.3 to define the circuit node **posout**.

Settings for **Port\_2** are completed as the following window shows.



4.3 Input Parameter File Name. For example: S\_para.

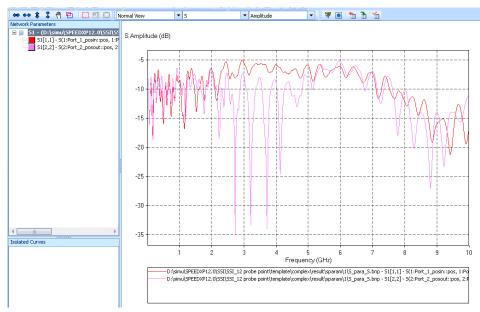
Parameter File S\_para

4.4 Select Parameter File Format: bnp.



5. Click the **Extract** button.

S Parameter curves for Port\_1 and Port\_2 show as the following figure.



S Parameter document is automatically generated in the folder of <working folder path>\result\sparam\.

### 4.3.2.2 Differential Mode

1. Click the **Differential Mode** tab.

S Parameter Extraction	n					×
Differential Mode	Single-ended Mode				Frequency Range:	0 ~ 10e9 Hz
Component		Ground: ngnd	-	Image: Book of the second	# of Frequency Points:	1000 AFS
BackPlane	channel	Ckt Node	Net		Parameter File Name:	5_para
conn1	conn1	posin	pos		Parameter File Format:	bnp
conn2 card2	conn2 daughter2	negin	neg			- Chip
txpkg	txpkg	posout	pos			
rxpkg	rxpkg	negout	neg			
card1	daughter 1					
				>		
		<	•		Extract	
Select a component, a	and then select positive node and					
negative node in the o differential port.	ckt node list. Click '>>' to add a	😑 Right mouse click				
and a second borer						OK Cancel Apply

2. Select the **txpkg** component to extract S Parameter.

Differential Mode S	iingle-ended Mode
Component	Circuit
BackPlane	channel
conn1	conn1
conn2	conn2
card2	daughter2
txpkg	txpkg
rxpkg	rxpkg
card1	daughter1

3. Set up ports.

3.1 Choose **ngnd** as Ground node.



3.2 Click the circuit node **posin** to define positive node.

3.3 Right-click the circuit node **negin** to define negative node.

iround: ngnd 💌		-C- Diff_Channel_Tx1_\$_Rx
Ckt Node		
🕀 posin		
😑 negin		
posout		
negout		
	>>	
		4
<ul> <li>▲</li></ul>		▲ ▶

3.4 Click the  $\geq$  button.

The port **Diff\_Port\_1** is generated automatically.

3.5 Repeat Step 3.1 to 3.3 to define the **rxpkg** component (define **posout** as positive node and **negout** as negative node).

The port **Diff\_Port\_2** will be generated automatically.

S	Parameter Extraction						
	Differential Mode Single	e-ended Mode					
	Component	Circuit	Ground:	ngnd	-		-C- Diff_Channel_Tx1_\$_Rx1
	BackPlane conn1 conn2 card2 txpkg rxpkg card1	channel conn1 conn2 daughter2 txpkg rxpkg daughter1	Ckt No pos new pos	de in	Vet pos neg pos neg	>>	<ul> <li></li></ul>
	< []	<b>&gt;</b>	•		•		■ Port_4
	Select a component, and ther negative node in the ckt node differential port.		-	mouse click t mouse click			
4.	Set up paramete	rs.					
	4.1 Define Freq	uency Range.					
	Frequency Range: 0	~ 10e9 Hz					

#### 4.2 Define No. of Frequency Points.

# of Frequency Points: 1000

4.3 Input Parameter File Name. For example: S\_para.

Parameter File S\_para

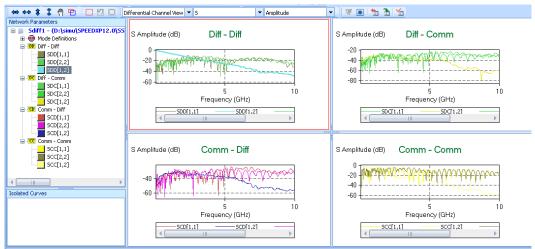
4.4 Select Parameter File Format: bnp.

rameter File Format:	bnp	-
	touchstone	
	bnp	

5. Click the **Extract** button.

Pa

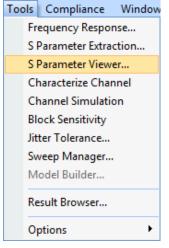
S Parameter curves show as the following figure.



## 4.3.3 S-Parameter View

1. Select

Tools > S Parameter Viewer.





Port Curves				
*******	Normal View	<b>▼</b> 5	<ul> <li>Amplitude</li> </ul>	• • • • • • •
Vetwork Parameters				
	_			
olated Curves				
	_			

- 2. Right-click the **Network Parameters** pane.
- 3. Select **Load** in the pop-up menu list.

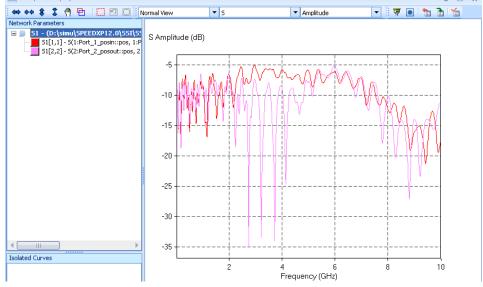
Port Curves					- 🗆 X
*******	Normal View	▼ S	<ul> <li>Amplitude</li> </ul>	- 🛛 🐑	3 5
Network Parameters	Result Browser				
	Load Unload All Networks				
Isolated Curves					

The **Open** window opens.

Open					? X
Look jn:	i 3	•	G 🦻	<del>ب</del>	
My Recent Documents	S_para_S.bnp SParamExtracti	on.sp			
Desktop					
My Documents					
My Computer					
	File <u>n</u> ame:	S_para_S.bnp		-	<u>O</u> pen
My Network	Files of type:	All Curve Files (*.bnp, *.bds, *.ts, *.Sr	nP)	- (	Cancel

- 4. Select an S Parameter, e.g. **S\_para\_S.bnp**.
- 5. Click Open.

The curve shows in the result pane like the following figure.



4.4 S-Parameter Wrapping and BBS Integration

# 4.4.1 Add S Parameter Block

 Click the Add S Parameter Block icon S on the tool bar, and click in the Layout window. The S Parameter block S1 is added.

SystemSI - [Serial Link Analysis : D:\W	orking\	.SystemS\10gbase_test1.ssix]
₩ File Edit View Setup To	ools C	Compliance Window Help
🗋 💕 🚽 🧳 🕶 🔛	10	🖸 !    🕨 🔳 ! 🤜 🖀 🗙 🖬 ! == 🖪 @ @ @    🕨 🖉 🐘 🚳 🧐 🕪
Workflow: SystemSI	×	Add S Parameter Block
Serial Link Analysis	×	
SFP+ Compliance	×	
HDMI Compliance	×	S
PCIe 3 Compliance	∛	5
10GBASE-KR Compliance	*	S1
Channel Setup		
Choose a Template		
Edit Channel Models Set AMI Parameters		

2. Double-click the **S1** block.

The **Property** pane opens.

erty k Name: S1				File Name:Sub-circuit Name:
Connection	Connect To	Block	Block Connection	
	111			Extract BBS Model Load S Parameters Edit Sub-circuit Defin
onnection				
				OK Cancel Apr

# 4.4.2 Load S Parameter File

1. Click the **Load S Parameters...** button.

The Load S Parameters window opens.

Load S Parameters						×
💮 🌍 – <u></u> « Loo	al Disk	(D:) • Working • SystemSI • 10gbase_test1	► <b>▼</b> 49	Search 10gbase_test1		9
Organize 🔻 Nev	v folde	r				2
E Desktop     Downloads     Recent Places	^	Name history	Date modified 5/2/2013 3:29 PM		Size	
🧟 My Site		Image: Image	4/9/2013 4:27 PM 7/21/2012 2:30 A		4	414 K
<ul> <li>□ Libraries</li> <li>□ Documents</li> <li>↓ Music</li> <li>□ Pictures</li> <li>□ Videos</li> </ul>	Е					
🖳 Computer 🏝 Local Disk (C:) 🦲 Local Disk (D:)						
Sa Network	Ŧ	•	III			Þ
	File <u>n</u> a	ame:	T	S Parameter File(*.bnp; *.s	s?p; *.e ▼ ancel	

2. Load the S Parameter file (BNP or Touchstone) to the S Parameter block.

The kr\_test\_channel.BNP file is used as an example in this part.

Once the selected S Parameter file is successfully loaded, an .sp file will be automatically generated and loaded to the block.

- The .sp file will be displayed in the File Name field
- The S Parameter file will be displayed in the S Parameter File field

File Name: D:	:\Working\SystemSI\10gbase_test: Sub-circuit Name: S1	_kr_test_channel 💌
S Parameter F	ile: D:\Working\SystemSI\10gbase_test1\kr_test_channel.BNP	
.SUBCKT + + + +	S1_kr_test_channel_BNP 1 ngnd 2 3	
**************************************		013
* [Power Nets * [Ground Nets * [Signal Nets * * [MCP End] *	ts]	
	les the MCP section s is the SystemSI generated sub-circuit definition for the S Param	eter File:
Extract BBS N	Model ] Load BBS Model ] Load S Parameters ] Edit S	ub-circuit Definition

• For the BNP file, if the BNP file has the MCP information, the MCP section will be automatically added to the .sp file for connection

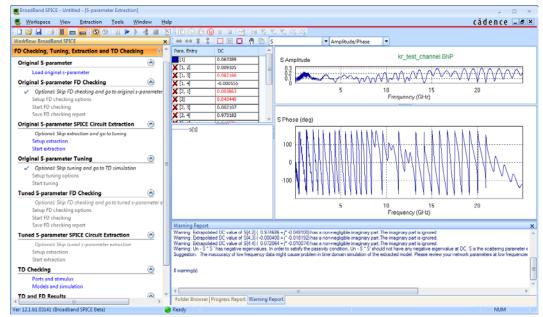
• For the Touchstone file, or the BNP file which does not have the MCP information, you are expected to manually add the MCP through the MCP Editor

**NOTE!** Editing of .sp file for the S Parameters is NOT recommended.

## 4.4.3 Extract the BBS Model

1. Click the Extract BBS Model... button.

The BroadbandSPICE application is launched.



2. Click the **FD Checking, Tuning, Extraction and TD Checking** workflow to check and tune the S Parameters.

For the application of BroadbandSPICE, please refer to *BroadbandSPICE\_Tutorial.pdf* and *BroadbandSPICE\_UG.pdf*.

3. If the S Parameter checking result looks good to you, click **Setup extraction** in the workflow.

riginal S-parameter	<u></u>
Load original s-parameter	
riginal S-parameter FD Checking	$\bigcirc$
Optional: Skip FD checking and go to original	s-param
Setup FD checking options	
Start FD checking	
Save FD checking report	_
riginal S-parameter SPICE Circuit Extraction	
Optional: Skip extraction and go to tuning	
Setup extraction	
Start extraction	-
riginal S-parameter Tuning	<u></u>
Optional: Skip tuning and go to TD simulation	n
Setup tuning options	
Start tuning	_
uned S-parameter FD Checking	<u></u>
Optional: Skip FD checking and go to tuned s	-paramet
Setup FD checking options	
Start FD checking	
Save FD checking report	_
uned S-parameter SPICE Circuit Extraction	<u></u>
Optional: Skip tuned s-parameter extraction	
Setup extraction	
Start extraction	
D Checking	<u></u>
Ports and stimulus	
Models and simulation	
D and FD Results	<u></u>
TD results	
FD results	

The **Options** window opens.

Options			
General File Manager Simulation (Basic)	Change the 'Extraction settings' options in BroadBand SPICE		
General Report template	Highlight Error		
Settings	Highlight Errors Greater Than : 9.02		
Checking settings	Extraction Mode		
Tuning settings	Passivity mode     Maximum number of iterations for passivity enforcement:     200     Precision mode		
	BBS Circuit		
	HSPICE Compatible     General SPICE Compatible		
	File Name : [kr_test_channel_BBSckt.txt		
	S-parameter of BBS Circuit		
	Export O Touchstone Format		
	File Name : kr_test_channel_BBSckt_sp.s4p		
	Model Order Reduction		
	Ignore off-diagonal S parameters that are less than:		
	Don't use reduction on these ports: Preserved Port(s)		
	Reduce upper frequency limit: 25 GHz		
	MCP File		
	File Name :		
	Default Apply OK Cancel		

- 4. Set up the extraction settings as desired.
- Click Start extraction in the workflow to extract the BBS model.
   If the BBS model extraction is successful, a .txt file will be automatically generated.
- Click the Load BBS Model... button to load the generated BBS model kr test channel BBSckt.txt.

	_				
များLoad BBS Model					×
💮 🌍 🗕 📔 « Syste	emSI	▶ 10gbase_test1 → BBSResult_kr_test_channel	► <b>▼</b> 4	Search BBSResult_kr_t	est_chan 🔎
Organize 🔻 New	folde			•=== •	
🔆 Favorites	-	Name	Date modified	Туре	Size
🧮 Desktop		鷆 temp	5/2/2013 4:44 PM	File folder	
鷆 Downloads		kr_test_channel_BBSckt.txt	5/2/2013 4:47 PM	Text Document	217 K
🖳 Recent Places		kr_test_channel_BBSckt_2.txt	5/2/2013 4:47 PM	Text Document	162 K
🧟 My Site		kr_test_channel_for_RFM.txt	5/2/2013 4:47 PM	Text Document	1 K
		kr_test_channel_GSPICE.txt	5/2/2013 4:47 PM	Text Document	1,123 K
🥽 Libraries	=				
Documents					
J Music					
Pictures					
H Videos					
🖳 Computer					
Local Disk (C:)					
👝 Local Disk (D:)	-	٠			•
F	ile <u>n</u> a	me: kr_test_channel_BBSckt.txt	▼ BB	5 Model File(*.txt)	•
				<u>O</u> pen C	Cancel

Once the BBS model is successfully loaded, an .sp file will be automatically generated and loaded to the block.

- The .sp file will be displayed in the File Name field
- The BBS model .txt file will be displayed in the BBS Model File field

File Name: D:\Working\SystemSI\10gbase_test:
BBS Model File: D:\Working\SystemSI\10gbase_test1\BBSResult_kr_test_chi
.SUBCKT     S1_kr_test_channel_BNP       +     1       +     ngnd
+ 2 + 3
The following is the Cadence MCP(model connection protocol) Section
*[MCP Begin] *[MCP Ver] 1.1 *[MCP Source] Cadence Design Systems, Inc. SystemSI 12.1.b1.04022 5/2/2013
*[Power Nets] *[Ground Nets] *[Signal Nets] *
*[MCP End] *
*This concludes the MCP section
Load BBS Model Edit Sub-circuit Definition

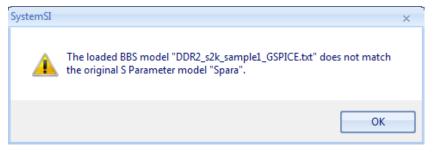
NOTE!	Editing of .sp file for the BBS model is NOT recommended.
-------	---

# 4.4.4 Load the BBS Model

If you have the BBS models for the selected S Parameters, click the **Load BBS Model...** button to directly load the BBS .txt file.

Please refer to Step 6 in Section 4.4.3 Extract the BBS Model for details.

If the loaded BBS model does not match the original S Parameters, an error message will be issued.



# 4.4.5 Switch the Models

1. Click the Go To S Parameters button to switch to the original S Parameters model.

File Name: D:	\Working\SystemSI\10gbase_test:	o-circuit Name: S1_kr_test_channel
BBS Model File:	D:\Working\SystemSI\10gbase_test1\BBSR	Result_kr_test_chi
.SUBCKT + + + + +	S1_kr_test_channel_BNP 1 ngnd 2 3 4	
*The following	is the Cadence MCP(model connection proto	ocol) Section
*	1 ] Cadence Design Systems, Inc. SystemSI 1:	2.1.b1.04022 5/2/2013
*[Power Nets] *[Ground Nets *[Signal Nets] *	5]	
*[MCP End] *		
*This conclude	es the MCP section	
•	111	•
Load BBS Mode	4]	Edit Sub-circuit Definition

2. Click the Go To BBS button to switch back to the BBS model:

File Name: D:\Working\SystemSI\10gbase_test: Sub-circuit Name: S1_kr_test_channel
S Parameter File: D:\Working\SystemSI\10gbase_test1\kr_test_channel.BNP Go To BBS
.SUBCKT S1 kr_test_channel_BNP
+ 1
+ ngnd
+ 2
+ 3
+ 4
*The following is the Cadence MCP(model connection protocol) Section
*[MCP Begin] *[MCP Ver] 1.1 *[MCP Source] Cadence Design Systems, Inc. SystemSI 12.1.b1.04022 5/2/2013 *
***************************************
*[Power Nets] *[Ground Nets] *[Signal Nets] *
*[MCP End] *
*This concludes the MCP section
* NOTE - This is the SystemSI generated sub-circuit definition for the S Parameter File:
Extract BBS Model Load BBS Model Load S Parameters Edit Sub-circuit Definition

**NOTE!** The MCP section will be shared among the BBS models and the original S Parameter model. Any MCP change to one .sp file through the MCP Editor will automatically update the MCP section in other .sp files.

# 4.5 Block Sensitivity

Understanding the contribution of different system components to Jitter and Noise, and subsequently tuning these components, are key to a successful design.

The eye opening is one of the well understood and useful metrics used to assess system performance. The conventional scheme for the eye opening is inconsistent; only the time scale is normalized.

NOTE!	Block Sensitivity analysis requires SystemSI to automatically short out blocks. This is dependent on the MCP signal names in the models associated with the blocks. For a given block, for each terminal in the model, SystemSI needs to be able to find an associated terminal with the same MCP signal name, in order to correctly place a shorting element between the 2 terminals. This requires that "thru" signals in the model have the same MCP signal name associated with its
	terminals.

# 4.5.1 Normalized Jitter and Noise

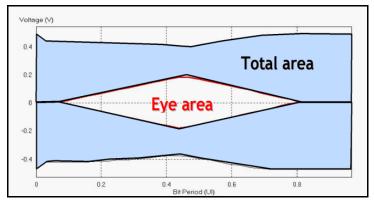
The **Block Sensitivity** feature uses a novel eye-area-based normalized Jitter and Noise (NJN) metric. The goal is to find out the effect of each system component on Jitter and Noise.

The NJN is obtained by using the entire area of the eye (**Total area**) as the normalizing parameter. Normalized Jitter and Noise is defined as:

NJN = 1- (Eye area/Total area)

This diagram shows the Total Area and Eye Area.

The **Total Area** includes the **Eye area**. The **Eye area** in this NJN metric reflects deterministic Jitter contribution.



A subtractive methodology is used to examine each component's contribution.

a. The entire topology is simulated to establish a baseline.

- b. A number of sweeps are automatically run.
- c. Each block is sorted in turn to quantify its effect on the overall result.

d. The relative contribution of each block is tabulated in terms of normalized Jitter and Noise (NJN).

e. The contributions of each block are displayed in bar graph.

A non-zero eye opening is required to run the **Block Sensitivity** feature at any data rate and get meaningful results.

At a high Data Rate using equalization (such as **amiffe**) may be necessary to meet the requirement.

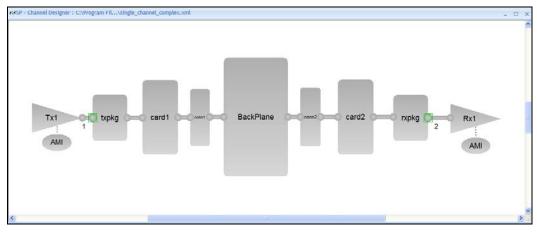
## 4.5.2 Launch Single Channel Complex Template

This exercise uses the same workspace as section Sweep Manager.

Click **Open** in the **File** menu to open the **single\_channel\_complex** Template.

Open							? X
Look in:	🚞 single_channe	l_complex	-	6	1 🖻	•	
My Recent Documents	history result single_channel_	_complex					
Desktop							
My Documents							
My Computer							
	File name:	single_channel_c	complex		-		<u>O</u> pen
My Network	Files of type:	SP Files (* xml)			-		Cancel

The **single\_channel\_complex** Template workplace is shown below.



The Single Channel Complex Template contains the following blocks:

- A Primary Transmitter (**Tx1**)
- A Primary Receiver (**Rx1**)
- Two Packages (txpkg and rxpkg)
- Two Add-in Cards (card1 and card2)
- Two Connectors (conn1 and conn2)
- A Backplane (**BackPlane**)
- Two AMI Models

## 4.5.3 Set up Simulation Parameters

This exercise uses most of the default parameters of the template. The Tx1 data rate is 10 Gbps. Equalization is needed to get an open eye.

We are using the results of the single sweep in the Sweep Manager section.

Set the number of forward tabs for the **amiffee** AMI model to 3 when you are running a single sweep.

This setting is necessary to get any eye opening.

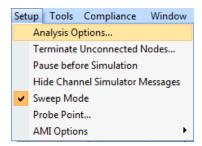
1. Double-click the AMI block connected to the Tx1 block to view its parameters.

AMI Parameter:	
(amiffe (fwd <u>3</u> ) (pre <u>1</u> )	Click the number to change the value
( coeffout <u>nil</u> ) ( UserTapsFile <u>nil</u> ) )	

2. Change the default **fwd** parameter to 4.

AMI Paran	neter:
( amiffe	
	(fwd <mark>4</mark> )
	(pre <u>1</u> )
	(coeffout <u>nil</u> )
	( UserTapsFile <u><b>nil</b></u> )
)	

3. In the Setup menu, click Analysis Options....



#### Circuit Simulator

F	-Characterization-	
	Circuit Simulator	
	OHSPICE	
	● SPDSIM	

**Circuit Simulator Options** 

* Add global .option and .include commands here. * They'll be used for time domain characterization. *.option delmax=1p	Circuit Simulator Options:
	* They'll be used for time domain characterization.

Make sure that the chosen circuit simulator is SPDSIM without .option delmax set.

### 4.5.4 Start Block Sensitivity

In the **Tools** menu, click

Block Sensitivity.

Once you start **Block Sensitivity**, the tool immediately prompts you to confirm the number of simulations to be run.

The channel has seven blocks. The number of simulations will be 7 + 1 = 8. The first simulation is used as the reference.

SP		×
?	This will require	8 simulations. Continue?
	ОК	Cancel

A net name must be associated with each node in the channel. If not, the Block Sensitivity will not work. You can assign Net names from the **Block Property** panes or from the **MCP Editor**.

In the following image the second column shows which block is being sorted.

ck Sensiti	vity Result:		Export Sho	w Result		Block Sensitivity Analysis
teration	Block Shorted	NJN	NJN Contribution (%)			
1	no block shorted					
2	BackPlane					
3	conn1				(%) N/N	
4	conn2				z	
5	card2				2	
6	txpkg				\$	
7	rxpkg				5	
8	card1				Ŧ	
					Contribution to	
					Ē	
					Ö	
k 'Show B	ecult' button to show the r	ecults of the checked iten	ns, or double click on an item to show	v the result		Block Name

## 4.5.5 View and Analyze the Results

After the simulations are done, the results appear as shown in the following illustrations.

Iteration	Block Shorted	NIN	NJN Contribution (%)	20
1	no block shorted	0.9267654751525719		
2	BackPlane	0.7674172185430463	17.19	. %) -
3	conn1	0.9312344656172328	-0.48	
4	conn2	0.9351932577739029	-0.91	
5	card2	0.9197059655074922	0.76	
6	txpkg	0.9267654751525719	0.00	e e e e e e e e e e e e e e e e e e e
7	rxpkg	0.9267654751525719	0.00	BackPlane conn1 conn2 card2 txpkg rxpkg card1
8	card1	0.9296979417268110	-0.32	BadeFlane conn1 2011172 card2 topig npig card1
				S S
				-10
				Block Name

#### 4.5.5.1 Block Sensitivity Result Pane

The Block Sensitivity Result pane has four columns:

- **Iteration** Lists the number of iterations in an ascending order.
- **Block Shorted** Lists which block is being shorted, with the first one used as the reference (no block shorted).
- NJN Normalized Jitter and Noise Results. Obtained from the formula:

NJN = 1- (Eye area/Total area).

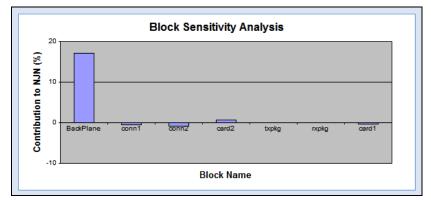
• NJN Contribution - Ratio of the NJN of the shorted block to the NJN when no blocks are shorted. Obtained from the formula:

NJN Contribution = [1- (NJN (shorted block) / NJN (no block shorted))]\*100

Iteration	Block Shorted	NJN	NJN Contribution (%)
1	no block shorted	0.9267654751525719	
2	BackPlane	0.7674172185430463	17.19
3	conn1	0.9312344656172328	-0.48
4	conn2	0.9351932577739029	-0.91
5	card2	0.9197059655074922	0.76
6	txpkg	0.9267654751525719	0.00
7	rxpkg	0.9267654751525719	0.00
8	card1	0.9296979417268110	-0.32

### 4.5.5.2 Block Sensitivity Analysis

The Block Sensitivity Analysis chart displays NJN Contribution results.



Click on the column header NJN Contribution (%) to sort the results in ascending or descending order.

The largest contribution to Jitter and Noise comes from the **BackPlane**.

Multiple contributions are negative, which indicates that the eye gets worse when the component is shorted. It is difficult to predict what will happen when components are shorted. The **NJN Contribution** of **txpkg** and **rxpkg** blocks is 0% because these blocks already have dummy models in the default template.

#### 4.5.5.2.1

#### **To Show Results**

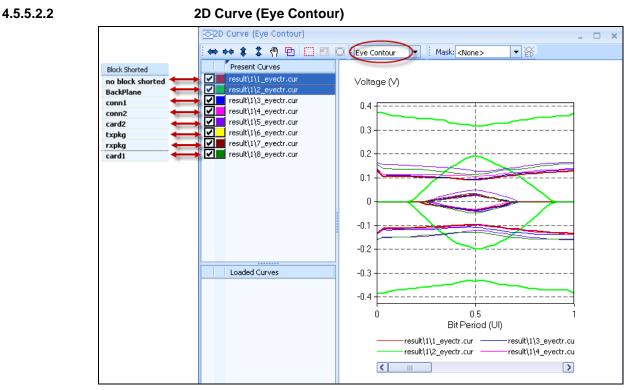
- 1. Right-click in the list to display all the curves. A pop-up menu opens.
- 2. Choose **Select All** in the pop-up menu.
- 3. Click the button Show Result...

Iteration	n Block Shorted		NJN	NJN Contribution (%)
1	no block shorted		0.9267654751525719	
2	BackPlane		0.7674172185430463	17.19
3	conn1		0.9312344656172328	-0.48
4	conn2		0.9351932577739029	-0.91
5	card2	Select /	All I	0.76
6	txpkg	Deselec	+ 41	0.00
7	rxpkg			0.00
8	card1	Select All Highlighted Items		-0.32
		Deselec	t All Highlighted Items	

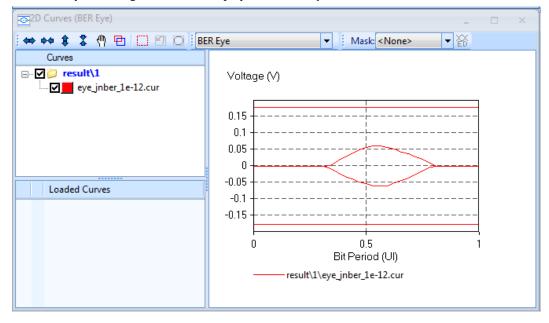
4. The Show Simulation Result window opens.

Show Simulation Result 🛛 🗙
Option
Show in the same windows
Select Result
✓ Eye Density
Eye Contour
✓ Bathtub
☑ Noise Bathtub
Ramp Response
✓ Impulse Respons
Modified Impulse Response
Rx Waveform
✓ Report
Select All Deselect All
OK Cancel

See the following pages for more curves.

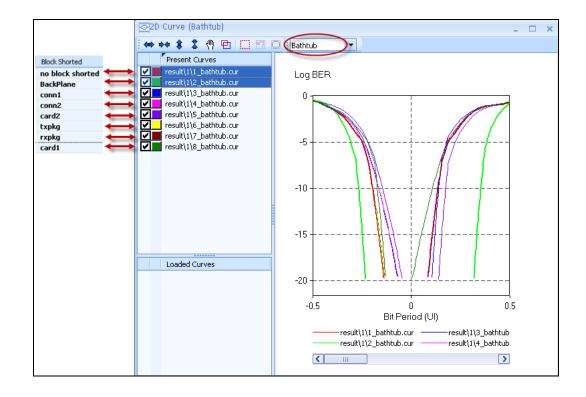


If **Both time and voltage** is checked in the **Analysis Options** window, a new curve for the JN\_BER\_Eye will be generated and displayed as **BER Eye**.



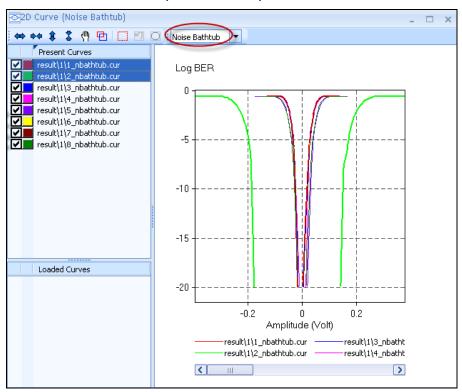


2D Curve (Bathtub)



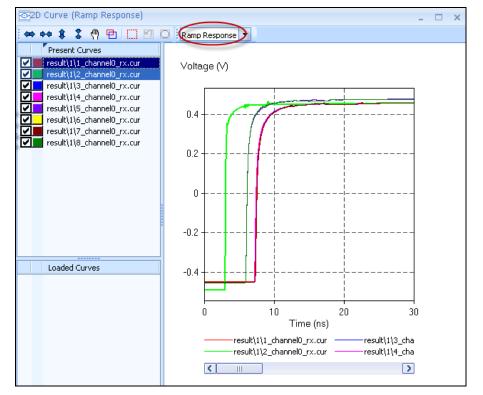


#### 2D Curve (Noise Bathtub)



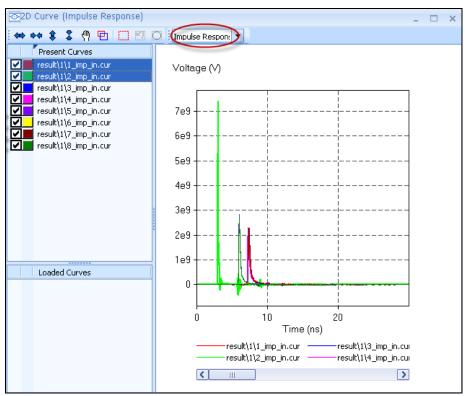


#### 2D Curve (Ramp Response)



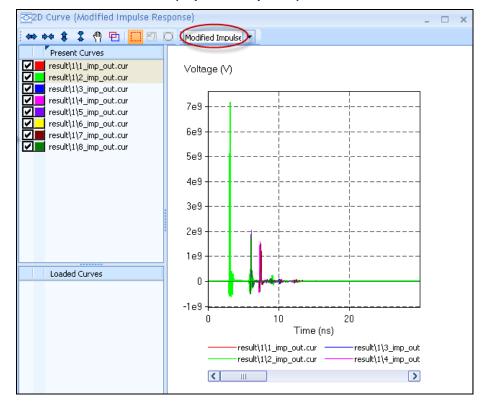
#### 4.5.5.2.6

#### 2D Curve (Impulse Response)



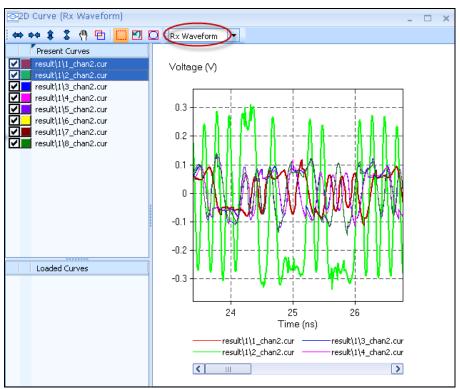
#### 4.5.5.2.7

#### 2D Curve (Impulse Response)



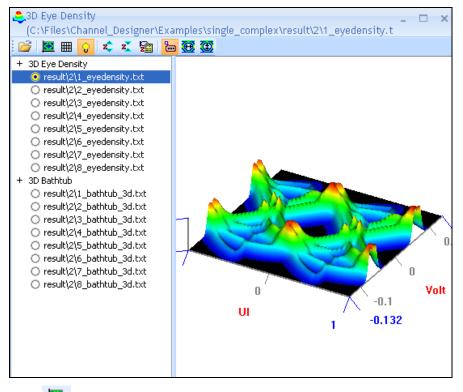
#### 4.5.5.2.8

#### 2D Curve (Rx Waveform)



#### 4.5.5.2.9

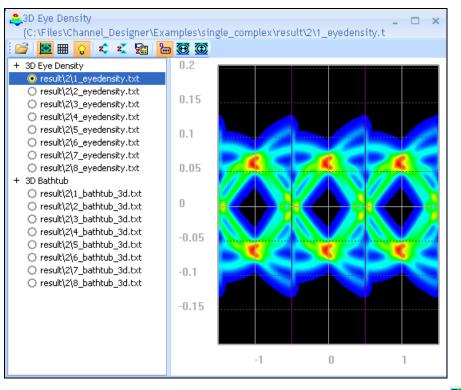
3D Eye Density



Click to view a **Top View** of the eye,

4.5.5.2.10

#### 3D Eye Density



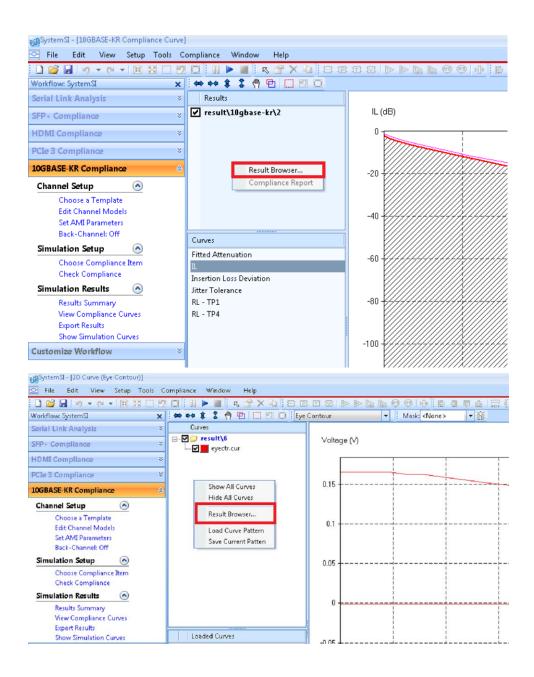
You can scroll the middle mouse button to zoom in and out, and you can also click  $\textcircled{\textcircled{0}}$  to add measurement cursors.

# 4.6 Result Browser and 2D Curve Presentation

### 4.6.1 Result Browser

The **Result Browser** option is available for all the curve windows, such as 2D Curve (Time Variation), 2D Curve (Frequency Response), S Parameter Viewer, Jitter Tolerance, Compliance Curve, 3D Eye Density, and Report View.

You can right-click in the curve panel, and choose **Result Browser...** from the pop-up menu to view the results. As shown in the following examples:



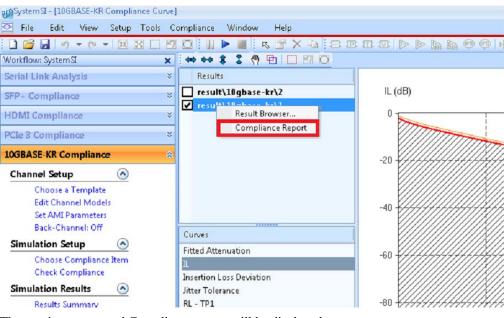
SystemSI - [Report View]			
land		ompliance Window Hel;	
Workflow: SystemSI	x	Report	Channel Report
Serial Link Analysis	×	result\6\report.txt	Mon Nov 26 08:10:23 2012
SFP+ Compliance	×		General:
HDMI Compliance	¥	Load	Data Rate
PCIe 3 Compliance	×	Result Browser	Number of Bits
10GBASE-KR Compliance	۵		Measurement Delay Channel Coding
Channel Setup			Primary Driver Data Pattern
Choose a Template			Number of Aggressors
Edit Channel Models			Characterization Data
Set AMI Parameters			Delay
Back-Channel: Off			Intra-Pair Skew
Simulation Setup			
Choose Compliance Item			

With this option, all the previous results under the **result**, **history** and **result**(**bat**) folders can be browsed and displayed from various curve windows.

Results Browser X		
Folder	Simulation Name	Simulation Type
bistory		
Image: State		
	1\1 1\2	Channel Simulation
3	1\3	Channel Simulation
	1\4	Channel Simulation
B 🔲 🖟 sweep		
Double click on an item to show the result; or select multiple ite	ms to display and click 'Show Results'.	Show Results Close

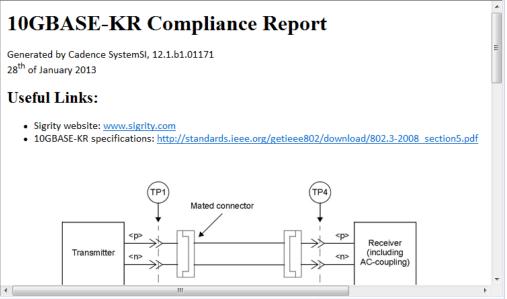
### 4.6.2 Compliance Curve Window

1. Right-click in the **Results** panel, and choose **Compliance Report** from the pop-up menu.



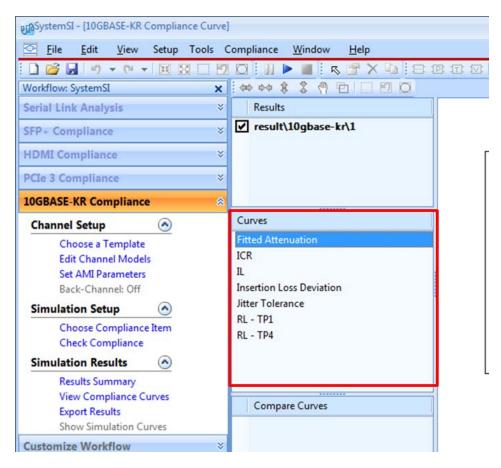
The previous generated Compliance report will be displayed.





This works for all four SLA Compliance kits.

2. In the **Curves** panel, click the desired Compliance plot to view the plot.



### 4.6.3 Curve Pane Context Menu

To display the context menu, right-click on the Curve window.

Measure	
Marker	
Expression Calculator.	
Save	
Export To Excel	
Export Bitmap File	
Black Background	
Embed-Ctrl Visibility	Þ
Embed-Ctrl Position	•
Ctrl Bar Position	•
BarChart	
Auto Tip	

You can perform the following operations in this menu.

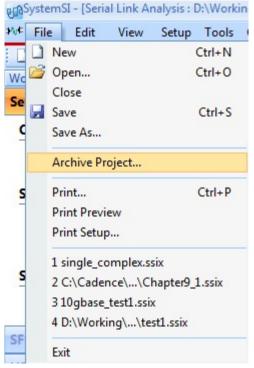
- Measure Toggle the horizontal and vertical measure lines.
- Marker Toggle the horizontal and vertical marker lines.
- **Expression Calculator**—Setup and calculation the expression.
- **Save**—Save the curve.

- **Export To Excel**—Export the curve to Excel.
- **Export to Bitmap File**—Export the curve to Bitmap file.
- **Black / White Background** Set the background of the curve window to be black or white.
- **Embed-Ctrl Visibility** Set the visibility of the sub windows (for example, the legend bar) in the display area.
- Embed-Ctrl Position Toggle the sub windows between floating and docking.
- Ctrl Bar Position If a sub window is docked, change the position of the docking.
- **Bar Chart** Toggle the plot style between a bar chart and a continuous line.
- Auto Tip Show / Hide the tip of the objects in the Curve window when moving the mouse.
- Add Annotation Add a text string in the Curve window.

All these common items are available for 2D Curve (Time variation), 2D Curve (Frequency Response), Compliance Curve, and Jitter Tolerance. Each 2D Curve may have additional items which are unique to the specific 2D Curve window.

# 4.7 Auto Archive SSI Project

1. Choose File > Archive Project....



The Archive Project window opens.

2. Specify **Project Location** and **Project Name** for the copied project.

Project Location:	D:\Working\SystemSI	 Project Name:	single_complex_copy
<b>D</b> 1 6 1			

By default,

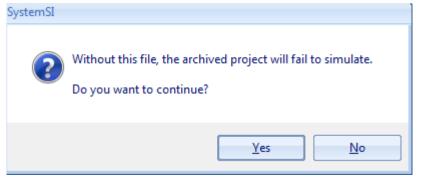
- The location of the original project is selected for the copied project location
- <original\_project\_name>\_copy is set for the copied project name

3. Check the files that should be included in the copied project.

By default,

- The files under the original project folder and under the original sub-folders are listed and checked for archiving, and will be copied to the new project folder
- The files used by the original project but located other than the original project folder are listed and checked, and they will be copied to a sub-folder named **Ref** under the new project folder
- The files under the **history**, **result**, and **result(bat)** sub-folders are not listed

If you want to uncheck a file used by the original project, the following message opens.



- 4. To add additional file, click the **Add File...** button.
- 5. To add additional folder, click the **Add Folder...** button.
  - The files checked and used by the original project are highlighted in blue

chive Project	•
Project Location: D:\Working\SystemSI	Project Name: single_complex_copy
Original File	Target File
D:\Working\SystemSI\single_complex\chan_xau	chan_xaui4.sp
D:\Working\SystemSI\single_complex\channel	channel_bga.s4p
D:\Working\SystemSI\single_complex\channel	channel_rlc.txt
D:\Working\SystemSI\single_complex\channel	channel_rlc_BBS.sp
D:\Working\SystemSI\single_complex\conn1.sp	conn1.sp
D:\Working\SystemSI\single_complex\conn2.sp	conn2.sp
D:\Working\SystemSI\single_complex\daughter	daughter 1.sp
D:\Working\SystemSI\single_complex\daughter	daughter2.sp
D:\Working\SystemSI\single_complex\hsd5ab.cir	hsd5ab.cir
D:\Working\SystemSI\single_complex\hsd5de.cir	hsd5de.cir
D:\Working\SystemSI\single_complex\hsd6ab.cir	hsd6ab.cir
D:\Working\SystemSI\single_complex\hsd6de.cir	hsd6de.cir
D:\Working\SystemSI\single_complex\hsd8ab.cir	hsd8ab.cir
D:\Working\SystemSI\single_complex\hsd8de.cir	hsd8de.cir
D:\Working\SystemSI\single_complex\hsd8gh.cir	hsd8gh.cir
D:\Working\SystemSI\single_complex\pkg_diff0	pkg_diff0.rlc
D:\Working\SystemSI\single_complex\rx_bhvr.sp	rx_bhvr.sp
D:\Working\SystemSI\single_complex\rx_ibis.sp	rx_ibis.sp
D:\Working\SystemSI\single_complex\rx_pkg.sp	rx_pkg.sp
D:\Working\SystemSI\single_complex\scd_exa	scd_example.ibs
D:\Working\SystemSI\single_complex\single_co	single_complex_copy.ssix
D:\Working\SystemSI\single_complex\strip.rlc	strip.rlc
D:\Working\SystemSI\single_complex\tx_bhvr.sp	tx_bhvr.sp
D:\Working\SystemSI\single_complex\tx_ibis.sp	tx_ibis.sp
D:\Working\SystemSI\single_complex\tx_pkq.sp	tx_pkq.sp
Add File Add Folder Uncheck All	Zip Project Archive Cancel

• The files not used by the original project are not highlighted, and they can be deleted from the list

Are	chiv	e Project		×
F	Proje	ect Location: D:\Working\SystemSI	Project Name: single_complex_copy	
		Original File	Target File	
	1	D:\Working\SystemSI\single_complex\chan_xau	chan_xaui4.sp	
	V	D:\Working\SystemSI\single_complex\channel	channel_bga.s4p	
	V	D:\Working\SystemSI\single_complex\channe De	elete el_ric.txt	
	1	D:\Working\SystemSI\single_complex\channel	channel_rlc_BBS.sp	
	1	D:\Working\SystemSI\single_complex\conn1.sp	conn1.sp	
	1	D:\Working\SystemSI\single_complex\conn2.sp	conn2.sp	
	V	D:\Working\SystemSI\single_complex\daughter	daughter 1.sp	
	1	D:\Working\SystemSI\single_complex\daughter	daughter2.sp	
	<b>V</b>	D:\Working\SystemSI\single_complex\hsd5ab.cir	hsd5ab.cir	
	1	D:\Working\SystemSI\single_complex\hsd5de.cir	hsd5de.cir	
	<b>V</b>	D:\Working\SystemSI\single_complex\hsd6ab.cir	hsd6ab.cir	-
	1	D:\Working\SystemSI\single_complex\hsd6de.cir	hsd6de.cir	
	<b>V</b>	D:\Working\SystemSI\single_complex\hsd8ab.cir	hsd8ab.cir	
	<b>V</b>	D:\Working\SystemSI\single_complex\hsd8de.cir	hsd8de.cir	
	1	D:\Working\SystemSI\single_complex\hsd8gh.cir	hsd8gh.cir	
	<b>V</b>	D:\Working\SystemSI\single_complex\pkg_diff0	pkg_diff0.rlc	
	<b>V</b>	D:\Working\SystemSI\single_complex\rx_bhvr.sp	rx_bhvr.sp	
	<b>V</b>	D:\Working\SystemSI\single_complex\rx_ibis.sp	rx_ibis.sp	
	<b>v</b>	D:\Working\SystemSI\single_complex\rx_pkg.sp	rx_pkg.sp	
	V	D:\Working\SystemSI\single_complex\scd_exa D:\Working\SystemSI\single_complex\single_co	scd_example.ibs single complex copy.ssix	
	V	D:\Working\SystemSI\single_complex\strip.rlc	single_complex_copy.ssix strip.rlc	
	V	D:\Working\SystemSI\single_complex\sub.ric D:\Working\SystemSI\single_complex\tx_bhvr.sp	tx bhvr.sp	
	V	D:\Working\SystemSI\single_complex\tx_brivi.sp	tx ibis.sp	
	V	D:\Working\SystemSI\single_complex\tx_bkg.sp	tx_pkg.sp	-
		dd File Add Folder Uncheck All	✓ Zip Project Cancel	5

• The files are highlighted in red if they are used by the original project but does not exist or are un-checked

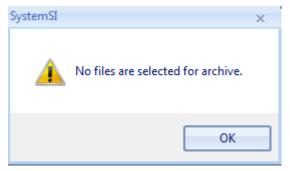
Archive Project	□ ×
Project Location: D:\Working\SystemSI	Project Name: single_complex_copy
Original File	Target File
D:\Working\SystemSI\single_complex\chan_xau	chan_xaui4.sp
D:\Working\SystemSI\single_complex\channel	channel_bga.s4p
D:\Working\SystemSI\single_complex\channel	channel_rlc.txt
D:\Working\SystemSI\single_complex\channel	channel_rlc_BBS.sp
D:\Working\SystemSI\single_complex\conn1.sp	conn1.sp
D:\Working\SystemSI\single_complex\conn2.sp	conn2.sp
D:\Working\SystemSI\single_complex\daughter	daughter 1.sp
D:\Working\SystemSI\single_complex\daughter	daughter2.sp
D:\Working\SystemSI\single_complex\hsd5ab.cir	hsd5ab.cir
D:\Working\SystemSI\single_complex\hsd5de.cir	hsd5de.cir
D:\Working\SystemSI\single_complex\hsd6ab.cir	hsd6ab.cir 🗧
D:\Working\SystemSI\single_complex\hsd6de.cir	hsd6de.cir
D:\Working\SystemSI\single_complex\hsd8ab.cir	hsd8ab.cir
D:\Working\SystemSI\single_complex\hsd8de.cir	hsd8de.cir
D:\Working\SystemSI\single_complex\hsd8gh.cir	hsd8gh.cir
D:\Working\SystemSI\single_complex\pkg_diff0	pkg_diff0.rlc
D:\Working\SystemSI\single_complex\rx_bhvr.sp	rx_bhvr.sp
D:\Working\SystemSI\single_complex\rx_ibis.sp	rx_ibis.sp
D:\Working\SystemSI\single_complex\rx_pkg.sp	rx_pkg.sp
D:\Working\SystemSI\single_complex\scd_exa	scd_example.ibs
D:\Working\SystemSI\single_complex\single_co	single_complex_copy.ssix
D:\Working\SystemSI\single_complex\strip.rlc	strip.rlc
D:\Working\SystemSI\single_complex\tx_bhvr.sp	tx_bhvr.sp
D:\Working\SystemSI\single_complex\tx_ibis.sp	tx_ibis.sp
D:\Working\SystemSI\single complex\tx pkq.sp	tx pkq.sp
Add File Add Folder Uncheck All	Zip Project Cancel

### **NOTE!** If there are any files highlighted in red, the copied project will fail to simulate.

- 6. Click the Archive button to generate the copied project.
  - All the checked files will be copied to the copied project
  - If the **Zip Project** option is checked, a zipped project file will be created. Otherwise, a new project folder will be created for the copied project

The following messages may appear while archiving:

• If no files are checked, the following message opens

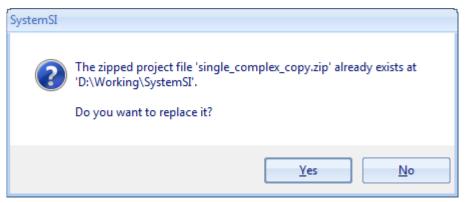


• If the file used by the original project does not exist, the following message opens

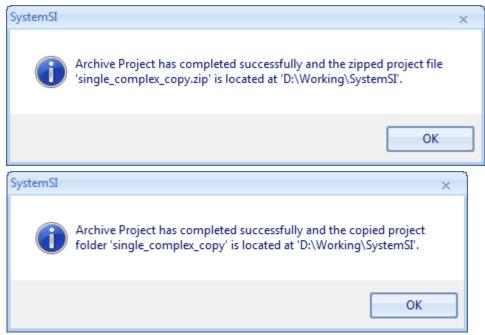
SystemSI	
?	Files highlighted in red are not found. Without them, the archived project will fail to simulate. Do you want to continue?
	<u>Y</u> es <u>N</u> o

• If the designated project folder already has the archived project folder or zipped file, the following messages open

SystemSI	
?	The folder 'single_complex_copy' already exists at 'D:\Working\SystemSI'. Do you want to replace it?
	<u>Y</u> es <u>N</u> o



• If the archive project process completes successfully, the following messages open



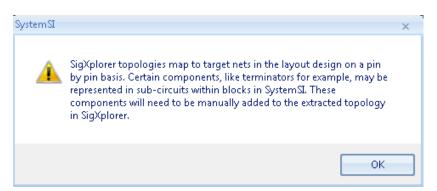
### 4.8 Integration with Allegro Signal Explorer

SystemSI provides a way to push a topology for a particular signal into SigXplorer for constraint capture. SigXplorer topologies are then used to drive electrical constraints, including the routing schedule, into Allegro layout. The topology database becomes an Electrical Constraint Set (ECSet) within the Allegro database that can be applied to one or many nets.

To export the results of Serial Link Analysis as electrical constraints, perform the following steps.

1. From the File menu, choose Export **Constraint Topology**.

Following message box is displayed.



2. Select **OK**, to close the message box.

SystemSI starts the export process. Once the process is complete, splash screen for Allegro SigXplorer appears, followed by the Product selection dialog box.

3. Select the appropriate license and click **OK**.

16.6 Allegro Sigrity SI Product Choices	<b>—</b>
Select a Product:	
Allegro PCB SI GXL (legacy) Allegro PCB SI XL	ОК
Allegro Sigrity SI Allegro PCB Multi-Gigabit Option Allegro PCB SI L (legacy)	Cancel
Allegro PCB SI Performance (legacy) Allegro PCB SI Serial Link (legacy)	Help
Available Product Options	
PCB SI Performance	
PCB SI Multi-Gigabit	
🗖 Use as default 👘 Reset license cach	ne

The exported topology is displayed in Allegro SigXplorer.

	For projects with cross-talk topologies, before the start of the export process, the Export Constraint Topology dialog box is displayed.
NOTE!	Export Constraint Topology       ×         Xtalk topology requires an AC sweep to determine the primary channel.       Do you want to continue?         # of Frequency Points:       16       Victim Rx:       RX_PRIMARY         Yes       No
	Before extracting the topology information, System SI runs AC Sweep analysis, which involves S parameter extraction of interconnects, to identify the main channel. To continue with the extraction process, select <b>Yes</b> .

# 5 Compliance Kits

This chapter lists the templates and corresponding compliance checks supported by SystemSI – Serial Link Analysis (SLA), for high-speed SerDes designs. Using compliance kits automates the compliance testing process, thus speeding up the design process.

The topics covered in this chapter are:

- Compliance Workflows
- SFP+ Compliance
- HDMI Compliance
- PCIe Compliance
- 10GBASE-KR Compliance
- USB 3.0 Compliance
- MIPI Compliance

### 5.1 Compliance Workflows

To view the compliance kits supported by SystemSI SLA, perform the following steps.

- 1. Launch SystemSI.
- 2. From the Compliance menu, choose Serial Link Analysis.

All supported compliance kits are listed in the sub-menu.

Compliance	Window	Help
Parallel Bu	s Analysis 🕨	
Serial Link	Analysis 🕨	SFP +
alvsis	;	HDMI
aiysis		PCIe 3
nce		10GBASE-KR
ince	ŧ	USB 3.0

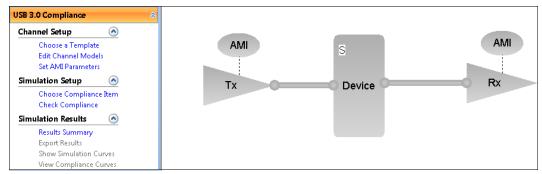
3. To display a compliance workflow, select the compliance from the submenu.

The workflow for the selected compliance is displayed.



### 5.1.1 Templates and Models

All compliance kits supported by SystemSI include predefined ready-to-run templates. While these templates have models associated to each block, you can replace default models with custom models. You can associate custom channel models for interconnects and IBIS-AMI models for transmitters and receivers.



Selecting a template and associating appropriate models to the template blocks are common tasks to be performed for all compliance kits. Depending on the compliance selected, you may need to perform an extra step of setting AMI Parameters.

### 5.1.2 Compliance Checks

After you have specified the models associated with each block, you can run the compliance checks.

1. To view the list of compliance checks supported for a template, from the workflow, select **Choose Compliance Item**.

Compliance options available for the template are listed.

Workflow: SystemSI	x	USB3 (SuperS	peed USB) Compliance Item ×				
Serial Link Analysis	×	~-					
SFP+ Compliance	×	Choo	se compliance item				
HDMI Compliance	×		1	1			
PCIe 3 Compliance	×	No.	Parameter	Symbol			
10GBASE-KR Compliance	×	Eye Heigh	nt At test point 1 (TP1) (Table 6-12)				
USB 3.0 Compliance	念	1	Eye Height	Use CP0			
Channel Setup	$\odot$	Tx Differe	ential Swing (Table 6-12)				
Choose a Template Edit Channel Models		2	Tx Differential Swing	V <sub>TX-DIFF-PP</sub>			
Set AMI Parameters		Total Jitter (Table 6-12)					
Simulation Setup Choose Compliance It		3	Total Jitter	Tj			
Check Compliance	cini	Jitter Tole	erance Test for Rx (Table 6-19)				
Simulation Results Results Summary	<u></u>	4	Stressed/Swept Jitter	Rx in BERT mode			
Export Results Show Simulation Curv View Compliance Curv Customize Workflow		Measure Tool		OK Reference Cable	Cancel		

2. You can either select all options by selecting the check box in the first row, or can select individual options by selecting the corresponding check boxes in the last column.

Choo	ose compliance ite	m	
No.	Parameter	Symbol	
Eye Heig	ht At test point 1 (TP1) (Table 6-12)		
1	Eye Height	Use CP0	V
Tx Differ	ential Swing (Table 6-12)		
2	Tx Differential Swing	V <sub>TX-DIFF-PP</sub>	
Total Jitt	er (Table 6-12)		
3	Total Jitter	Tj	V
Jitter Tol	erance Test for Rx (Table 6-19)		
4	Stressed/Swept Jitter	Rx in BERT mode	V

- 3. Click OK to save your selections.
- 4. To start the compliance checks, from the workflow, select **Check Compliance**.

The Run Simulation dialog box is displayed.

Run Simula	tion	×
?	The compliance check simulation is about to start. Do you want to continue?	
	Re-characterize channel	
	Redo frequency domain simulation	
	OK Cancel	

### • Re-characterize channel

Select this option if the design has been modified since last simulation, and channel characterization needs to be done again. Selecting this option ensures that all modification are accounted for in the channel simulation results.

### • Redo frequency domain simulation

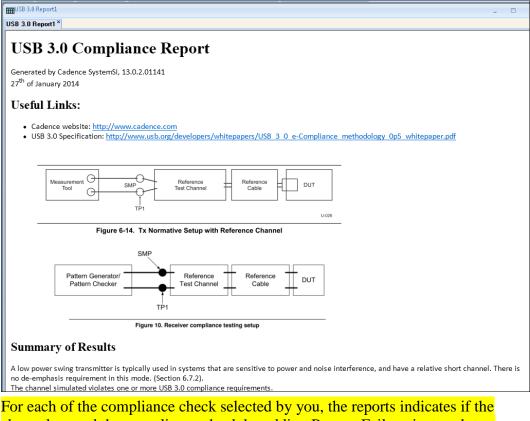
Select this option if you want to run the frequency domain simulation (This is same as signal connectivity check in other Sigrity tools). This option has no impact on the channel simulation results, and can be ignored.

5. Click OK to start the simulation.

Once the simulation is complete, the simulation report is generated and displayed.

### 5.1.3 Viewing Results

As the simulation is completed, the simulation report is displayed, as shown in the following figure.



For each of the compliance check selected by you, the reports indicates if the channel passed the compliance check by adding Pass or Fail against each test in the report.

Parameters	Symbol	Min	Max	Units	Simulation Results	Pass/Fail
Eye Height		0.1	1.2	V	<u>0.144</u>	Pass
Tx Differential Swing						
Parameters	Symbol	Min	Max	Units	Simulation Results	Pass/Fail
Tx Differential Swing	V <sub>TX-DIFF-PP</sub>	0.8	1.2	v	1.122	Pass
Total Jitter						
Parameters	Symbol	Min	Max	Units	Simulation Results	Pass/Fail
Total Jitter	Tj		0.66	UI	0.670	Fail
Jitter Tolerance Test 1	for Rx		1		1	
Parameters	Symbol	Min	Max	Units	Simulation Results	Pass/Fail

Post simulations, all Simulation Results options in the workflow are enabled.



### **Export Results**

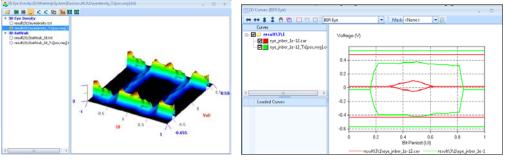
Select this option to save the simulation results in the specified location. Both, simulation results and simulation curves, are saved.

### **Show Simulation Curves**

Select this option to view the simulation results. On selecting this option, the **Show Simulation Result** dialog box displays.

Show Simulation Result 🛛 🗙 🗙					
Option					
Show in the same windows					
Select Result					
V Eye Density					
Eye Contour					
BER Eye					
Bathtub					
Voise Bathtub					
Ramp Response					
Ramp Response (single-ended)					
Impulse Response					
Modified Impulse Response					
Rx Waveform					
Report					
Select All Deselect All					
OK Cancel					

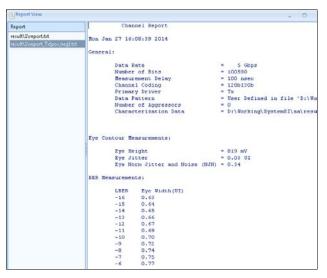
> Accept the default selection and click OK.



Simulation curves are displayed in 3 windows; 2D curves window, 3D Curves, and report.







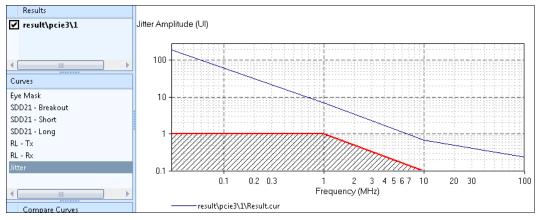
Report View

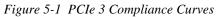
### **View Compliance Curves**

Select this option to view the compliance curves.

Each compliance kit has unique set of compliance curves.

> To view a compliance curve, select the appropriate curve in the **Curves** pane.





NOTE!

For more information on compliance curve, see the sections on Compliance Curve Window and Curve Pane Context Menu.

## 5.2 SFP+ Compliance

SystemSI-Serial Link Analysis provides SFP+ compliance checks for cable assemblies and for printed circuit boards. *Figure 5-2* and *Figure 5-3*, lists the SFP+ compliance checks available for cable assemblies and boards, respectively..

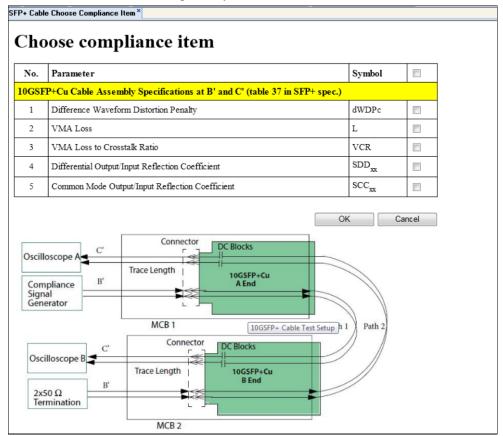


Figure 5-2: SFP+ Compliance Checks for Cables

No.	Parameter	Symbol		
Host 7	Fransmitter Output Specification at B (table 11 in SFP+ spec.)			
1	Termination Mismatch at 1MHz	DZ <sub>M</sub>		
	Differential Output S-parameter	SDD22		
2	Common Mode Output S-parameter	SCC22	<b>V</b>	
Host	Fransmitter Output Jitter and Eye Mask Specification at B (table 1	2 in SFP+ spec.)		
3	Signal Rise/Fall Time (20% to 80%)	Tr, Tf		
4	Total Jitter	TJ		
-	Data Dependent Jitter	DDJ		
5	Data Dependent Pulse Width Shrinkage	DDPWS		
б	Uncorrelated Jitter UI			
7	Transmitter Qsq	Qsq		
8	Eye Mask	Eye Mask		
Host I	Receiver Input Specification at C (table 13 in SFP+ spec.)			
9	Differential Input S-parameter	SDD11		
9	Reflected Differential to Common Mode Conversion	SCD11		
SFI C	hannel Transfer Recommendation (table 25 in SFP+ spec.)			
10	Channel Transfer from chip pad to point B or C	SDD21		
Additi	onal SFI Channel Recommendations			
11	SFI Channel Return Loss	SDD11, SDD22		
12	SFI Channel Ripple	Ripple		
SFP+	Host Output Specification at B for Cu (table 33 in SFP+ spec.)			
13	Voltage Modulation Amplitude(p-p)	VMA		
14	Transmitter Qsq	Qsq		
15	Host Output TWDPc	TWDPc		

Figure 5-3: SFP+ Compliance Tests For PCBs

If the *Waveform Distortion Penalty (dWDPc or TWDPc)* check is selected, following dialog box is displayed:

Setup for WDP	X
PRBS9 Pattern File: prbs9_950.txt	
Feedforward Equalizer Taps (EqNf):	14
Feedback Equalizer Taps (EqNb):	5
	OK Cancel

To accept the default values, click OK.

# 5.3 HDMI Compliance

SystemSI provide compliance kit for verifying channel performance against HDMI (High Definition Multimedia Interface) specifications.

No.	Parameter	Symbol	
Source	DC characteristics at TP1		
1	Single-ended output swing voltage	V <sub>swing</sub>	
2	Single-ended high level output voltage	V <sub>H</sub>	<b>V</b>
3	Single-ended low level output voltage	VL	<b>V</b>
Source	AC characteristics at TP1		
4	Rise time / fall time (20%-80%)		
5	Intra-Pair Skew at Source Connector, max		
6	Inter-Pair Skew at Source Connector, max		
7	Clock duty cycle, min / average / max		>
8	TMDS Differential Clock Jitter, max		<b>V</b>
9	Eye Mask		<b>V</b>
Engine	ering Target		
10	Differential Insertion Loss	SDD21	<b>V</b>
11	Differential Impedance	Z <sub>DIFF</sub>	
12	Single-ended Impedance	Z <sub>SE</sub>	
13	3 Intra-Pair Skew at Source Connector, max		<b>V</b>
14	Inter-Pair Skew at Source Connector, max		<b>v</b>
15	NEXT / FEXT		<b>v</b>

Figure 5-4 HDMI Compliance Checks

# 5.4 PCIe Compliance

The PCIe compliance kit follows the PCI Express Gen 3 standard, including IBIS-AMI models with back-channel support.

No.	Parameter	Symbol	
Channe	Tolerancing Eye Mask Values (table 4-27 in PCI Expres	ss Base spec.)	
1	Eye Height	V <sub>RX-CH-EH</sub>	
2	Eye Width at Zero Crossing	T <sub>RX-CH-EW</sub>	
3	Peak EH Offset from UI Center	T <sub>RX-DS-OFFSET</sub>	
4	Range for DFE d <sub>1</sub> Coefficient	V <sub>RX-DFE-COEFF</sub>	
5	Eye Mask		
Differen	ntial Insertion Loss (figure 4-66 in PCI Express Base spec	2.)	
6	Insertion Loss	SDD21	
Differer	ntial Return Loss (figure 4-56 in PCI Express Base spec.)		
7	Tx Return Loss	RL - Tx	
8	Rx Return Loss	RL - Rx	
Stresse	d/Swept Jitter Test (figure 4-74 in PCI Express Base spec	:.)	
9	Stressed/Swept Jitter		

Figure	5-5PCIe 3	Compliance	Checks
rigure	5-51 Cle 5	compnance	Checks

#### Jitter Tolerance

Most of the compliance kits included with SystemSI SLA have Jitter Tolerance included as one of the compliance checks. To run the Jitter Tolerance analysis in the **PICe 3** workspace:

Select the **Stressed/Swept Jitter** compliance check.

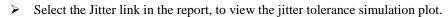
With this check selected, the compliance report includes the jitter tolerance results.

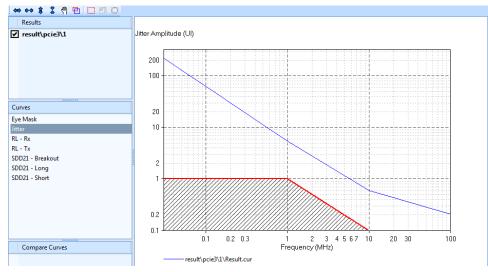
NOTE!
-------

The generated PCIe3 compliance report includes the jitter tolerance results.

Stressed/Swept Jitter Test	
----------------------------	--

Item	Value	Simulation Results	Pass/Fail
Stressed/Swept Jitter		Jitter	Pass





### 5.5 10GBASE-KR Compliance

No.	Parameter	Symbol	<b>V</b>		
Interfe	Interference Tolerance (Annex 69A of the 10GBASE-KR spec.)				
1	Interference Tolerance		<b>V</b>		
Interc	onnect Characteristics (Annex 69B of the 10GBASE-KR spec.)				
2	Skew between P and N Side of Thru Diff Pair		<b>V</b>		
3	Insertion Loss	IL	<b>V</b>		
4	Fitted Attenuation	А	<b>V</b>		
5	Insertion Loss Deviation	ILD	<b>V</b>		
б	Tx Return Loss at TP4	RL - TP4	<b>V</b>		
7	Rx Return Loss at TP1	RL - TP1	<b>V</b>		
8	Ratio of Insertion Loss to Crosstalk between TP1 and TP4	ICR	<b>V</b>		

Figure 5-6 10GBASE-KR Compliance Checks

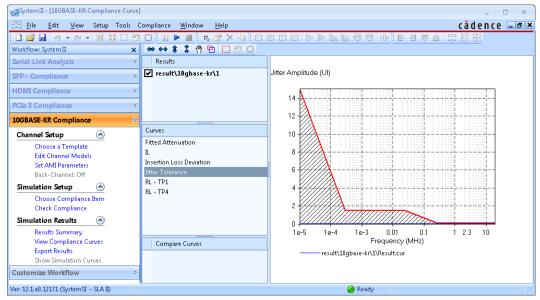
#### Jitter Tolerance

To run the Jitter Tolerance analysis for **10GBASE**-KR compliance, select **The Interference Tolerance** compliance check.

With this check selected, the compliance report includes the jitter tolerance results.

Interference Tolerance			
Item	Value	Simulation Results	Pass/Fail
mTC	1	0.977	Fail
bTC	Equation (69-A-7) of the 10GBASE-KR spec.	0.104	
Jitter Tolerance		Jitter Tolerance	Pass

Selecting the Jitter\_Tolerance link displays the compliance curve.



# 5.6 USB 3.0 Compliance

SystemSI – Serial Link Analysis provides support for compliance testing process for USB 3.0 serial links. For this, following templates are available:

- Device Short Channel (Host compliance channels)
- Device Long Channel (Host and Cable compliance channels)
- Host Short Channel (Device compliance channels)
- Host Long Channel (Device and Cable compliance channels).

No.	Parameter	Symbol			
Eye Heigl	Eye Height At test point 1 (TP1) (Table 6-12)				
1	Eye Height	Use CP0			
Tx Differe	ential Swing (Table 6-12)				
2	Tx Differential Swing	V <sub>TX-DIFF-PP</sub>			
Total Jitte	r (Table 6-12)				
3	Total Jitter	Тј			
Jitter Tole	Jitter Tolerance Test for Rx (Table 6-19)				
4	Stressed/Swept Jitter	Rx in BERT mode			

Figure 5-7USB 3.0 Compliance Checks

#### Jitter Tolerance

To run the Jitter Tolerance analysis for USB 3.0:

- Select the **Stressed/Swept Jitter** compliance check.
- Ensure that **usb3\_jtolmask** is selected as mask file in the Jitter Tolerance dialog box.
- $\succ$  Run the compliance checks.

The generated compliance report includes the jitter tolerance results.

**NOTE!** You can specify the jitter amplitude and the frequency values obtained from the jitter tolerance curve, in the Jitter & Noise Tab for Tx, and validate the channel simulation results.

# 5.7 MIPI Compliance

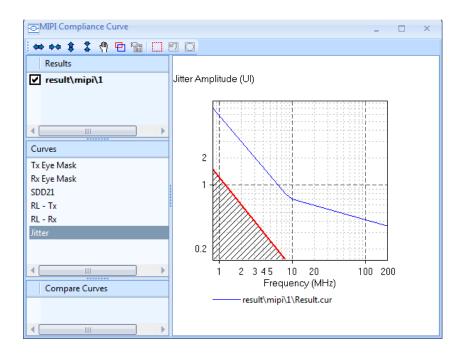
The MIPI standard is targeted for short range (less than one meter) mobile applications that require low pin count and low power consumption. SystemSI MIPI compliance kit supports high speed MIPI serial links.

No.	Parameter	Symbol	<b>V</b>
TX Eye	Mask Values (Table 16 in M_Phy spec ver	rsion 3.0)	
1	Eye Height	V <sub>DIF_AC_HS_G3_TX</sub> (>80 mv)	>
2	Eye Width at Zero Crossing	T <sub>EYE_HS_G3_TX</sub> (>0.55 UI)	<b>V</b>
3	Eye Mask		<b>V</b>
Channe	l Tolerancing Eye Mask Values(Table 21 in	n M_Phy spec version 3.0)	
4	Eye Height	V_DIF_AC_HS_G3_RX <sup>(&gt;80 mv)</sup>	>
5	Eye Width at Zero Crossing	T <sub>EYE_HS_G3_RX</sub> (>0.58 UI)	<b>V</b>
6	Eye Mask		>
Differe	ntial Return Loss(Figure 31 and 43 in M_Pl	ıy spec version 3.0)	
7	Tx Return Loss	RL - Tx	<b>V</b>
8	Rx Return Loss	RL - Rx	>
Stresse	d/Swept Jitter Test(Figure 44 in M_Phy spe	ec version 3.0)	
9	Stressed/Swept Jitter		>

Figure 5-8 MIPI Compliance Checks

### Jitter Tolerance

To run the Jitter Tolerance analysis for **MIPI** compliance, select **Stressed/Swept Jitter** compliance check. With this check selected, the compliance report includes the jitter tolerance results.



# A Appendix: Batch Mode Support

# A.1 Batch Mode Command

1. List of the batch mode commands for the regular SystemSI - Serial Link Analysis:

```
<ExeFileFullPath\SystemSI.exe> -b -sim <workspace file>
\\for the default SSI simulation
<ExeFileFullPath\SystemSI.exe> -b -sim:sweep <workspace file>
\\for the sweep
<ExeFileFullPath\SystemSI.exe> -b -sim:freq <workspace file>
```

 $\backslash\!\backslash$  for the frequency response

<ExeFileFullPath\SystemSI.exe> -b -sim:sparam <workspace file>

 $\parallel$  for the S-parameter extraction

Example for Windows OS:

```
Path\SystemSI.exe -b -sim
"C:\Working\SystemSI\SFPP\sfpp\sfpp.ssix"
Path\SystemSI.exe -b -sim:sweep
"C:\Working\SystemSI\SFPP\sfpp\sfpp.ssix"
Path\SystemSI.exe -b -sim:freq
"C:\Working\SystemSI\SFPP\sfpp\sfpp.ssix"
Path\SystemSI.exe -b -sim:sparam
"C:\Working\SystemSI\SFPP\sfpp\sfpp.ssix"
```

2. List of the batch mode commands for the SystemSI – Serial Link Analysis compliance kits:

```
<ExeFileFullPath\SystemSI.exe> -b -compliance:sfpp <workspace file>
```

 $\backslash\!\!\backslash$  for the sfpp compliance. It will generate the compliance report and export the html file to the default folder

```
<ExeFileFullPath\SystemSI.exe> -b -compliance:hdmi <workspace file>
```

 $\backslash\!\!\backslash$  for the hdmi compliance. It will generate the compliance report and export the html file to the default folder

<ExeFileFullPath\SystemSI.exe> -b -compliance:pcie3 <workspace file>

 $\backslash\!\!\backslash$  for the pcie3 compliance. It will generate the compliance report and export the html file to the default folder

```
<ExeFileFullPath\SystemSI.exe> -b -compliance:10gbase-kr <workspace file>
```

 $\backslash\!\!\backslash$  for the 10 gbase-kr compliance. It will generate the compliance report and export the html file to the default folder

```
<ExeFileFullPath\SystemSI.exe> -b -compliance:usb3 <workspace file>
```

 $\backslash\!\!\backslash$  for the USB 3.0 compliance. It will generate the compliance report and export the html file to the default folder

Example for Windows OS:

```
Path\SystemSI.exe -b -compliance:usb3
"C:\Working\SystemSI\SFPP\sfpp\sfpp.ssix"
```

# A.2 Run a .bat file

1. Put down the batch mode commands into a .bat file.

```
Example for Windows OS:
(sfpp.ssix: a SLA workspace)
set systemsi="the full path of the systemsi.exe"
%systemsi% -b -sim "C:\Working\SystemSI\SFPP\sfpp\sfpp.ssix"
%systemsi% -b -sim:sweep
"C:\Working\SystemSI\SFPP\sfpp\sfpp.ssix"
%systemsi% -b -sim:sparam
"C:\Working\SystemSI\SFPP\sfpp\sfpp.ssix"
%systemsi% -b -compliance:pcie3
"C:\Working\SystemSI\SFPP\sfpp\sfpp.ssix"
```

To run the tests simultaneously, make the following modification in the tested .bat file: set systemsi=start "the full path of the systemsi.exe"

**NOTE!** Replace the red font part with the full path of SystemSI.exe file.

2. Double-click the . bat file to run it.

# A.3 Result Folder for Batch Mode

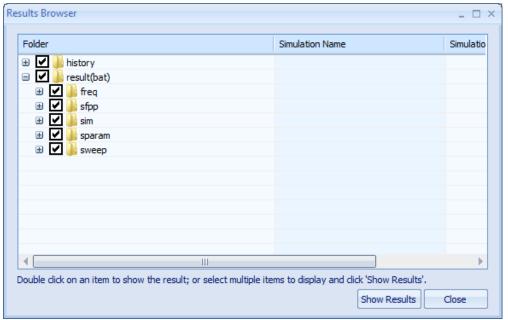
A new sub-folder **result(bat)** will be added for all batch mode results. Each batch mode simulation has its own sub-folder under the **result(bat)** folder.

Example

For a SLA workspace, each common result is located in its responding sub-folder. The compliance result is in a folder named with the compliance type.

► Local Disk (C:) ► Working ► SystemSI	▶ SFPP ▶ sfpp ▶ result	(bat) 🕨
Include in library 🔻 Share with 👻	Burn New folder	
Name	Date modified	Туре
퉬 freq	1/28/2013 1:50 PM	File folder
퉬 sfpp	1/28/2013 1:50 PM	File folder
📗 sim	1/28/2013 1:50 PM	File folder
퉬 sparam	1/28/2013 1:50 PM	File folder
퉬 sweep	1/28/2013 1:50 PM	File folder

The result(bat) folder is available from Results Browser:



In Sweep Manager, all the sweep results including those in the result(bat) folder are listed under the Results > History tab.

teration	Folder	Eye Height (mV)	Eye Jitter (UI)	Eye NJN	TX_THRU>amiffe>fwd	BitSamplingRate
1	history\3\1\1	181	0.53	0.93	1	
2	history\3\1\2	205	0.53	0.93	2	
1	history\3\2\1	0	1	0	1	1
2	history\3\2\2	0	1	0	2	1
3	history\3\2\3	0	1	1	1	2
4	history\3\2\4	0	1	1	2	2
1	result(bat)\sweep\1\1	181	0.53	0.93	1	
2	result(bat)\sweep\1\2	205	0.53	0.93	2	
			111			

# A.4 batch\_mode.log File

If a simulation does not run or fails for any reasons, please check the batch\_mode.log file located under the **result(bat)** folder.

Error messages are added to batch\_mode.log for the following failures:

- The sweep parameter is not defined for the sweep analysis
- The ports are not defined for the S-parameter extraction
- The workspace does not have the fixed blocks, nodes, and MCPs pre-defined for the specified compliance kit