cadence

# SystemSI – Serial Link Analysis Tutorial

Product Version 16.6
July 2014

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

# Table of Contents

# 1 Single Channel Analysis

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

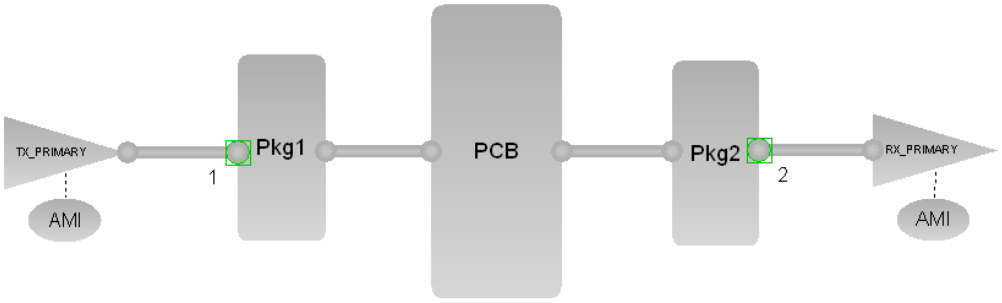
- Assign models to components
- Run the single channel simulations
- Set up the simulation parameters
- Set up the simulation options

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.

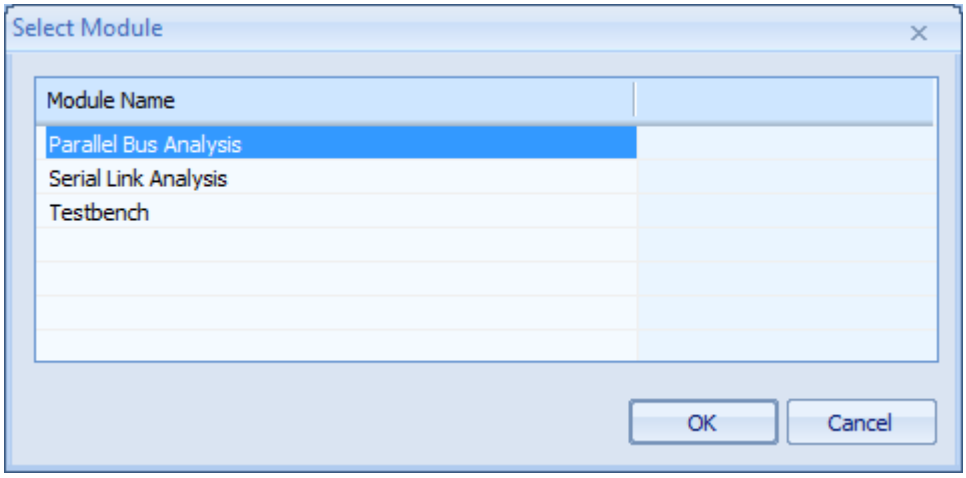
You can modify the template; for example, adding additional blocks which are part of your actual channel.

## 1.2 Starting the Single Channel Template

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

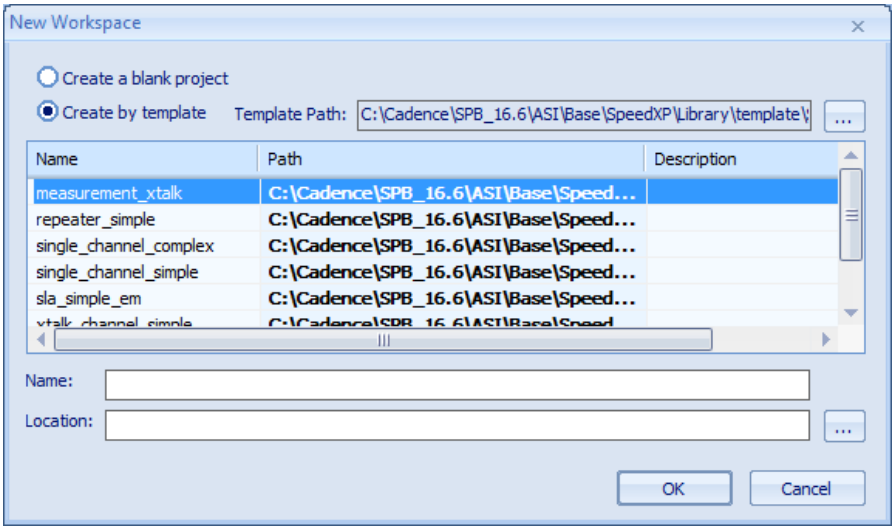
   or, in the **Main** toolbar, click the **New** button .

   The **Select Module** dialog opens.




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.




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.

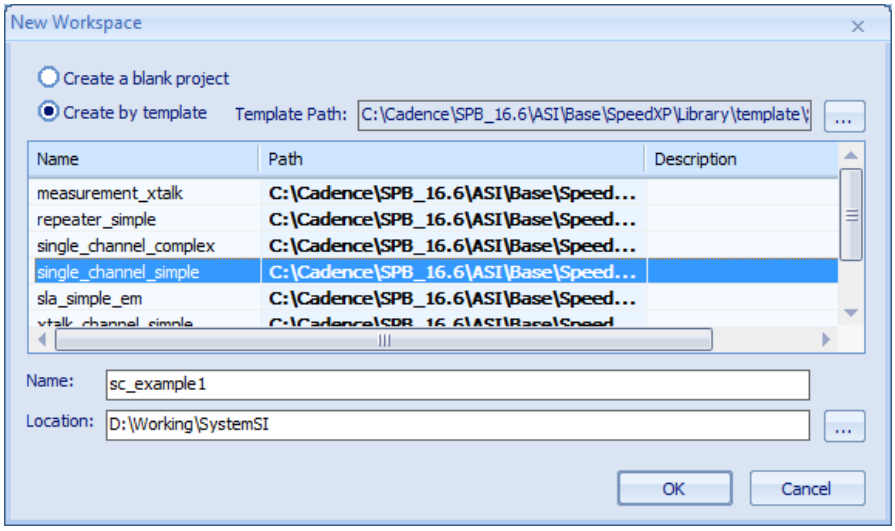
**<INSTALL_DIR>\SpeedXP\Library\template\SystemSI\Serial Link Analysis\single_channel_simple**

**<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**

| **NOTE!** | Do not change the name or location of the template folders. |
|---|---|




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.

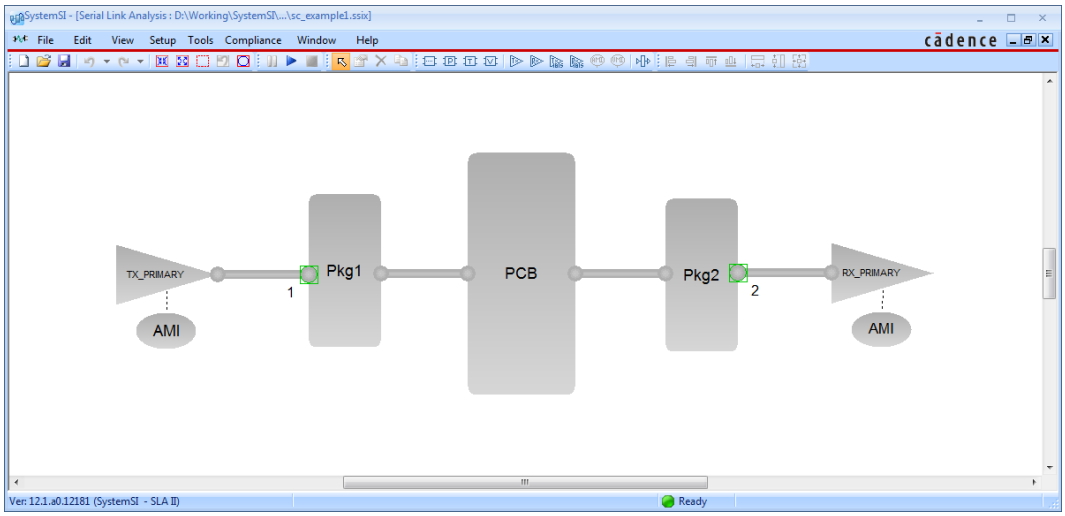
**Pkg1** and **Pkg2**: Two packages.

**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. |
|---|---|

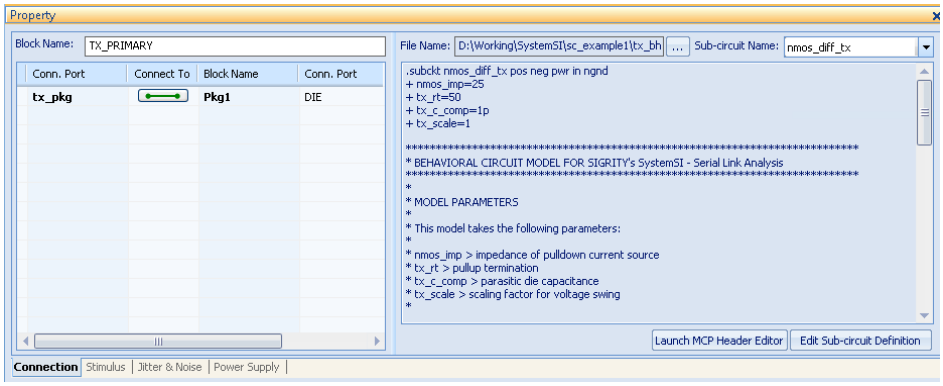
## 1.3 Channel Components

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.




The **Property** pane of the **TX_PRIMARY** block contains four tabs:

- **Connection**
- **Stimulus**
- **Jitter** & **Noise**
- **Power Supply**

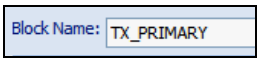
### 1.3.1.1 Connection Tab

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

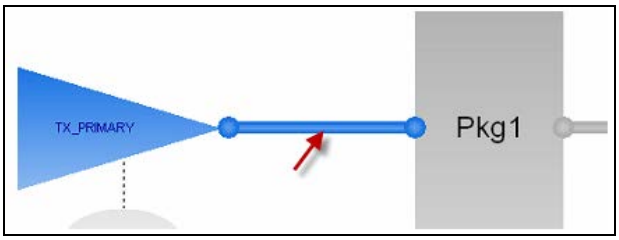
Change the Block Name in the **Block Name** window.




#### 1.3.1.1.2 Connection List

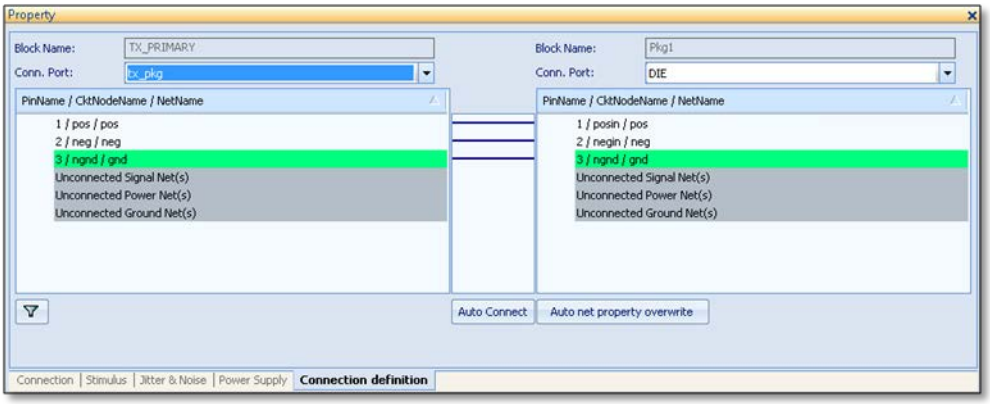
The **Connection List** displays connections between blocks. The **TX_PRIMARTY** block has a connection **(tx_pkg)** which connects to the **Pkg1** block at the **DIE** side.


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.



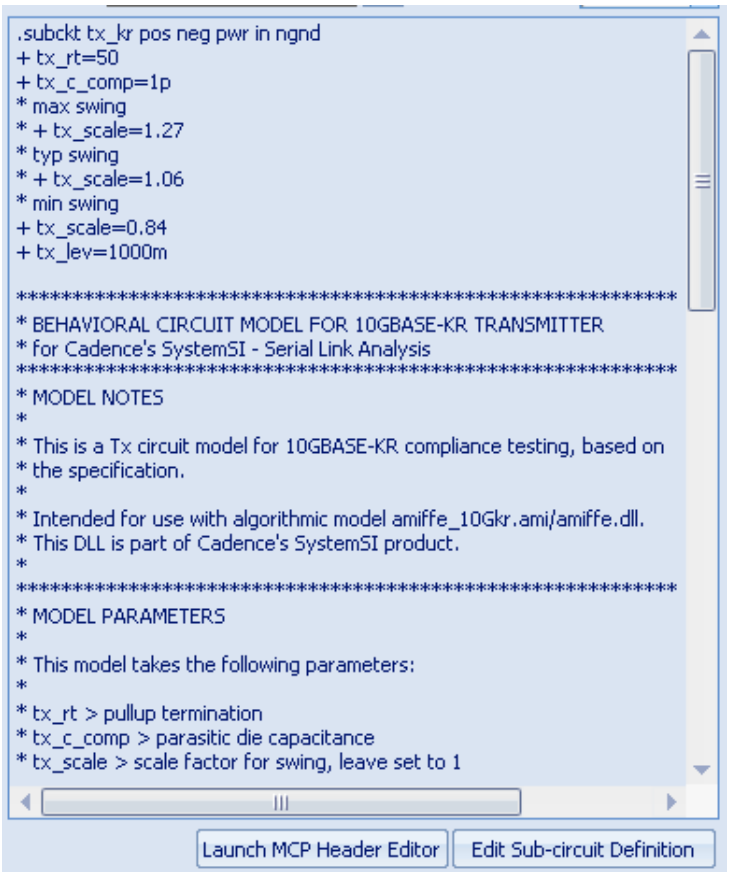

#### 1.3.1.1.3 File Name and Sub-circuit Name

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

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.



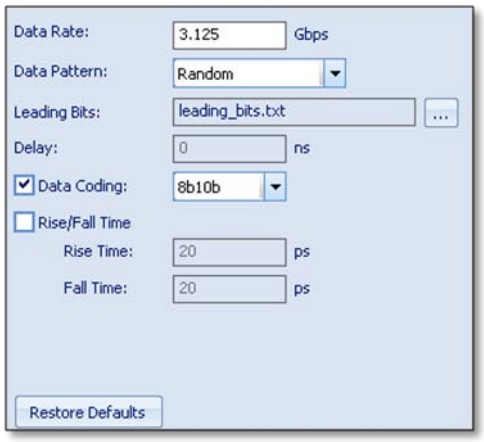

It is recommended that you should not edit the MCP section. Make sure the sub-circuit nodes are called properly so that the connections are correct.

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]. SystemSI needs this section to manage the channel connections. |
|---|---|

### 1.3.1.2 Stimulus Tab




- **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..

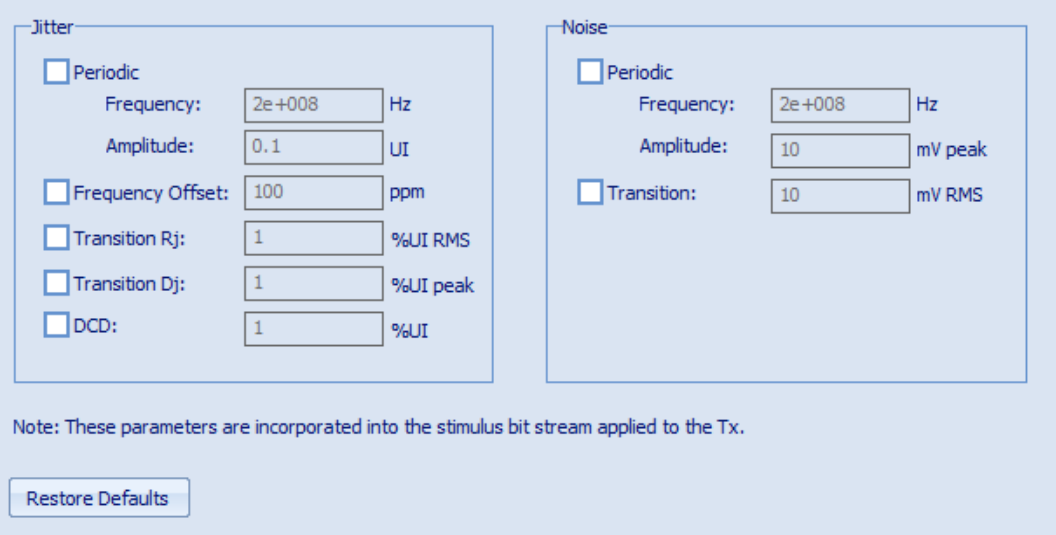
The two periods at the end are required for the pattern to be repeated over and over until the desired number of bits has been reached. If the two periods are not included, just a short bit stream will be run. No post-processed outputs (ex. eye contour) will be generated.

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.




**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.

| **NOTE!** | Jitter elements are disabled by default. |
|---|---|

#### 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** – 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),

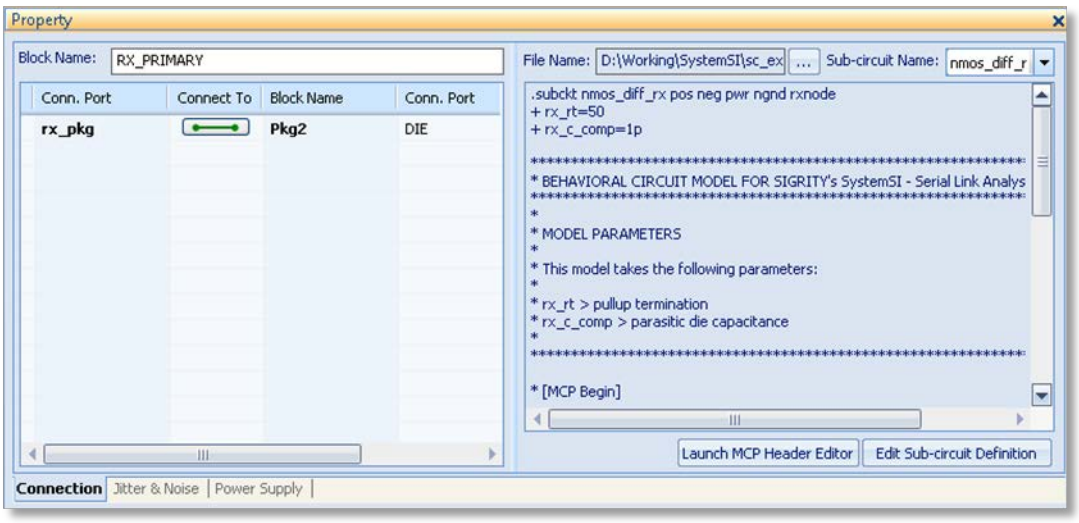
  – If the `.sp` file assigned to the Tx or Rx does not have its own power supply defined in the .sp file, **Ideal Power Supply** should be enabled in the **Property** GUI.

  – 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**.




### 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

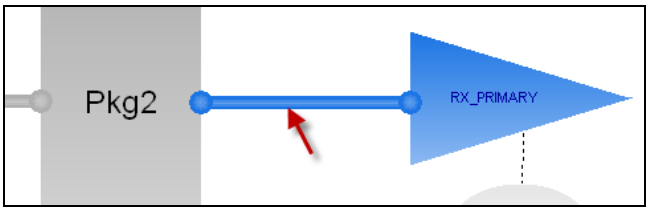
#### 1.3.2.1.1 Block Name

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


#### 1.3.2.1.2 Connection List

This pane displays the connections between the blocks.


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.

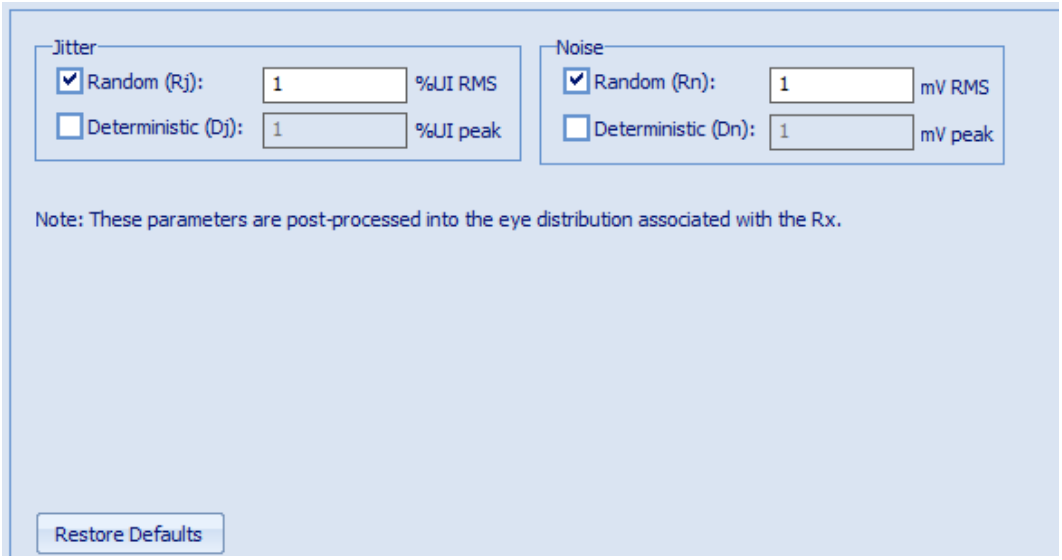

By default, the Single Channel Template has only one sub-circuit model specified for each block. You can choose your SPICE netlist file. Your file might contain several sub-circuit models.

#### 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.




- **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

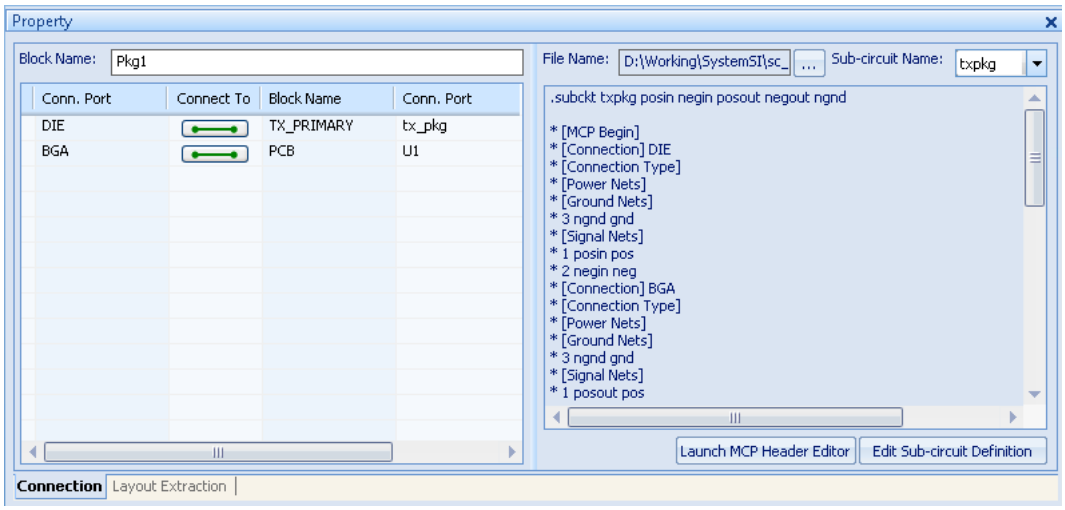

- **Power Supply** – Ideal voltage source that is used at the transmitter. Default value is 1 V.

  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**.




#### 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


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

##### 1.3.3.1.2 Connection List

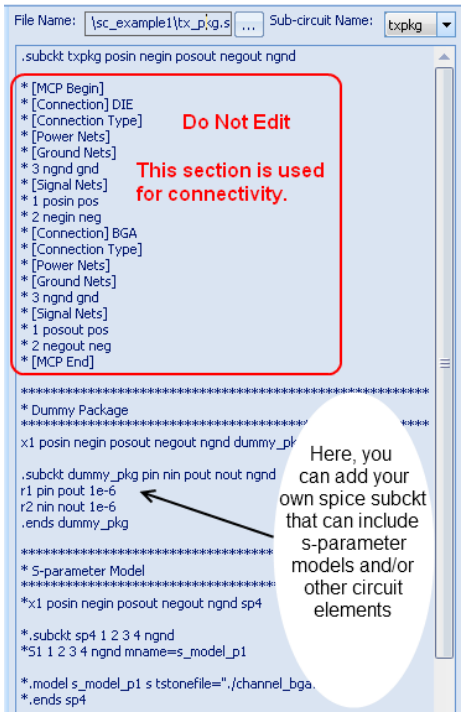
This pane displays the connections between the blocks.

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

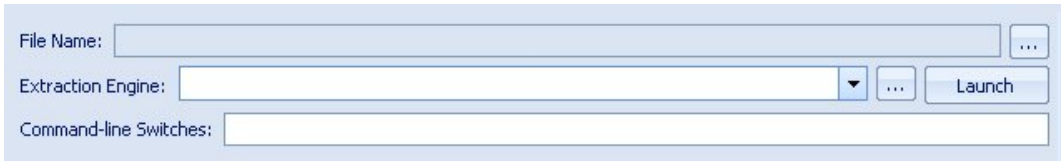
The **Pkg1** block has two connections:

- 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




### 1.3.3.2 Layout Extraction Tab




## 1.3.4 Pkg2

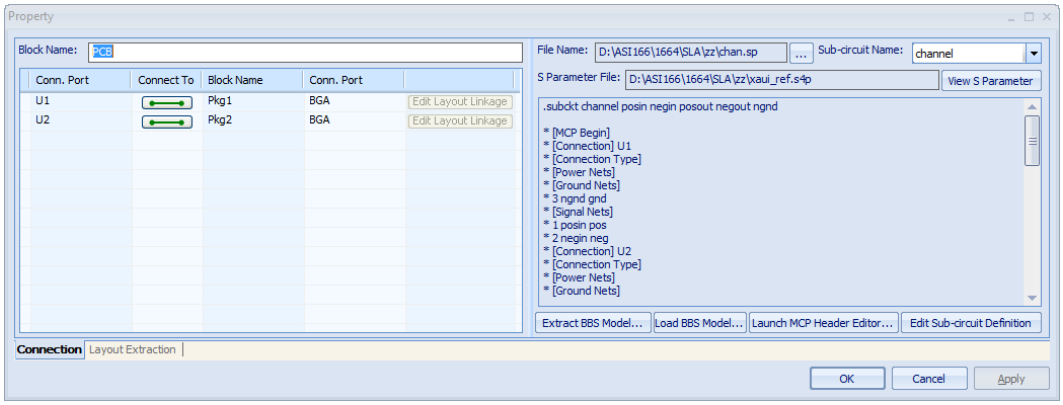
The difference between **Pkg1** and **Pkg2** is in the connectivity and the model name. Both use the same package model.

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:

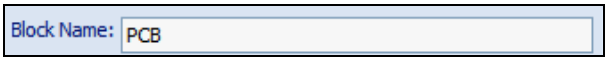



The **Connection** tab contains these parts.

- Block Name
- Connection List
- SPICE Netlist File

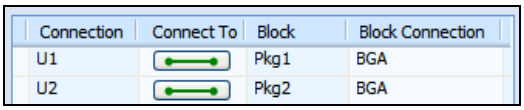
### 1.3.5.1 Block Name

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




### 1.3.5.2 Connection List

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



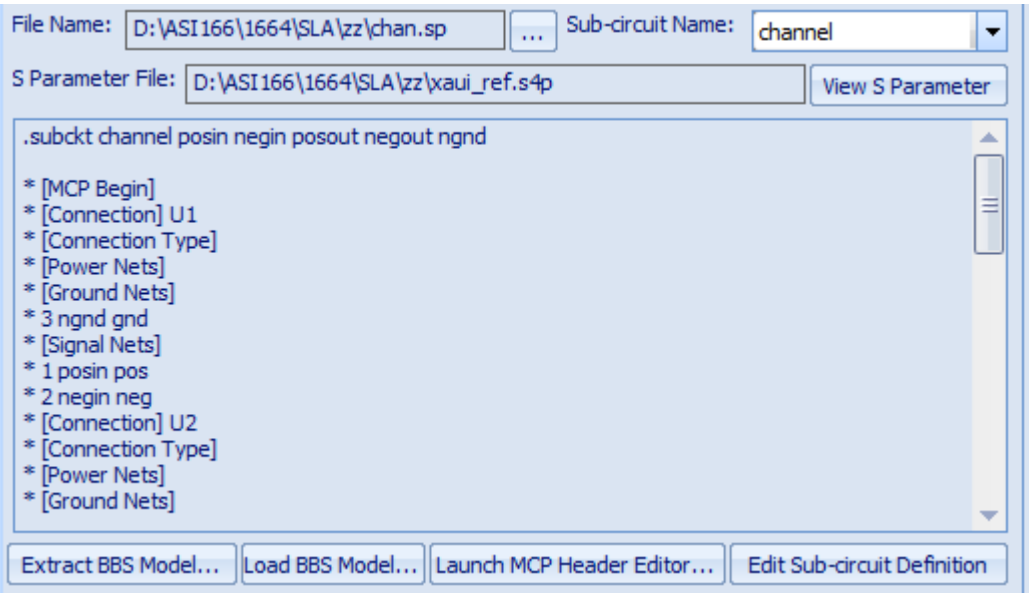

The **PCB** block has two connections:

- 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




- **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**

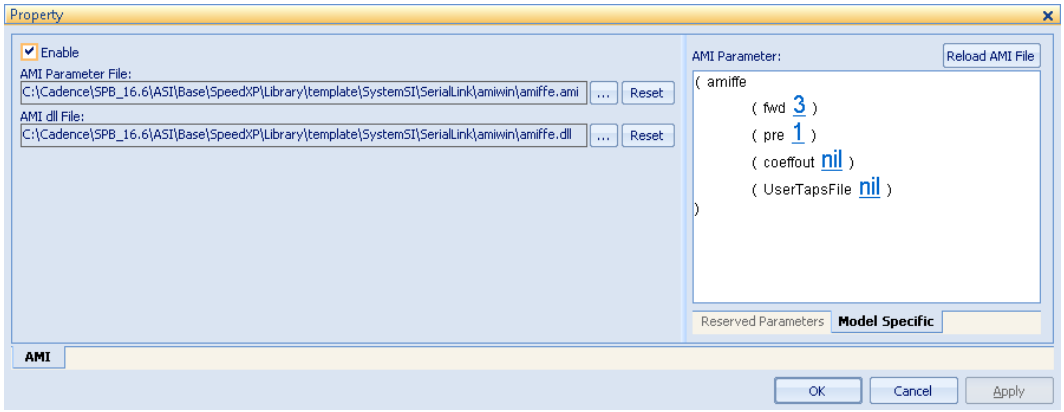
— SystemSI - Serial Link Analysis first characterizes the channel by means of an impulse or step response, and then calls the AMI DLL files to apply equalization to the channel.

**Related Topic**

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

### 1.3.6.1 Transmitter AMI

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



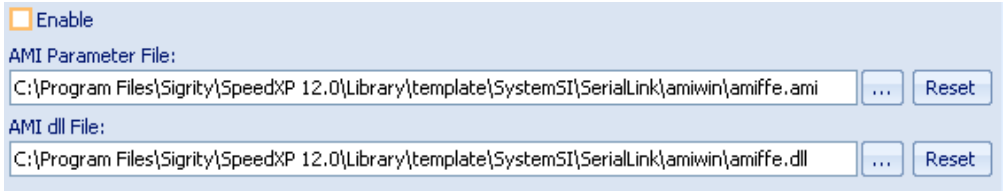

The **AMI** property pane contains two sections:

- AMI parameter and DLL file locations
- AMI parameter list

The AMI parameter list contains two tabs:

- Model Specific
- Reserved Parameters

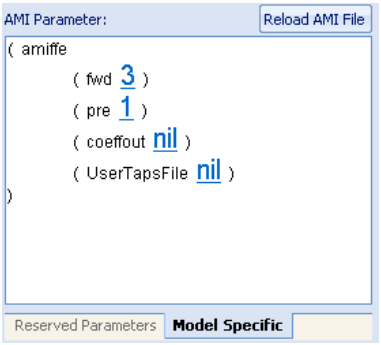
#### 1.3.6.1.1 AMI Parameter and DLL File Locations



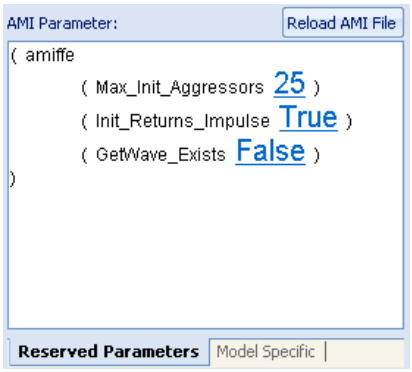

- **AMI parameter file** − Contains all AMI parameters such as **Forward** tab, **Precursor** tab and tap **Coefficient,** as well as usage definitions of those parameters.
- **AMI dll file** − Library file represents the AMI model in an executable format and is called by the channel engine.

| **NOTE!** | When installing the Channel Analysis tool, a template directory is created with all the template files, as well as a new folder called **`amiwin`**. That folder contains all the AMI models. |
|---|---|

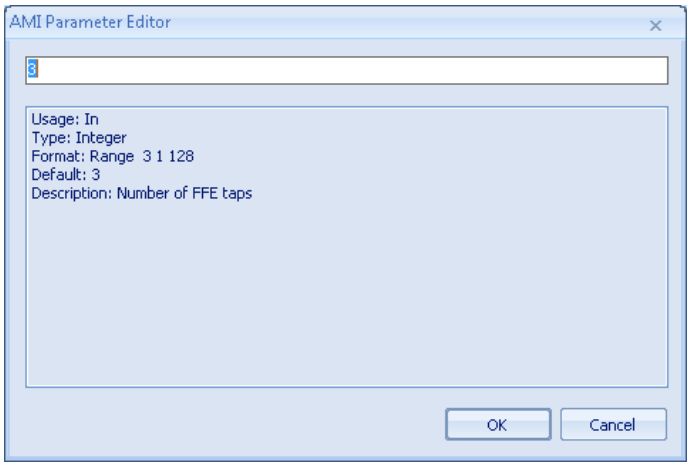
#### 1.3.6.1.2 AMI Parameter List – Model Specific




#### 1.3.6.1.3 AMI Parameter List – Reserved Parameters

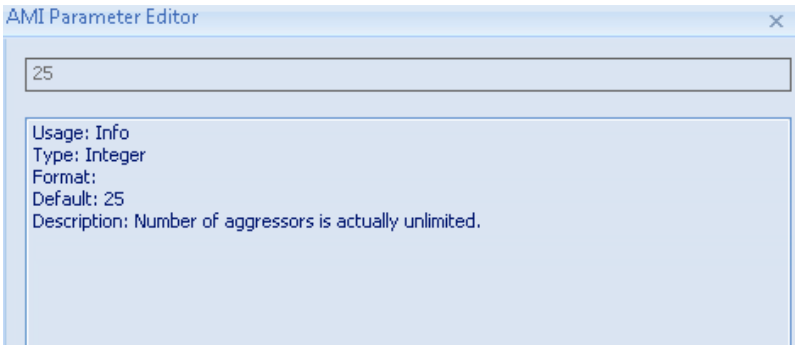



Generally, you can edit the AMI parameters from the **AMI Parameter Editor** as shown below.

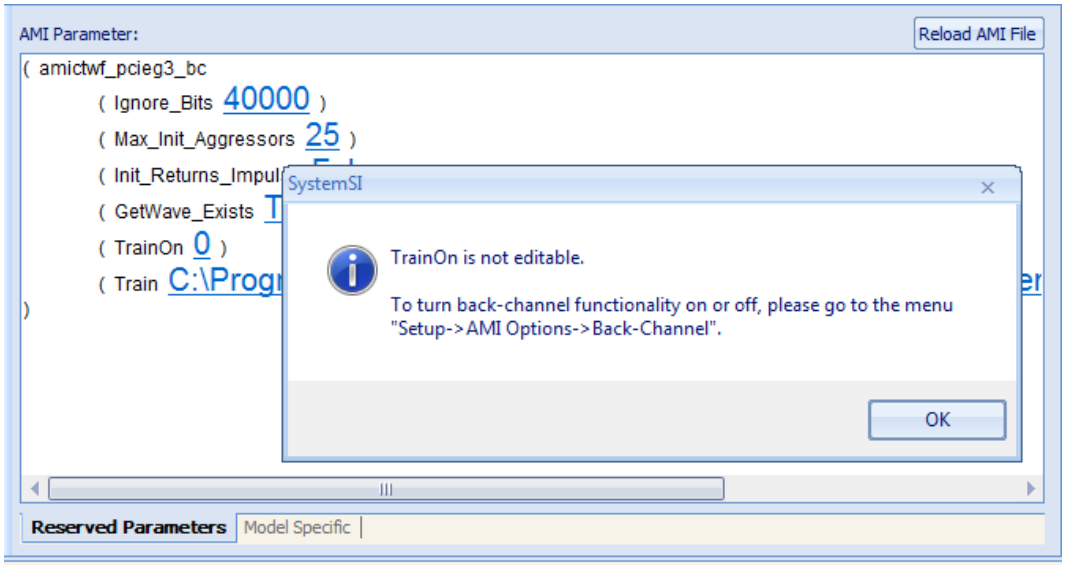



Some reserved parameters are not editable or supported in SystemSI.

- 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.

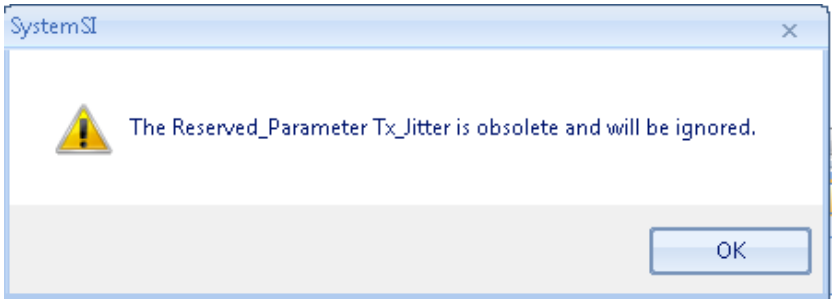



- If you try to edit TrainOn, the following message will be issued.



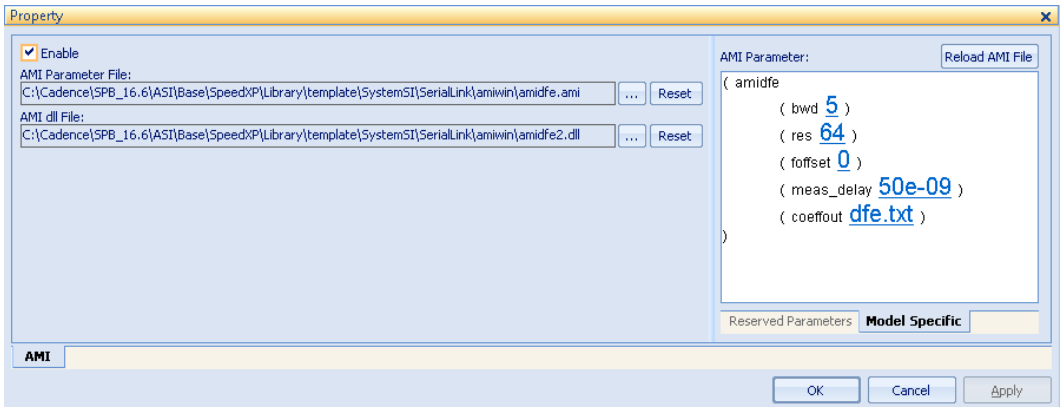

- The reserved parameters Tx_Jitter and Rx_Clock_PDF are not supported by SystemSI, as post BIRD 123 they are obsolete.

  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.



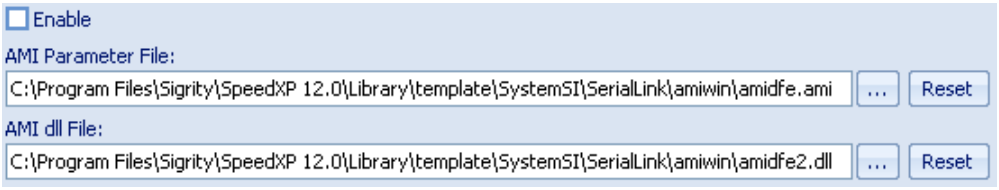

The **AMI** property pane contains two sections:

- AMI parameter and DLL file locations
- AMI parameter list

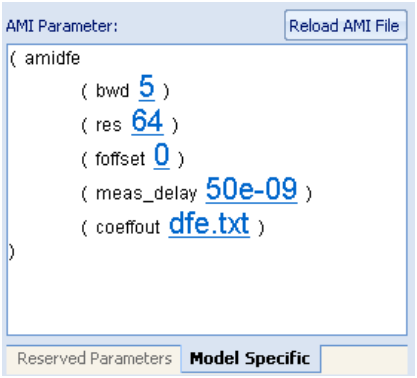
The AMI parameter list contains two tabs:

- Model Specific
- Reserved Parameters

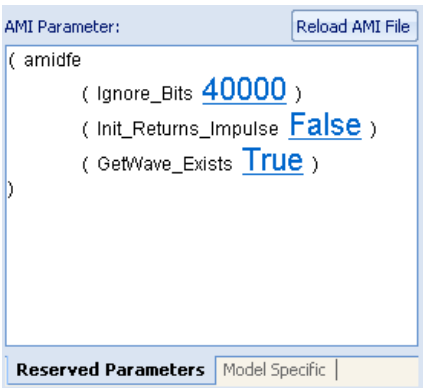
#### 1.3.6.2.1 AMI parameter and DLL File Locations




#### 1.3.6.2.2 AMI Parameter List – Model Specific



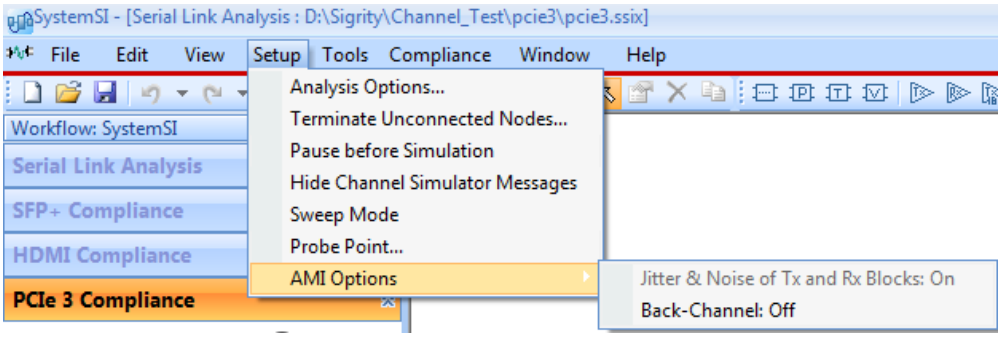

#### 1.3.6.2.3 AMI Parameter List – Reserved Parameters




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

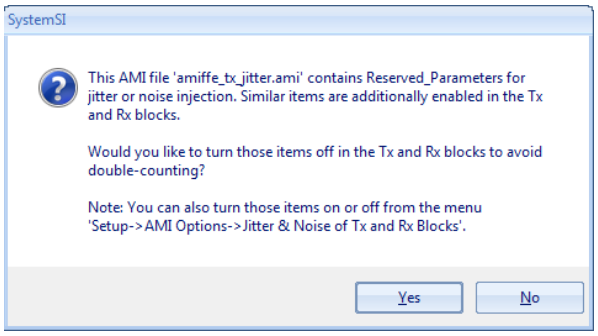
### 1.3.6.3 AMI Options

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




- 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

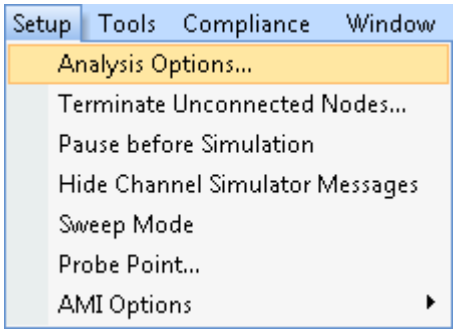
- If any AMI files has the Reserved Parameter TrainOn, AMI Options > Back-Channel can be turned on or off. Otherwise, it is turned off and grayed out.
- When loading an AMI file, if the AMI has any Jitter and Noise parameters (as listed above) as Reserved Parameters, and AMI Options > Jitter & Noise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



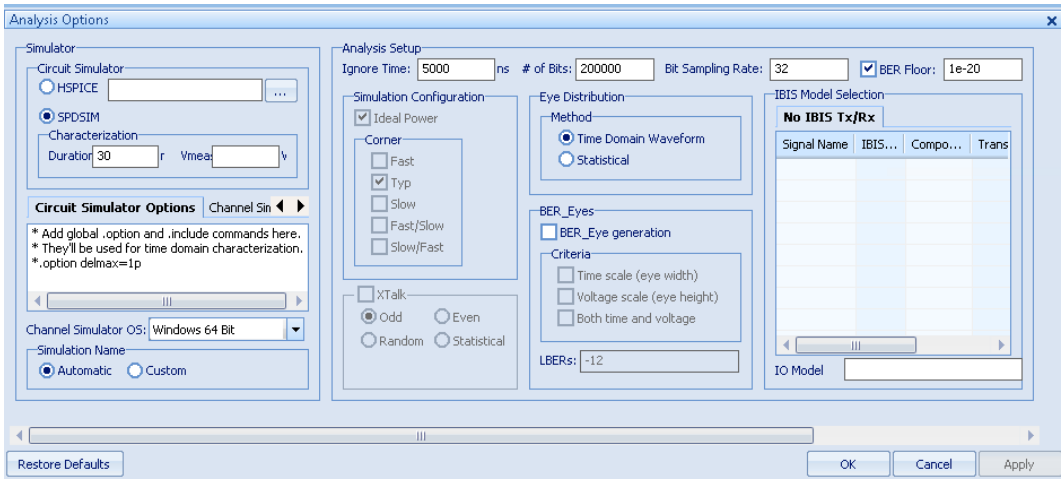

If you click the **Yes** button, AMI Options > Jitter & Noise of Tx and Rx Blocks will be turned off automatically.

## 1.4 Analysis Options

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




The **Analysis Options** pane opens.




The **Analysis Options** interface has two parts:

- **Simulator**

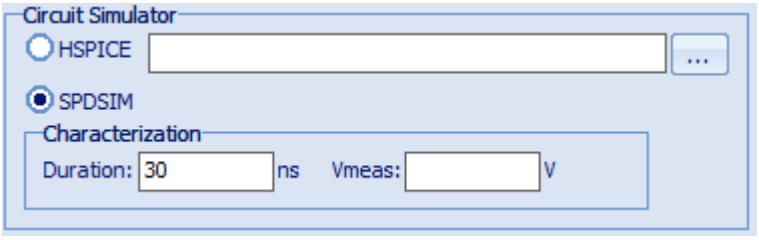
It includes **Circuit Simulator**, **Circuit Simulator Options/Channel Simulator Controls**, **Channel Simulator OS**, and **Simulation Name**.

- **Analysis Setup**

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

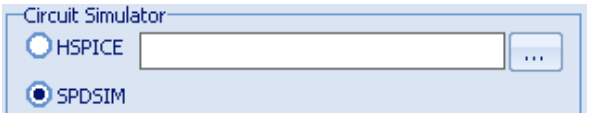
## 1.4.1 Simulator

### 1.4.1.1 Circuit Simulator




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:



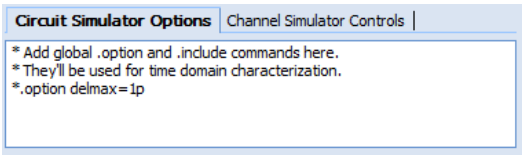

- Specify the path for HSPICE executable file in the **HSPICE** text field.
- SPDSIM is a built-in circuit simulator. It is a SPEED2000 simulator.

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.
- $\mathbf{V_{meas}}$ refers to the voltage threshold at which delay is measured from the Characterization. This information is included in the Channel Report as Delay. If $\mathbf{V_{meas}}$ value is not explicitly called out in an IBIS file associated with the Tx block, $\mathbf{V_{meas}}$ 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




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

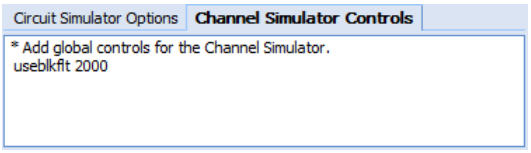
```
.option delmax=2p.
.option delmax
```

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




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:

  - Tx input
  - 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)

- 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.

| **NOTE!** | 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.

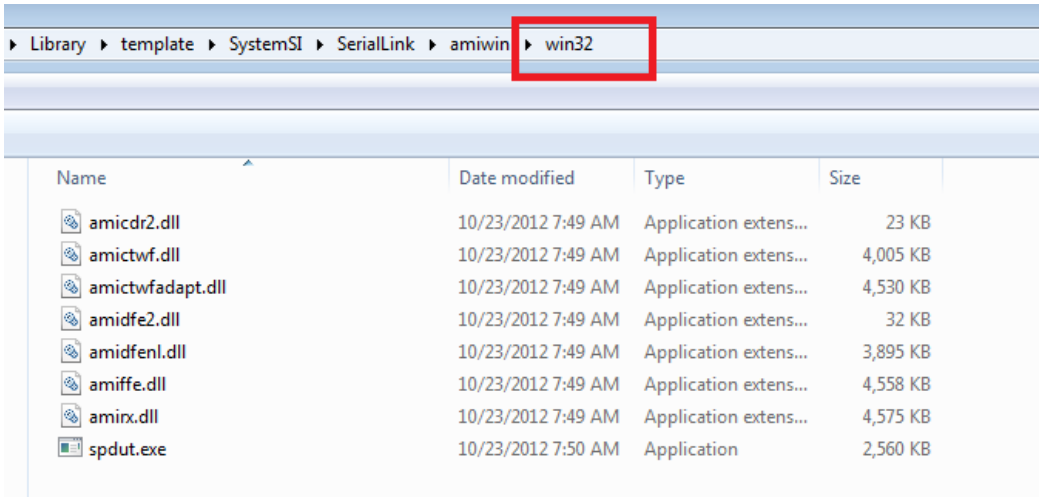
#### 1.4.1.3 Windows 32 Bit AMI DLL Support in a Windows 64 Bit SystemSI Installation

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

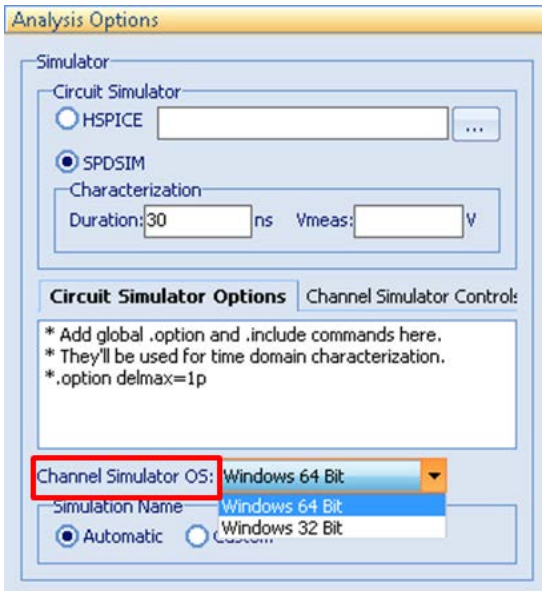
When running channel simulations, it is necessary that the channel simulator (spdut.exe) is compiled for the same Operating System (OS) as the algorithmic models (AMI DLLs) that are being used. As it is fairly common for SerDes suppliers to distribute win32 AMI DLLs, and fairly common for you to install 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




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

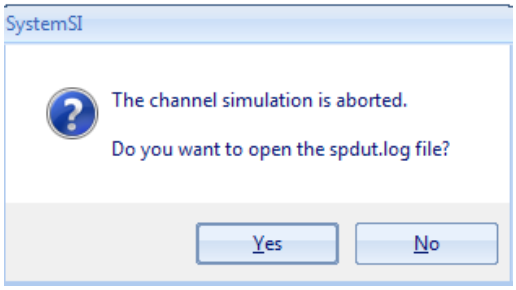



It has two options:

- 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.




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

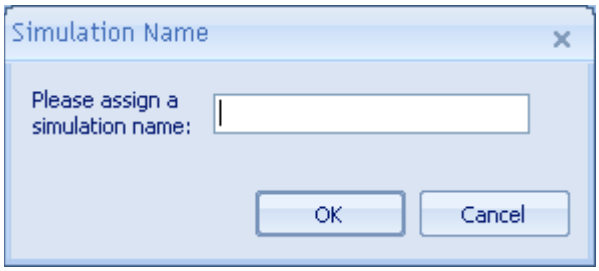
| **NOTE!** | SystemSI installation has both, win64 and win32, versions of the AMI models. Based on the value specified in the **Channel Simulator OS** field, appropriate version of the DLLs and channel simulator are used. |
|---|---|

#### 1.4.1.4 Simulation Name

Define Simulation Result Name.


- 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.




### 1.4.2 Analysis Setup

#### 1.4.2.1 General Channel Simulation Parameters


- **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.

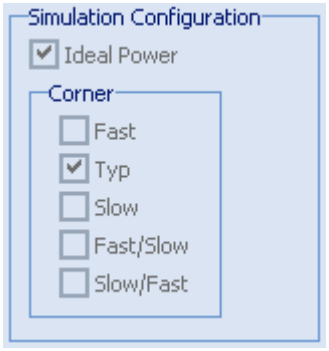
- **Bit Sampling Rate**

  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



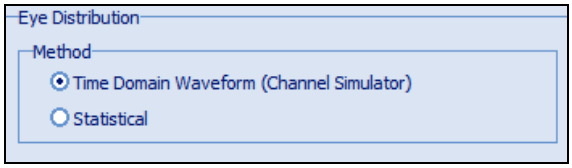

- The **Corner** settings are for the IBIS block and the VRM block. It will be available for selection only when the design has an IBIS block or a VRM block.
- The **Ideal Power** setup is for the VRM block. It will be checked and grayed out if the project does not have any VRM block.

#### 1.4.2.3 Xtalk

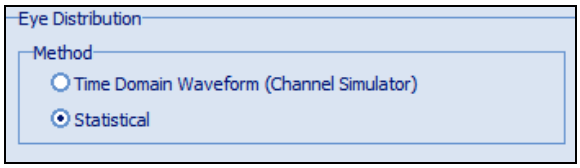

**Xtalk** is not available in Single Channel Analysis. For the details of this part, please refer to ***Section 2.4.2 Xtalk*** in Crosstalk Channel Analysis.

#### 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)**.




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

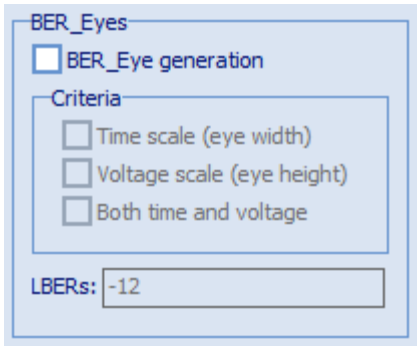



If you select **Statistical**, in the next step **BER_Eye generation** is automatically selected by default.

### 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:

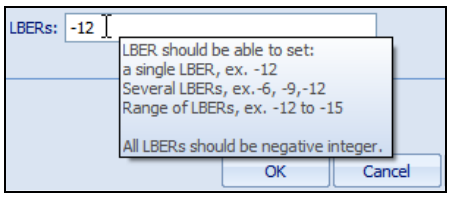
- **Time scale (eye width)**
- **Voltage scale (eye height)**
- **Both time and voltage**




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.

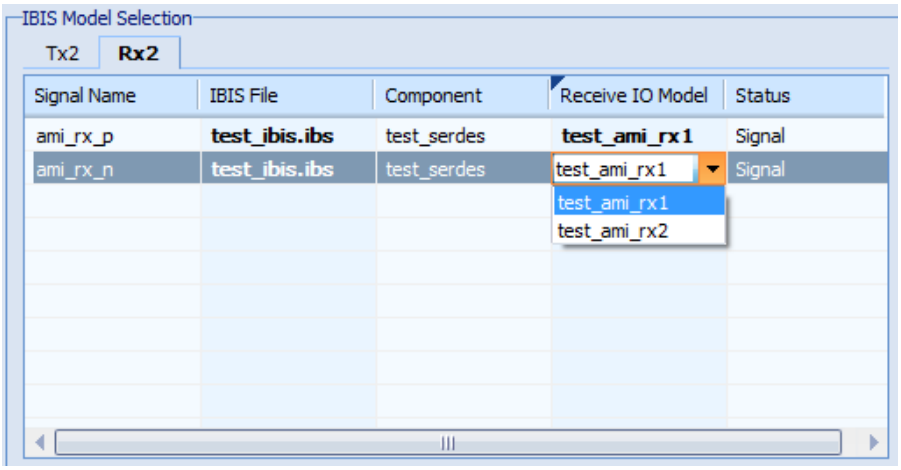
If you place the cursor in the **LBERs** field, an indication box appears. The LBER values are displayed.




| 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.




## 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

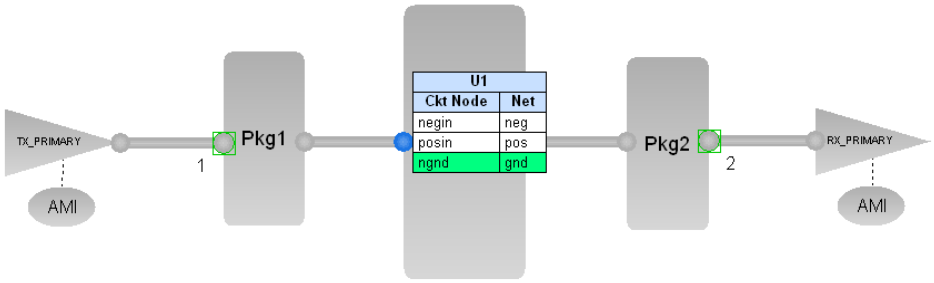
<INSTALL_DIR>\SpeedXP\Library\template\SystemSI\Serial Link Analysis

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.

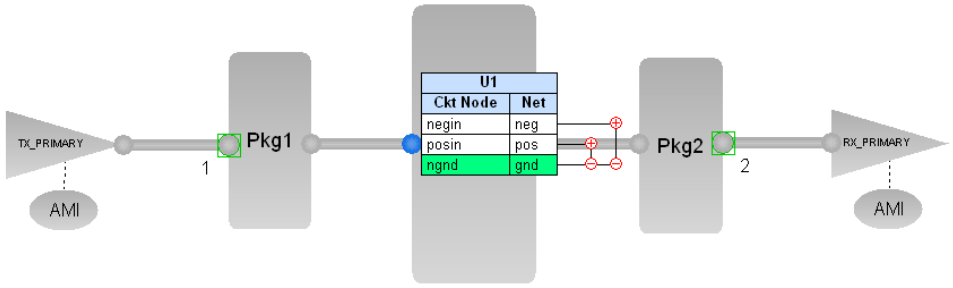
| NOTE! | Do not edit the **`standard_step.sp`** file unless you are an advanced user and want to set up a different stimulus. |
|---|---|

### 1.5.1 Setting up Probe Point

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



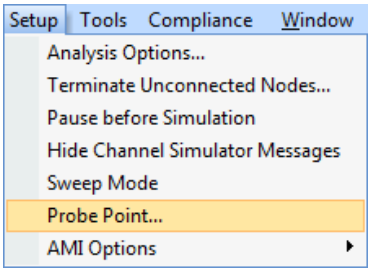

2. To set Probe Point for the simulation, click two Ckt Nodes in the menu.



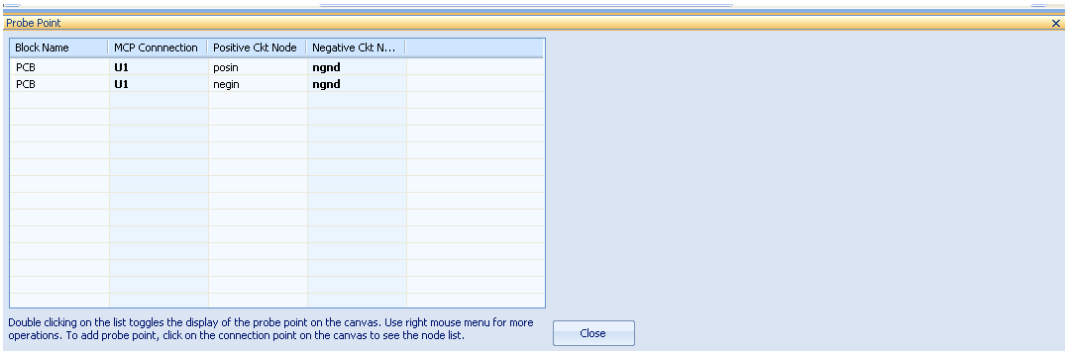

- The two selected nodes are connected.
- The node clicked first is defined as positive, and the second is negative.

3. To view and edit the defined Probe Points, select

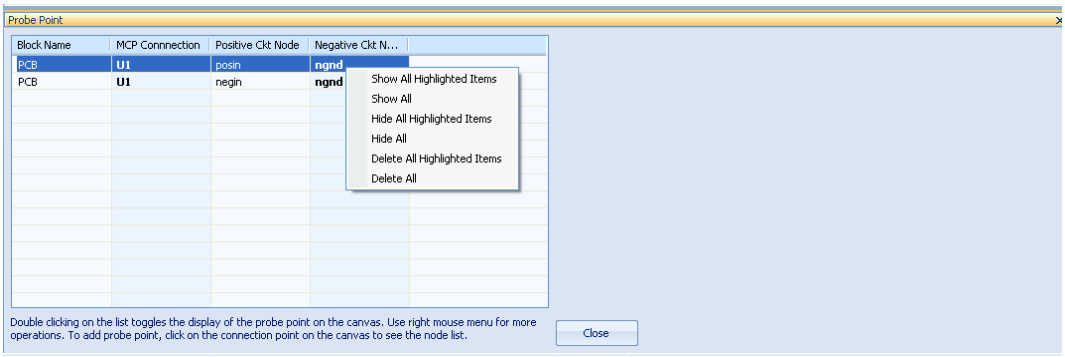
   Setup> Probe Point….



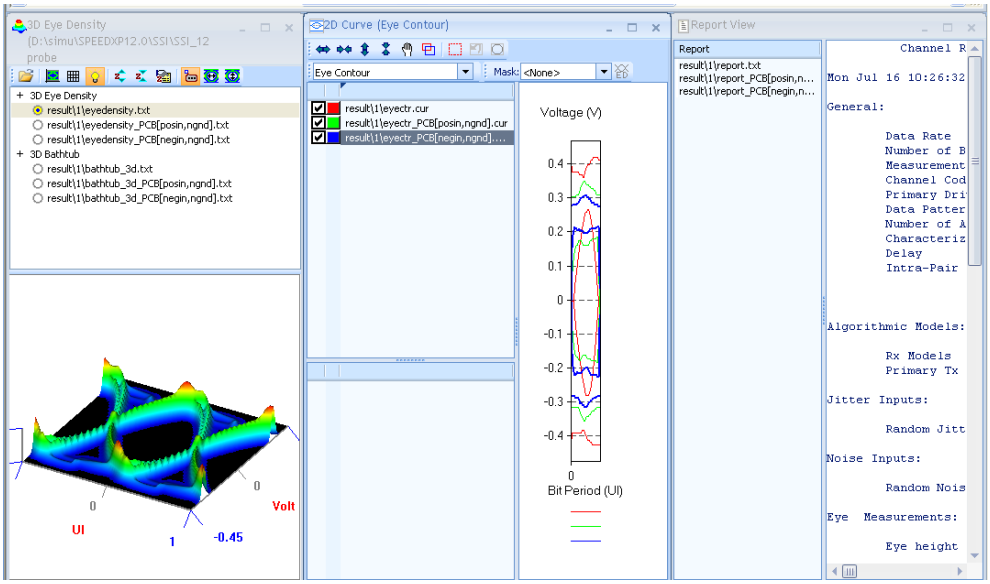

   The **Probe Point** window opens below.




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

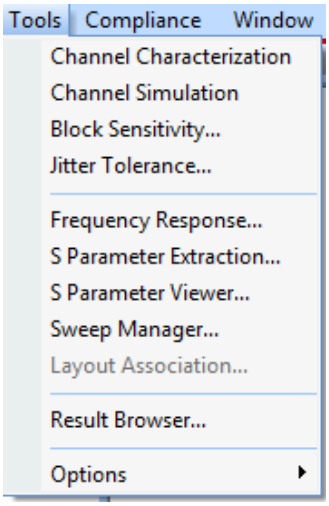



5. After the simulation, the results of all the defined Probe Points can be viewed from the **2D Curve (Eye Contour)** window, **3D Eye Density** window and **Report View** window.

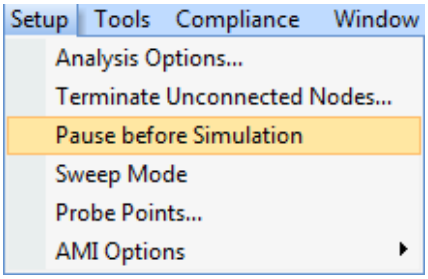



## 1.5.2 Begin Running the Simulation

1. Click the **Start Simulation** button 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**.

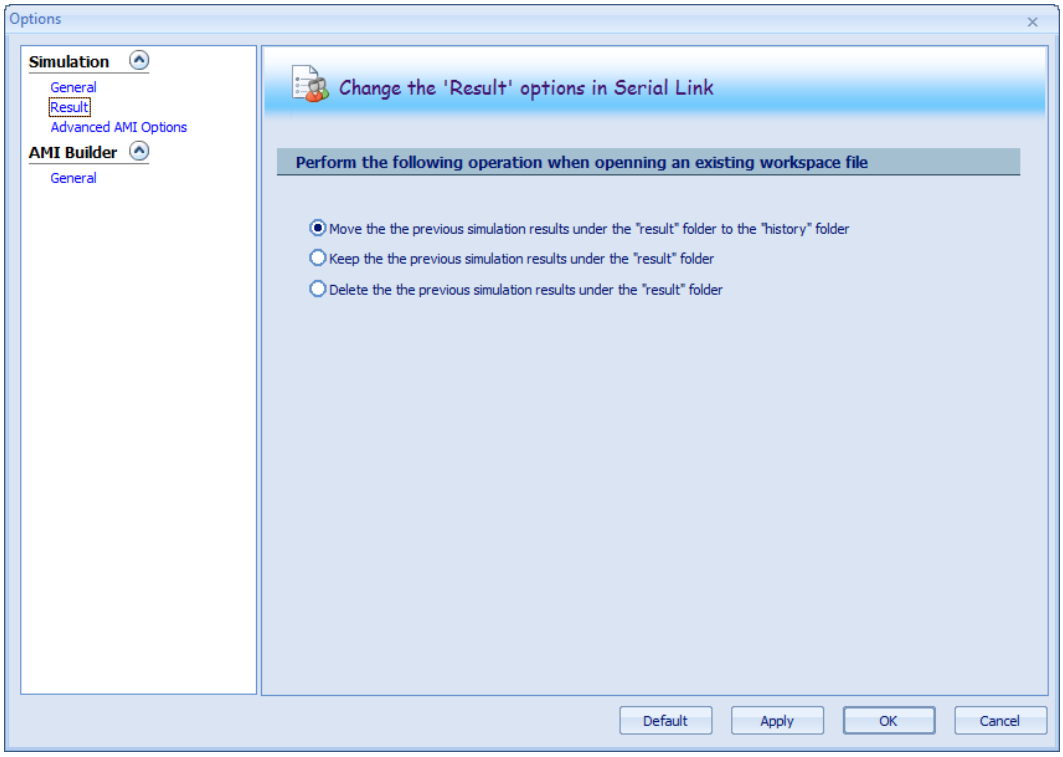


5. If you have enabled **Pause before Simulation** from the **Setup** menu, a message will prompt you whenever a new simulation is about to be run.

### 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**.




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.

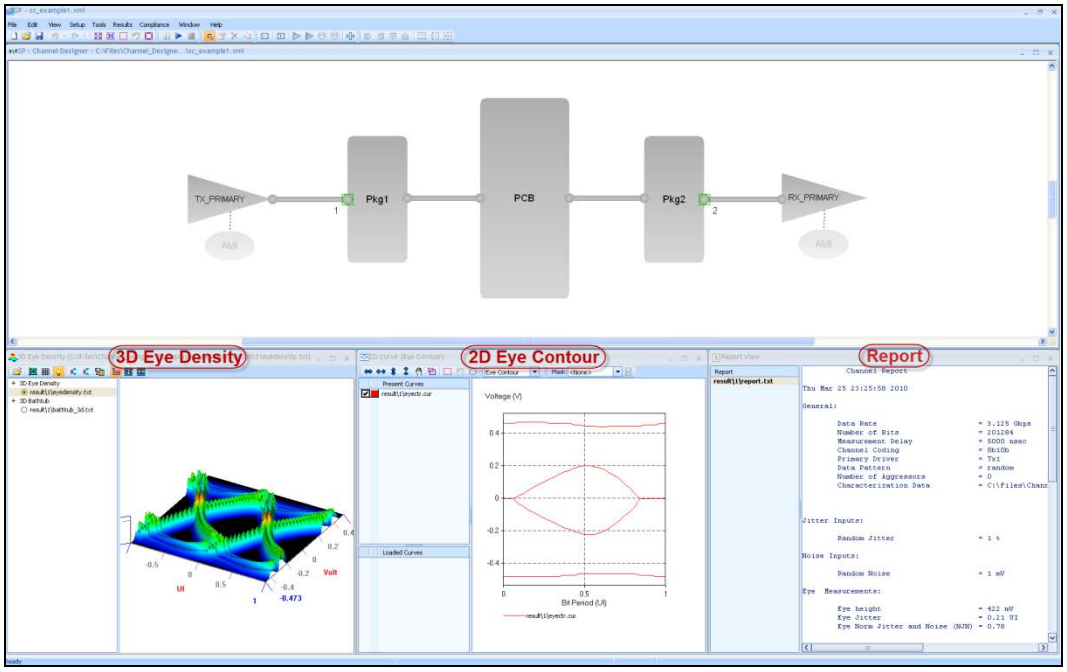
Basically, the .ssix, .html and .txt windows should be put together at the top, and other curve windows should be put together at the bottom.

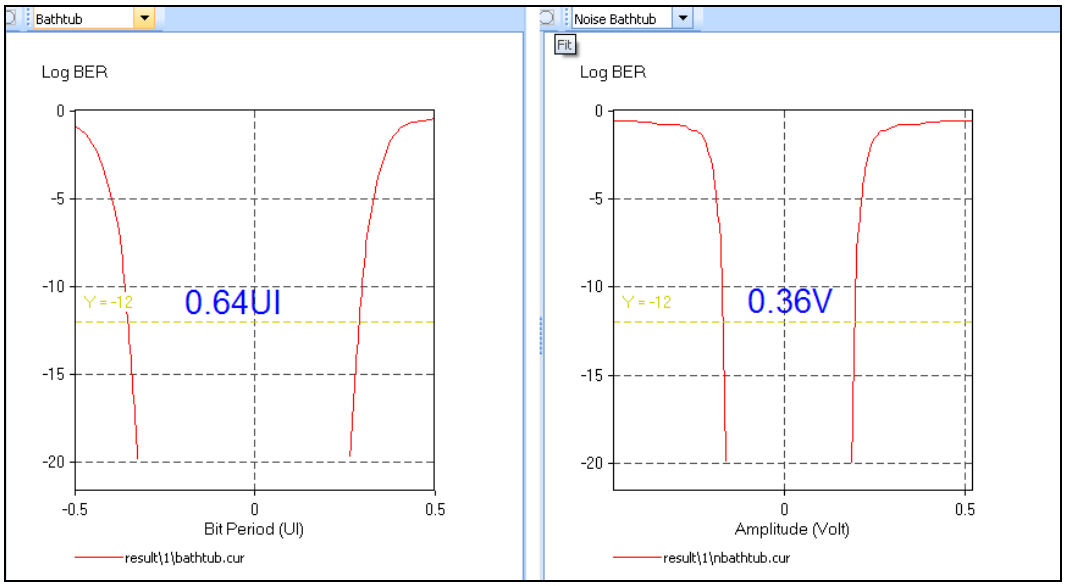
The waveforms in the **2D Curve** window are the most useful ones. The eye contour and bathtubs are very good measures of the quality of any channel.

- **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.

The following figure shows the bathtub curves from the previous simulation.




## 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  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.

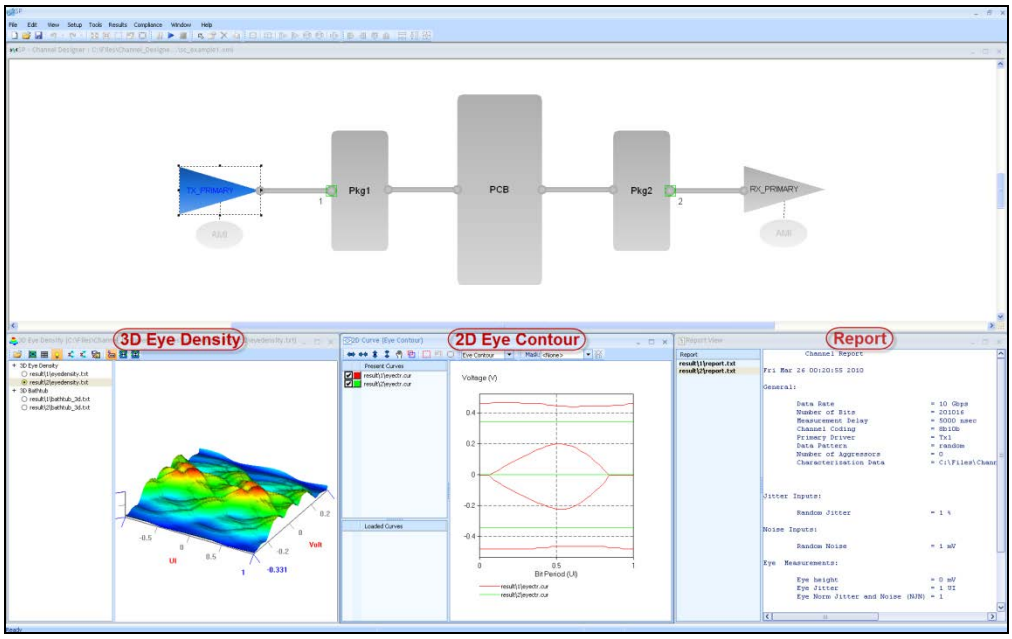
Changes to the channel could mean:

- 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.

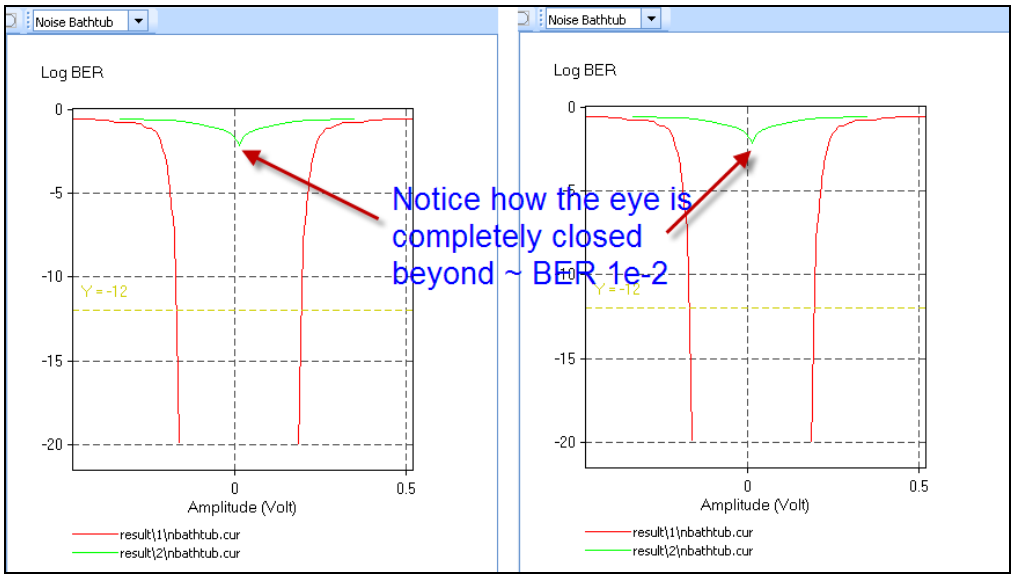
## 1.6 Increasing Data Rate

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:

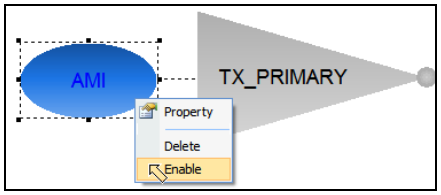



4. Notice how the eye (green curve) is completely closed.
5. Check the bathtub curves.

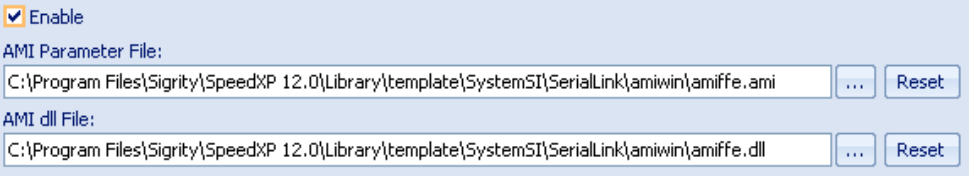



## 1.7 Enable AMI

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



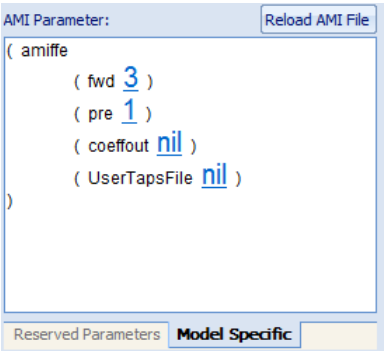
3. Double-click on the AMI block to view its properties. Two files have already been loaded by default:



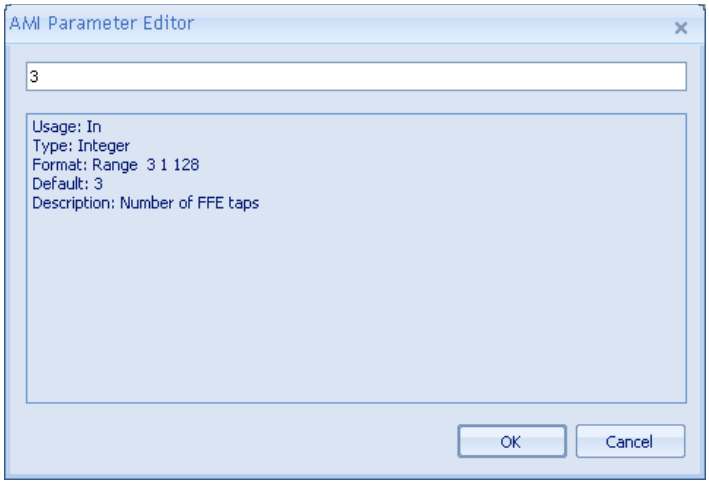

The **AMI** model assigned to the **TX_PRIMARY** block is either:

- **AMIFFE**
- **Feed Forward Equalizer**

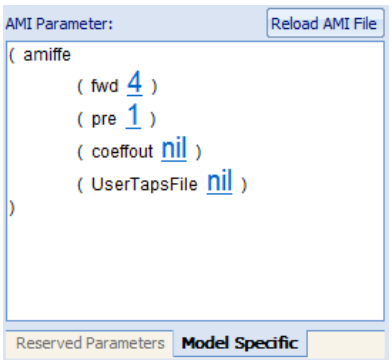
The parameters are displayed in the AMI parameter window.




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




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




6. Rerun the simulation.

### 1.7.1 Coeffout Parameter

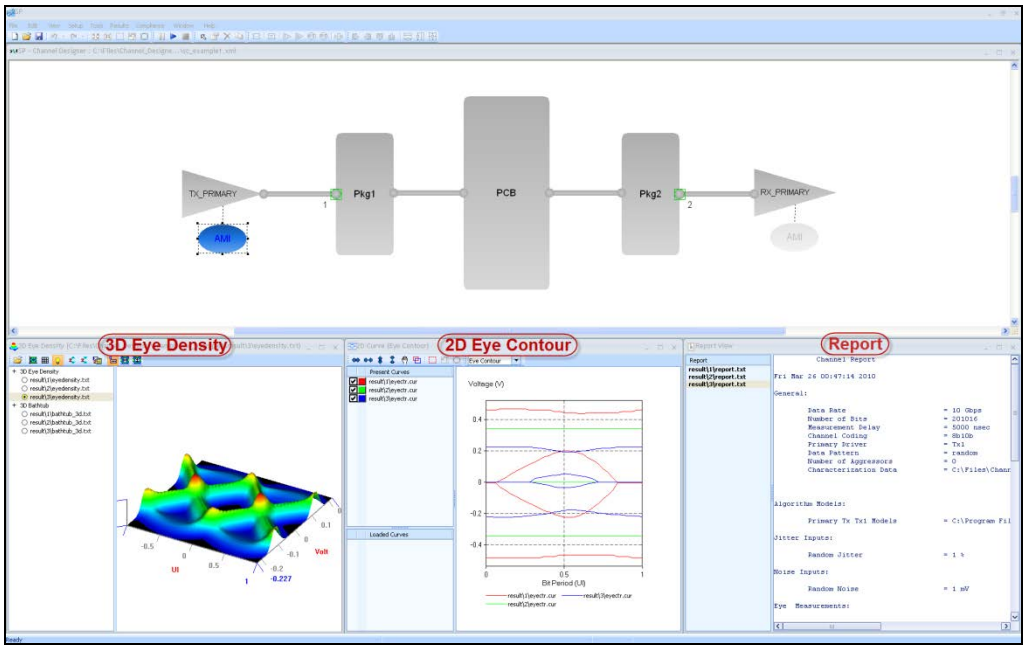
You can put a value for the **coeffout** parameter.

**Example**

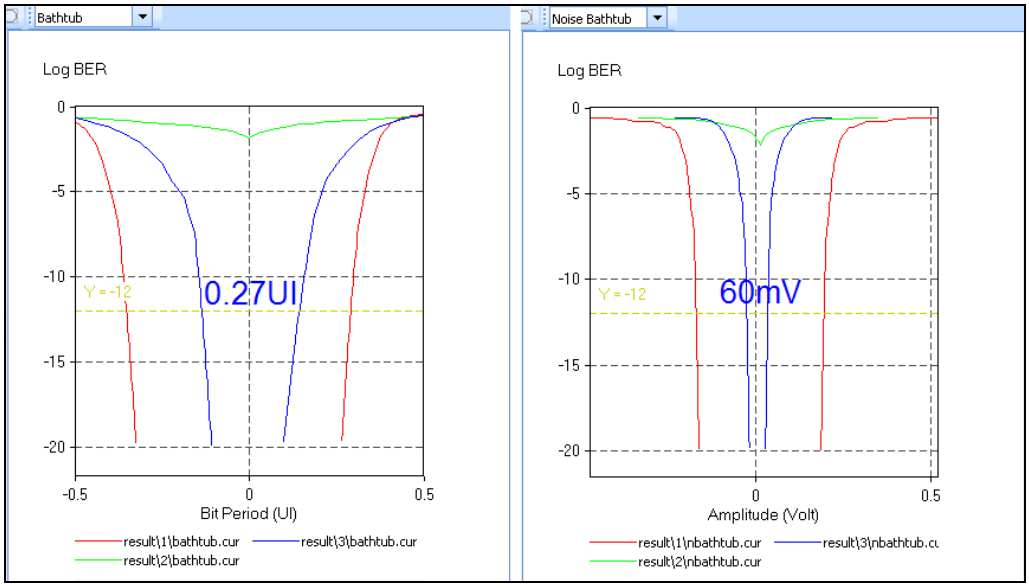
**`coeffout`** = **`coeff.txt`**.

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.

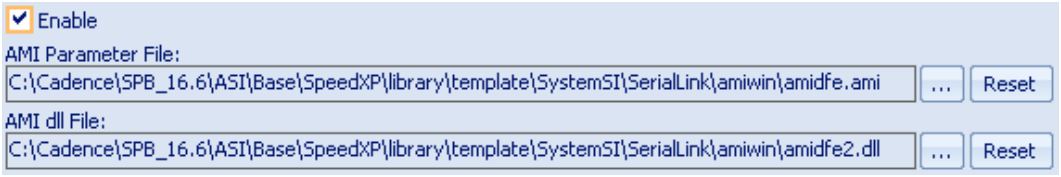



The closed eye is now open (blue curve). The next illustration shows the 2D Bathtub curves for the data rate = 10Gbps + TXFFE. Although the used FFE model does open the eye, the eye width at BER 1e-12 is only 0.27 UI compared to 0.64 UI for data rate of 3.125 Gbps. The eye height is only 60 mV compare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



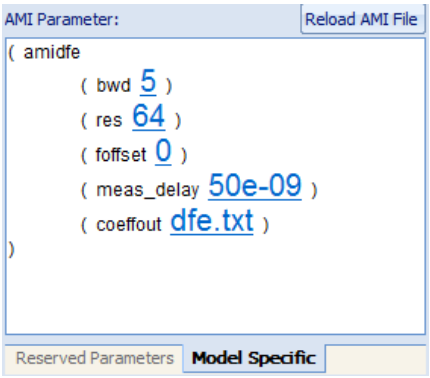

## 1.7.2 Enabling AMI DFE @ RX_PRIMARY

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



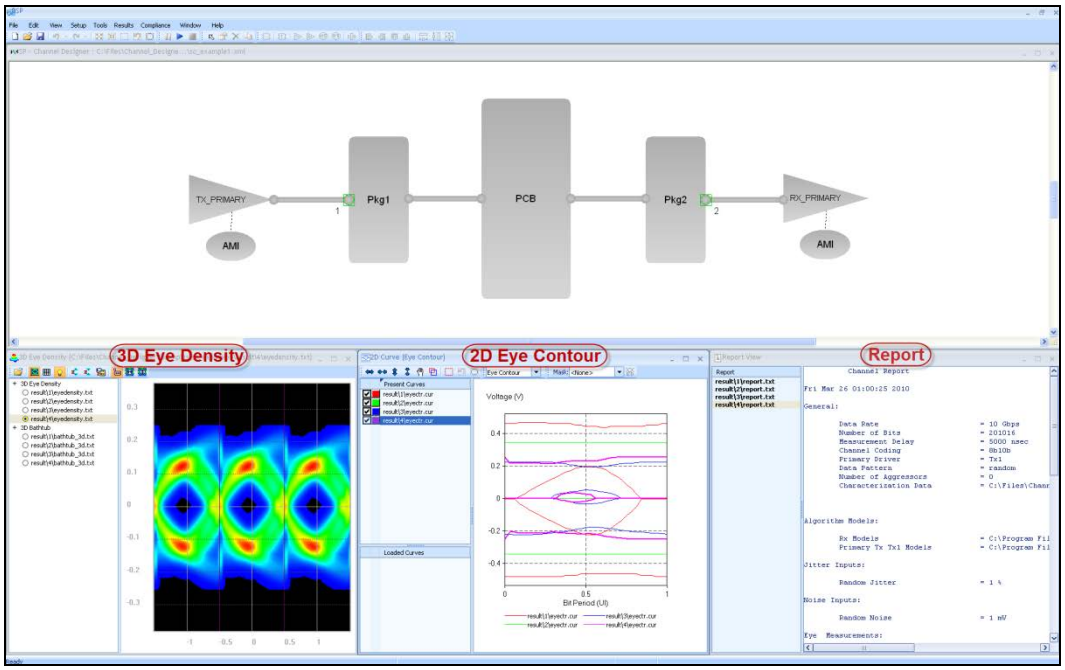

The default **AMI** model used at the receiver is **amidfe2** or **Decision Feedback Equalizer**.

3. Use the default parameters.



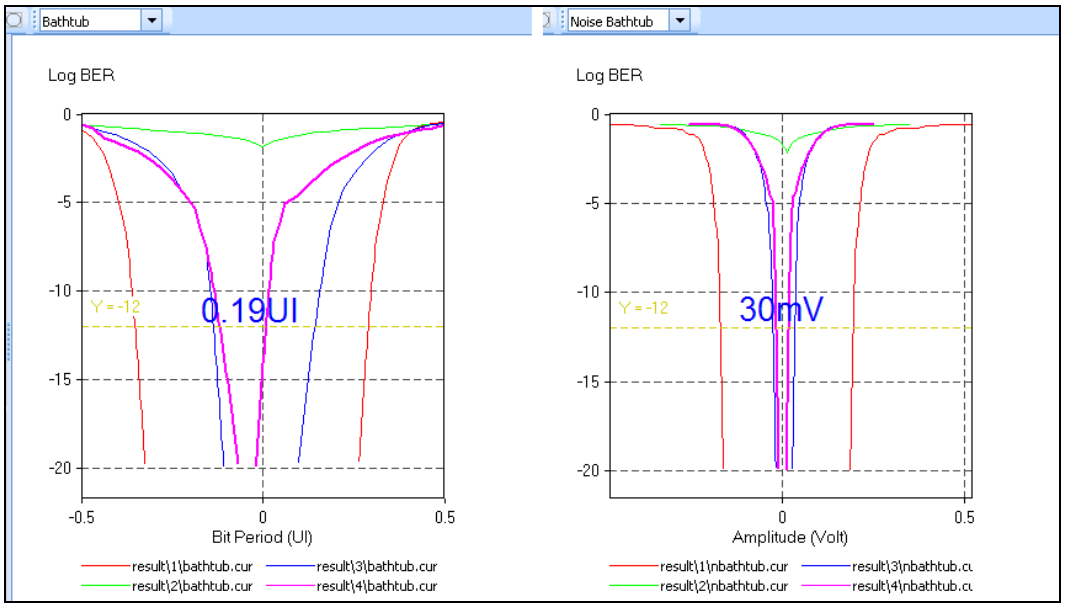

The Channel simulation results for data rate = 10 Gbps + TX FFE + RX DFE are shown in the next illustrations.

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.

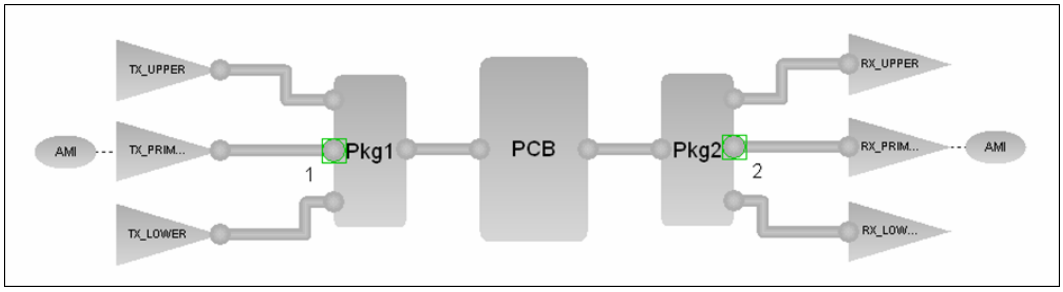
What-if scenarios are used to see how the channel behaves to changes in the data rate as well as the inclusion of Crosstalk and Equalization via **AMI** models.

## 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.



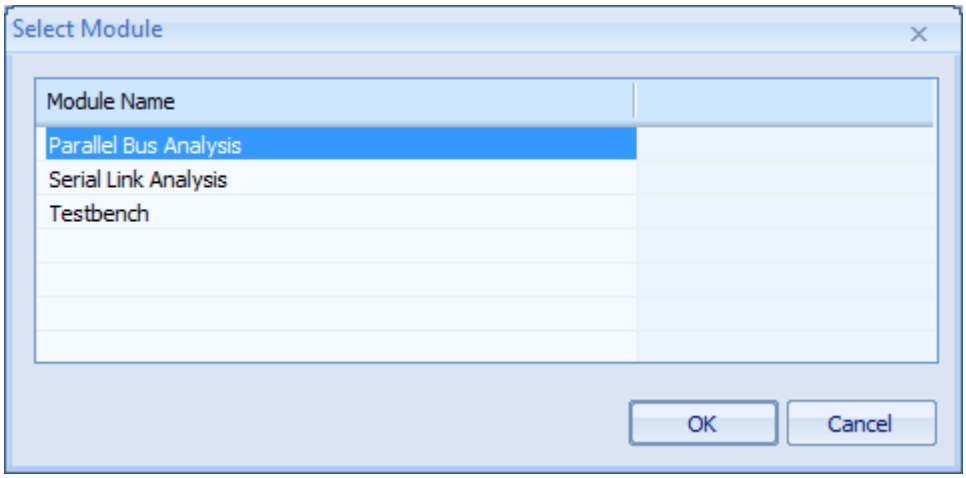

## 2.2 Starting the Crosstalk Channel Template

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**.

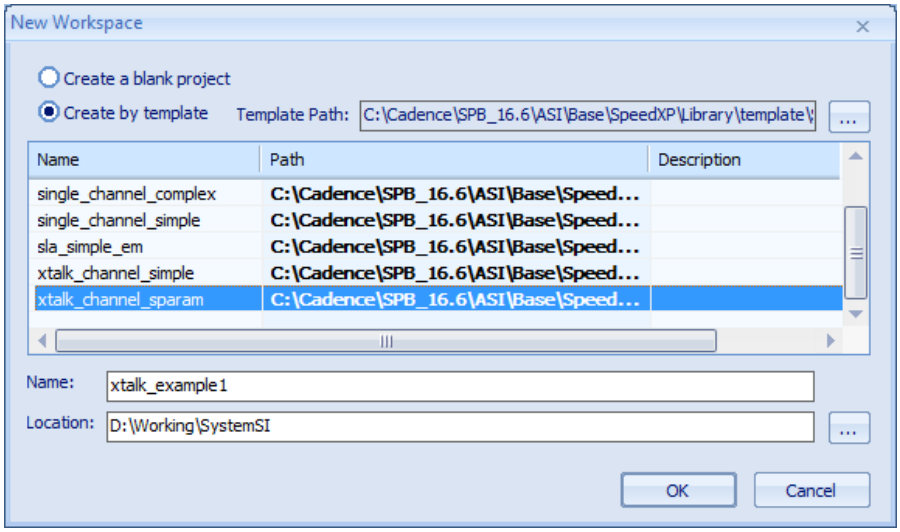
   Alternatively, you can select the **New** button  .

   The **Select Module** dialog opens.



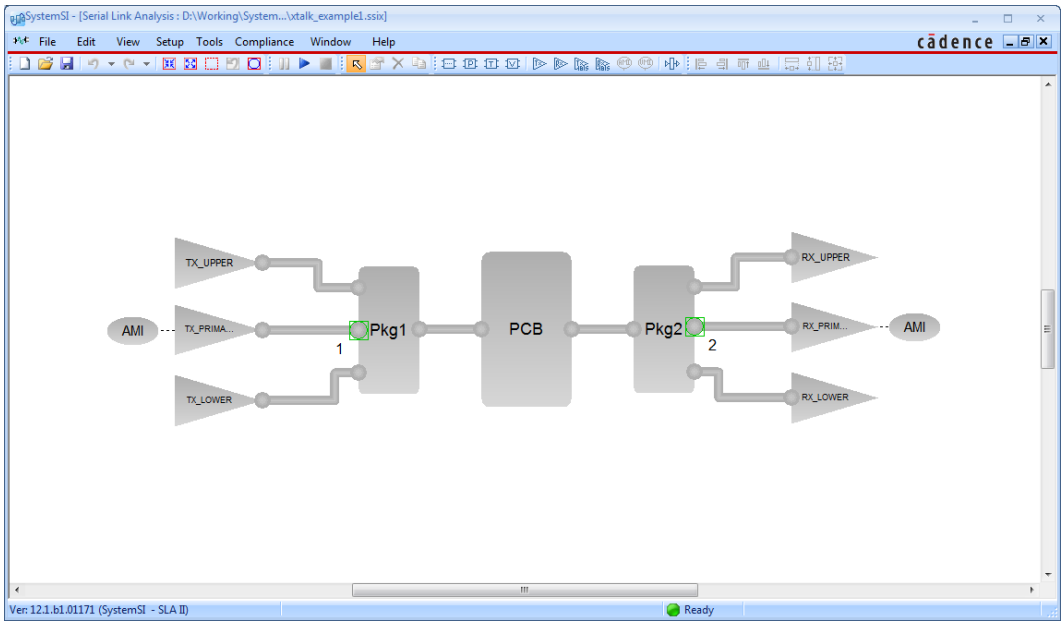

3. Select **Serial Link Analysis**.
4. Click **OK**.

   The **New Workspace** dialog for Crosstalk Channel appears.




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.

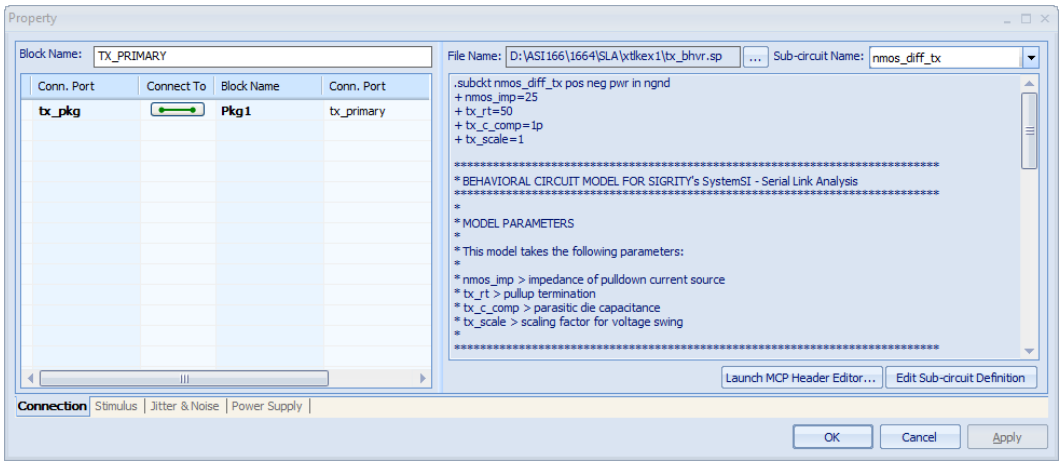
All of the properties for each block are already set in the Crosstalk Channel Template.

| | |
|---|---|
| **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.

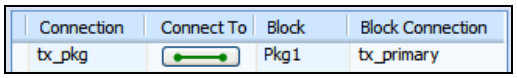



The template has three transmitter blocks and each block has four tabs.


### 2.3.1.1 Connection Tab

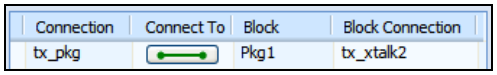
This tab shows information about the connections between the transmitter's block and other blocks.

#### 2.3.1.1.1 TX_PRIMARY



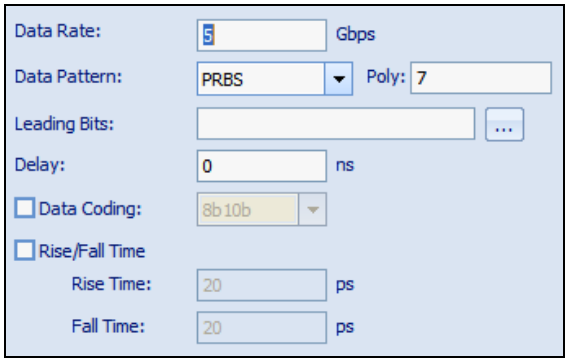

#### 2.3.1.1.2 TX_UPPER


#### 2.3.1.1.3 TX_LOWER



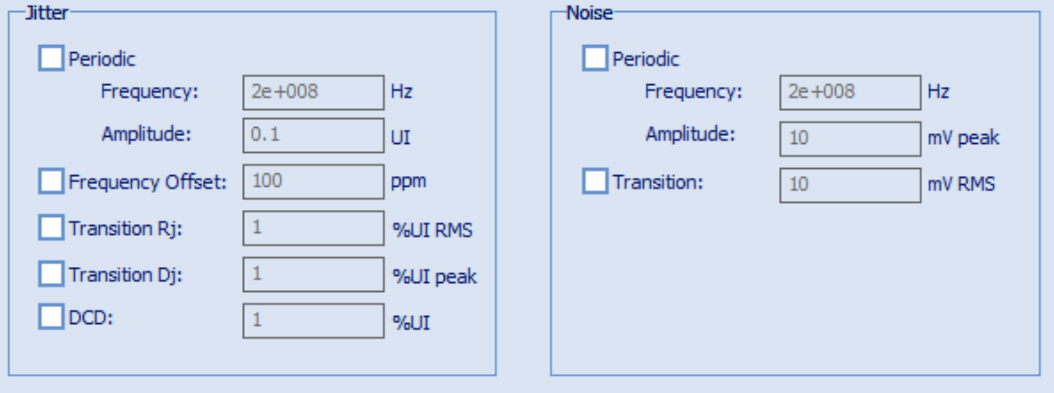

The primary difference between the three connections is in the block connection name. The names are for the connections at **Pkg1**.

### 2.3.1.2 Stimulus Tab



Please refer to ***Section 1.3.1.2 Stimulus Tab*** for details.

#### 2.3.1.3 Jitter & Noise Tab




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

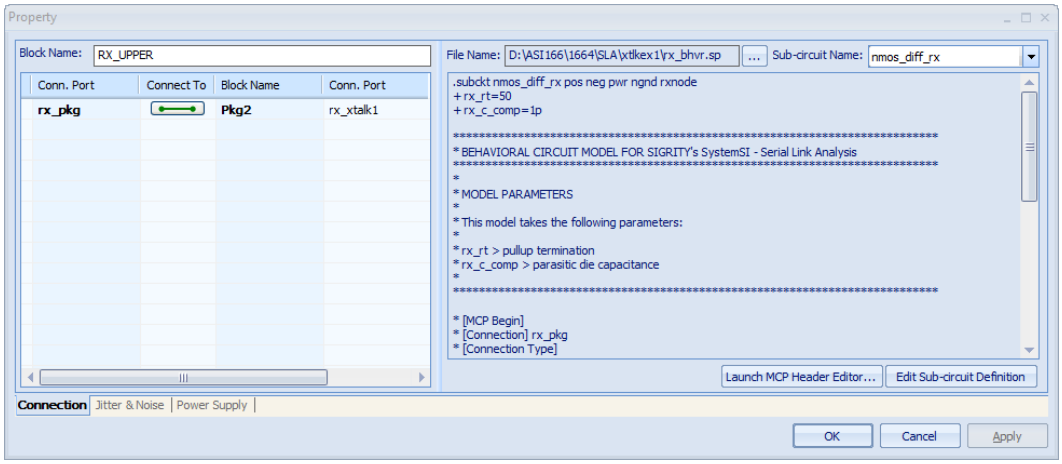
#### 2.3.1.4 Power Supply Tab


- **Power Supply** − Ideal voltage source that is used at the transmitter. Default value is 1 V.

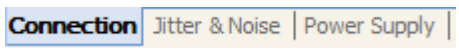
  Please refer to ***Section 1.3.1.4 Power Supply Tab*** for details.

### 2.3.2 Receivers

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



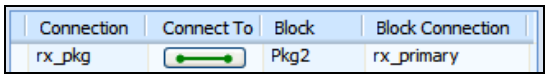

The property pane contains three tabs.



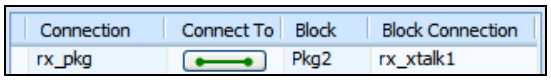

#### 2.3.2.1 Connection Tab

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

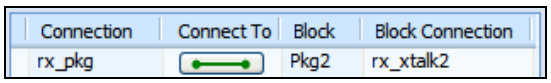
##### 2.3.2.1.1 RX_PRIMARY




#### 2.3.2.1.2 RX_UPPER

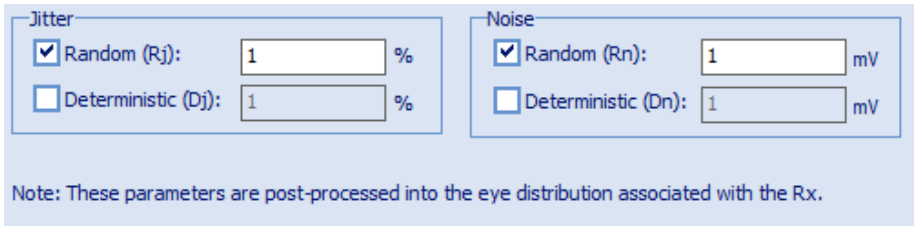



#### 2.3.2.1.3 RX_LOWER




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




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

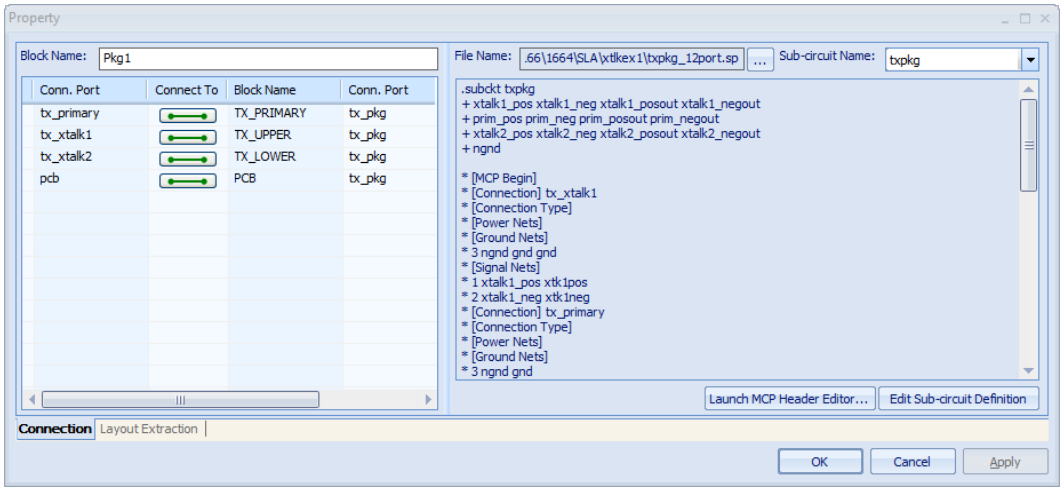
### 2.3.2.3 Power Supply Tab


- **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

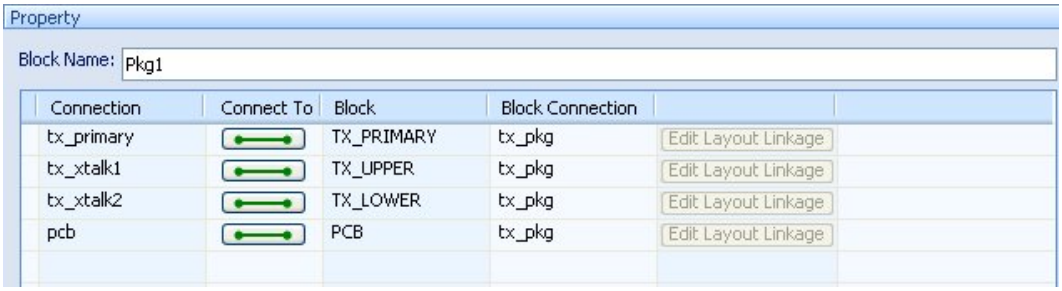
Double-click on the **Pkg1** block to view its properties.




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

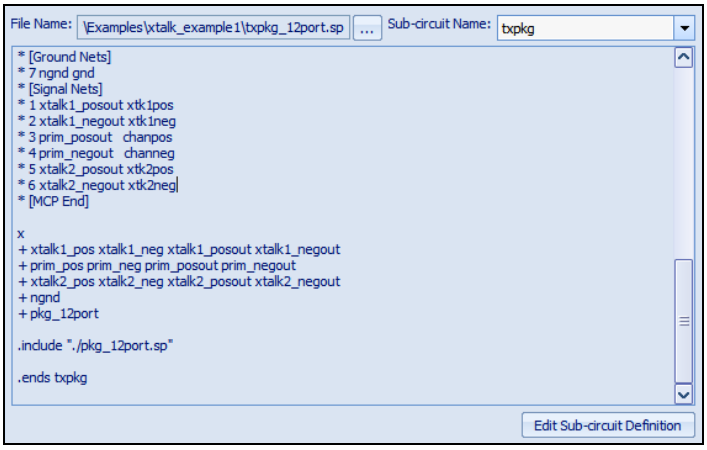
- Connection
- File and Content

#### 2.3.3.1 Connection




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

#### 2.3.3.2 File and content




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.

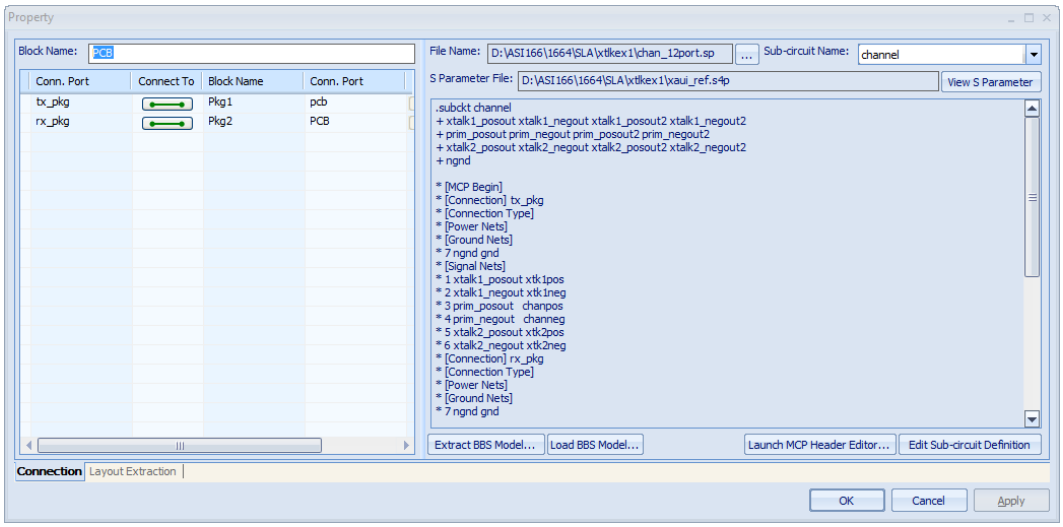
| **NOTE!** | Make sure to assign the nodes properly to maintain the connections with other blocks. |
|---|---|

### 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

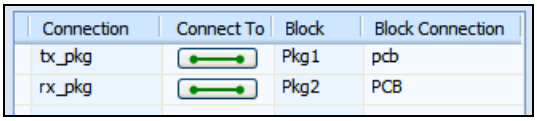
Double-click the **PCB** block to view its properties.




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

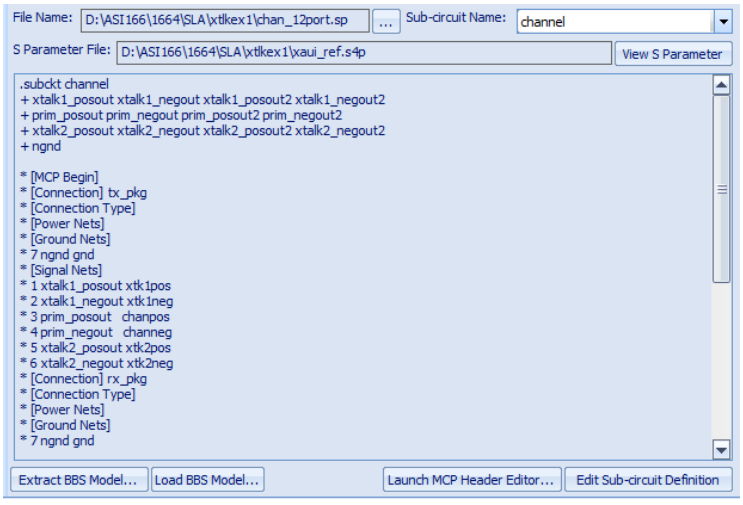
- Connection
- File and Content

### 2.3.5.1 Connection




The connection window shows the connections between the blocks.

### 2.3.5.2 File and content




## 2.3.6 AMI

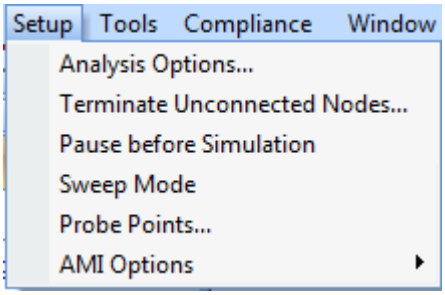
The **AMI** model (@ RX_PRIMARY) in gray are disabled by default.

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.

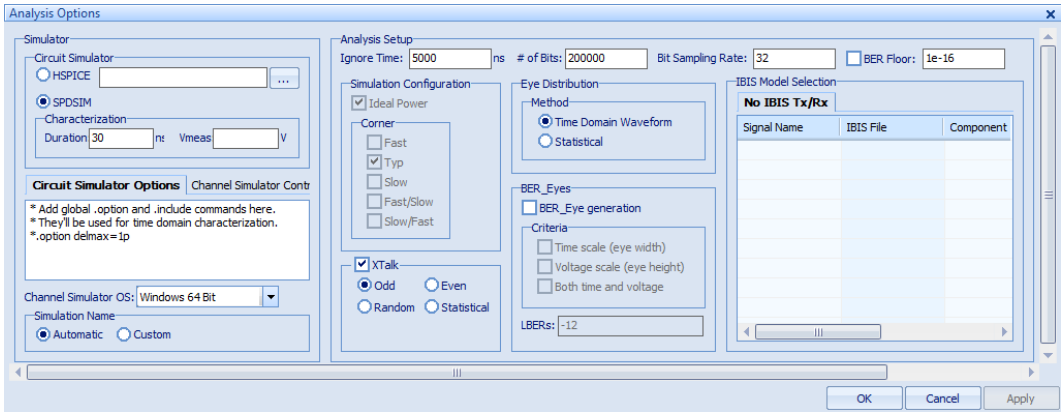
## 2.4 Analysis Options

This section covers the **Analysis Options** in the **Setup** menu.

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

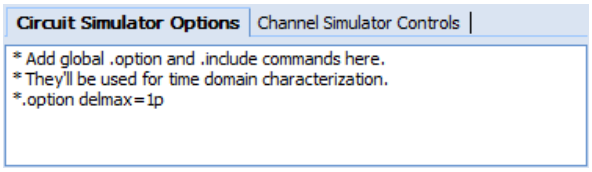



The **Analysis Options** interface opens.




### 2.4.1 Circuit Simulator Options

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




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

**`delmax=1p`** or **`delmax=2p`**.

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 180º 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

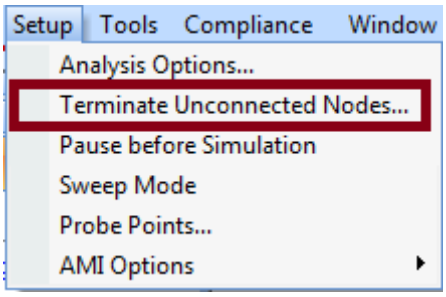
If there are some unused nodes in the block, SystemSI will automatically terminate the unused nodes.

By default, the termination value for Signal node is 50Ohm, for Power node is 1e+008Ohm and for Ground node is 0Ohm.

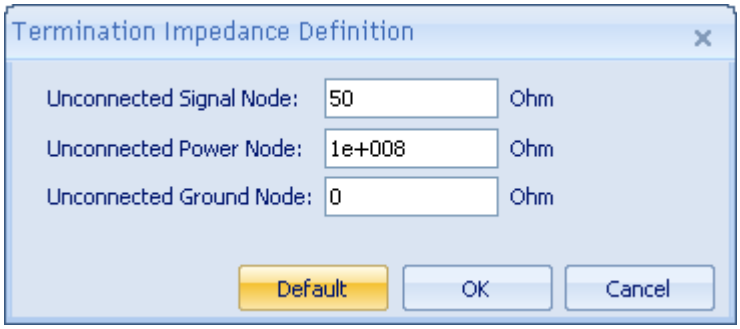
To change the termination value, follow these steps:

1. Select

   Setup > Terminate Unconnected Nodes….




2. The **Termination Impedance Definition** window pops up.




| 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:

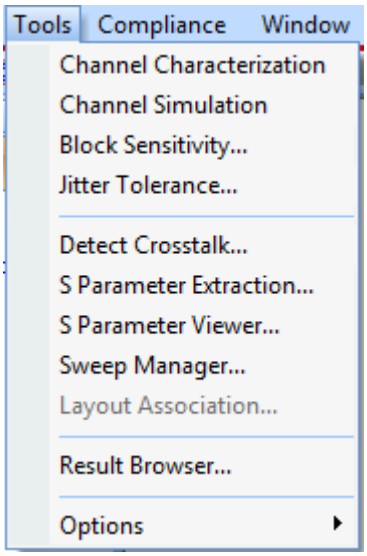
- AC Sweep
- First Simulation (Default settings)
- Enabling AMI DFE @ RX PRIMARY

- Turning on Statistical crosstalk

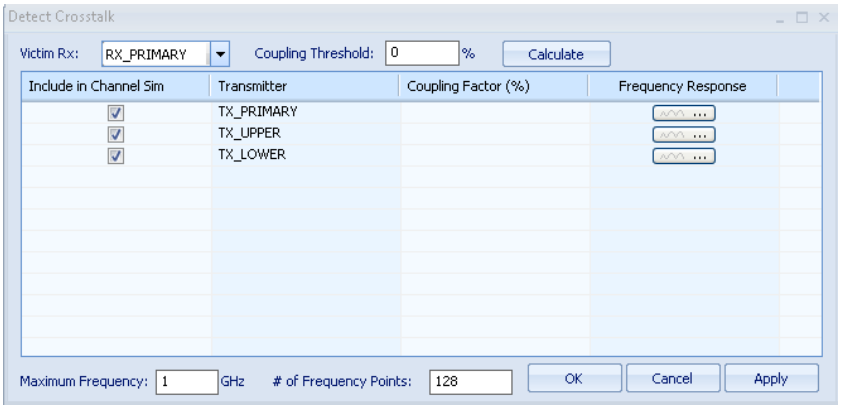
### 2.5.1 AC Sweep

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…**.

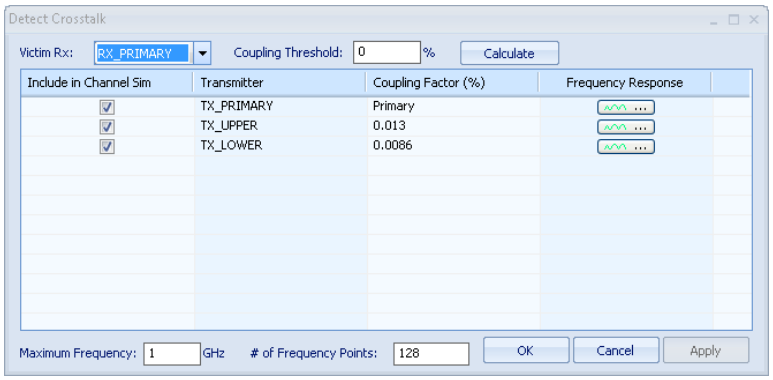



2. Select all transmitters.
3. Click **Calculate**.




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

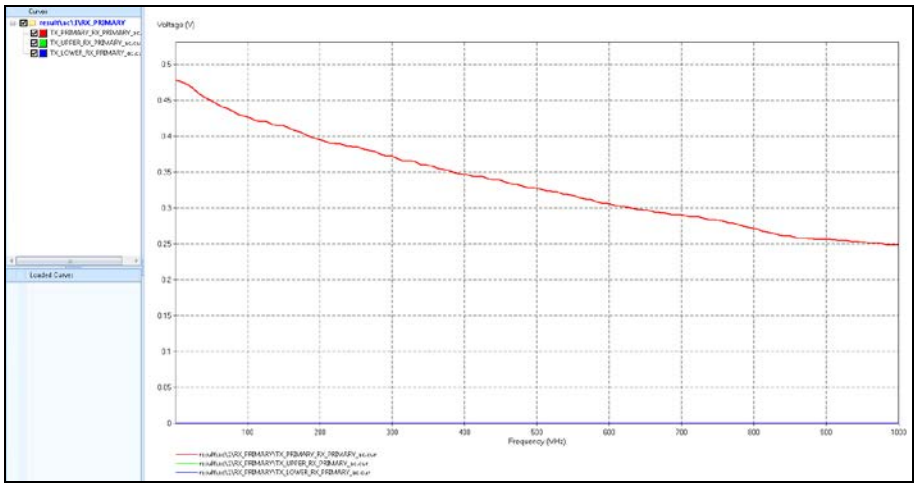
The **Primary** channel is identified as the one with the most power transferred from **TX** to **RX**.



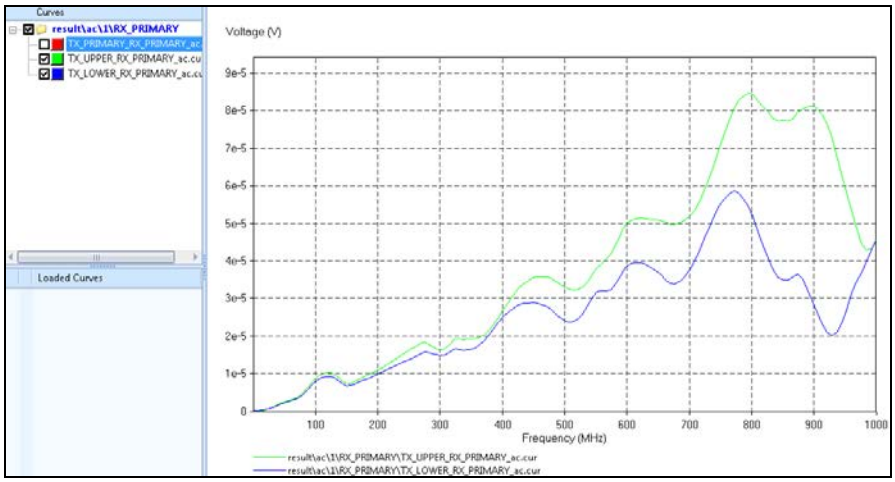

The default maximum frequency is 1 GHz. You can change the value to a higher frequency. Notice the impact on the coupling factor.

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

**Primary**



**Upper and Lower**



## 2.5.2 First Simulation (Default Settings)

Click the **Start Simulation** button 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. |
|---|---|

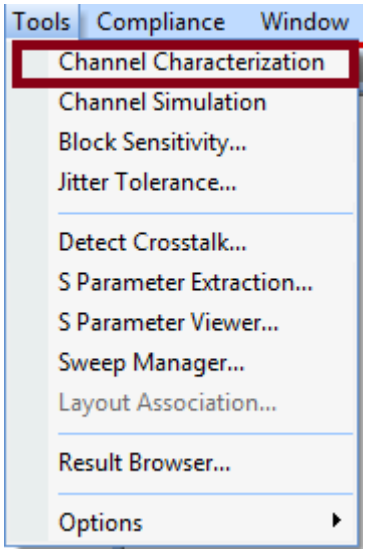
All the characterization information for the channel is stored in the Workspace directory.

### 2.5.2.1 Change Parameters

If you click 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.

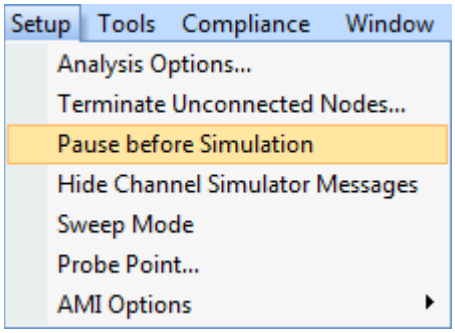
1. Select the **Tools** menu.
2. To characterize the channel only without running the channel simulation, click **Channel Characterization.**



### 2.5.2.2 Pause before Simulation

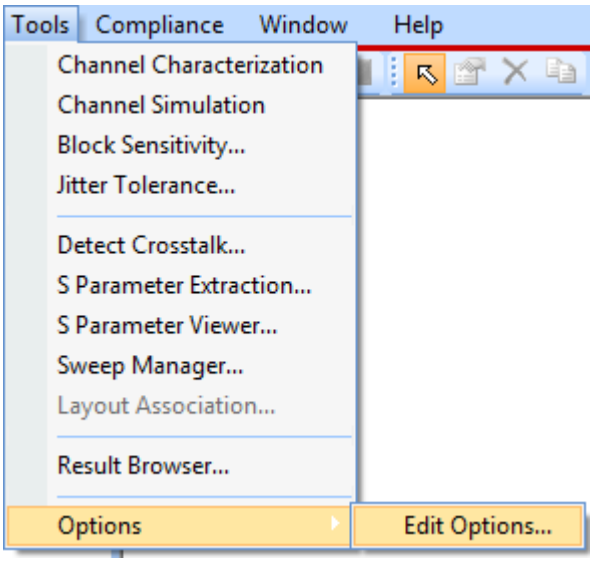
The **Pause before Simulation** option prompts you with a message whenever a new simulation is about to be run.

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

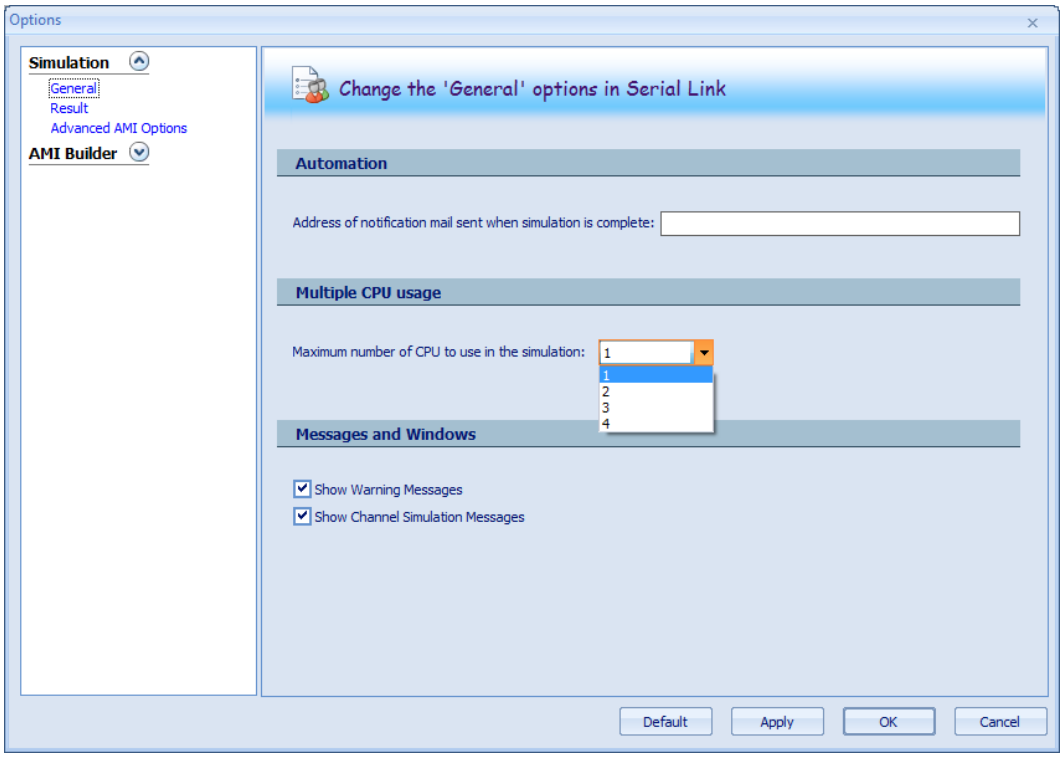



### 2.5.2.3 General Options

Select Tools > Options > Edit options….




The **Options** window opens.




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

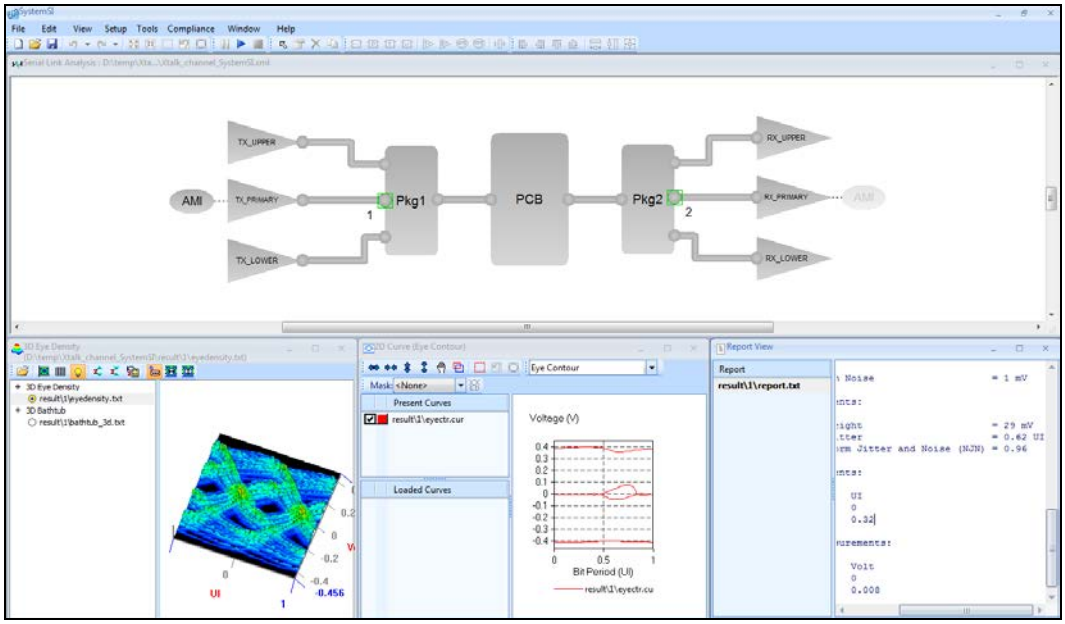
The simulation results consist of the following waveforms:

- 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.

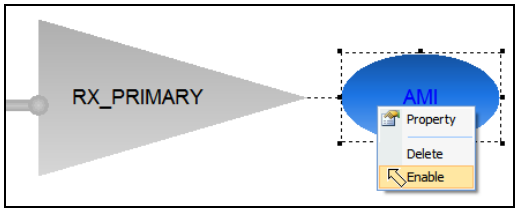
The following figure shows the Channel simulation results for Data rate = 5 Gbps. Running the channel simulation at a Data rate of 5 Gbps produces an eye contour that is not open very well.



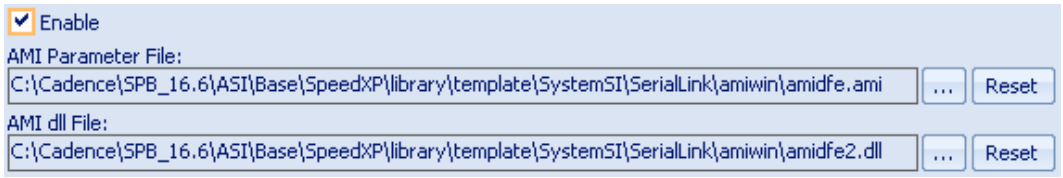

## 2.5.4 Enable AMI DFE

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

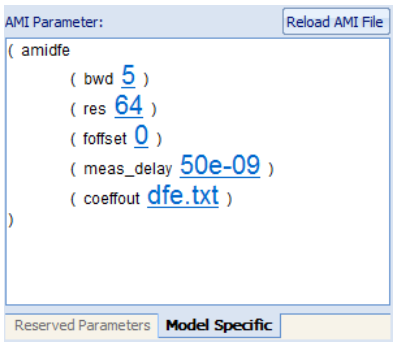
1. Run the simulation again.
2. Right-click on the **AMI** block.
3. Select **Enable** to enable the **AMI** block at the primary receiver.




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



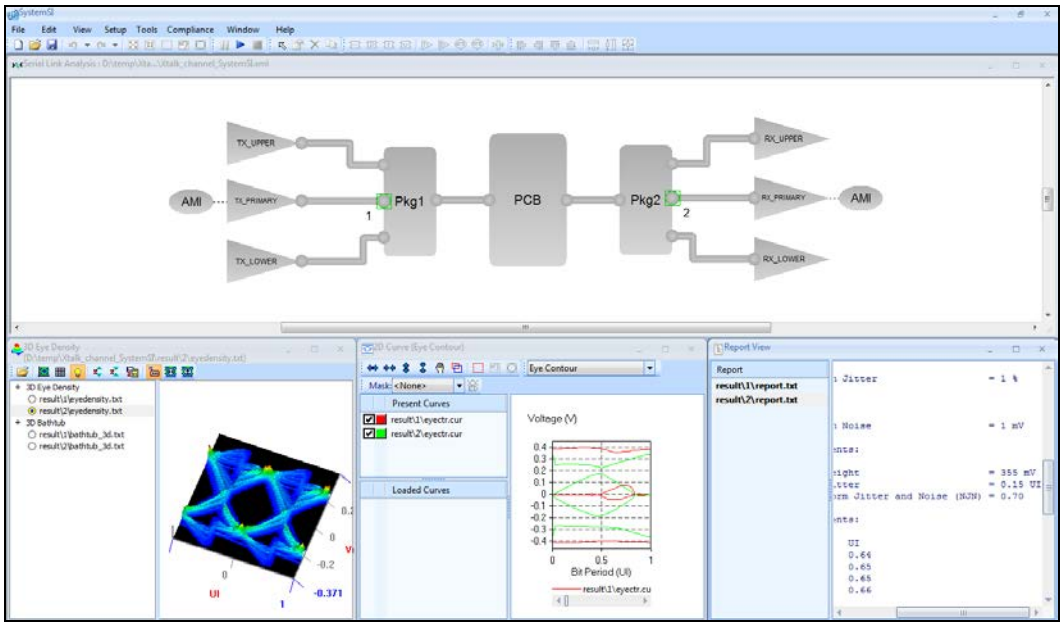

4. Use the default parameters shown in the example below:




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

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

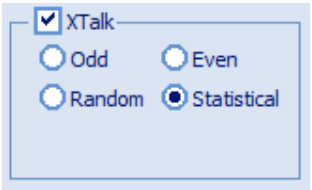
7. Notice the eye contour opens up nicely.



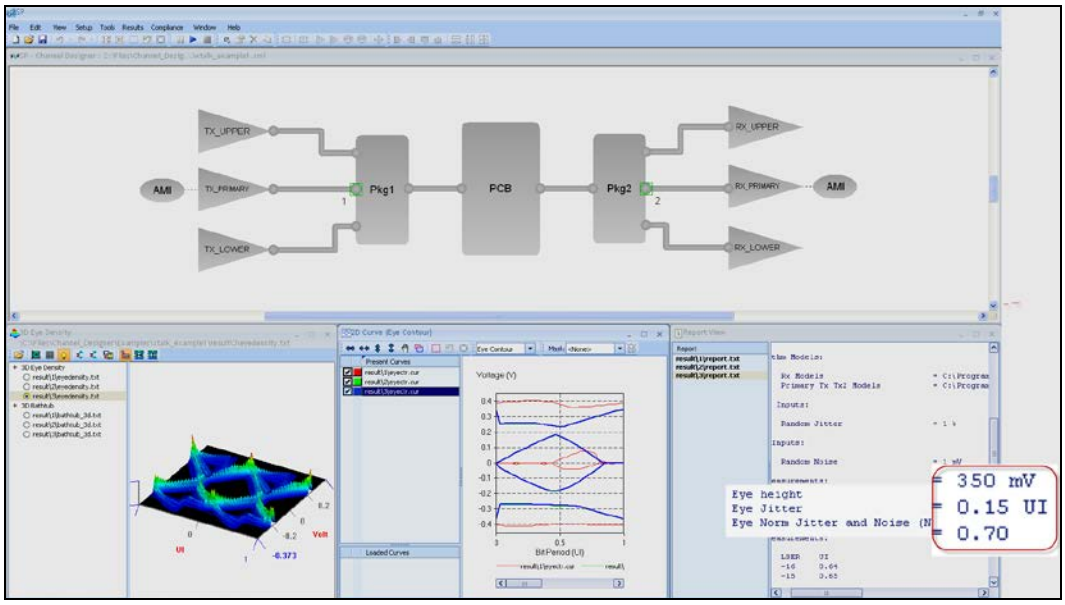

## 2.5.5 Turn on Statistical Crosstalk

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

1. Choose **Setup > Analysis Options…**.
2. Select **Statistical**.




The results show a Channel simulation results for Data rate = 5 Gbps + TXFFE + Statistical xtalk

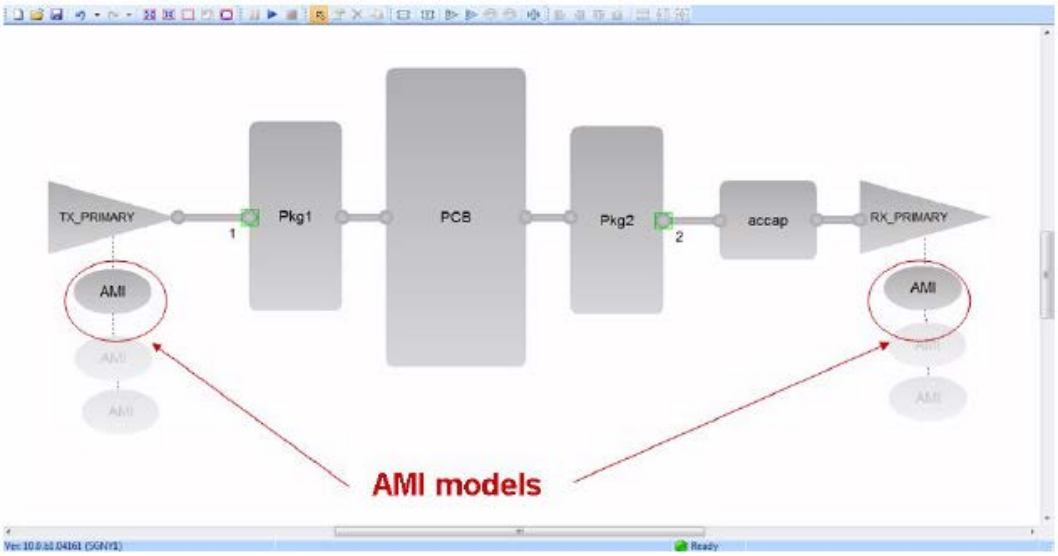



# 3 Sigrity AMI Models

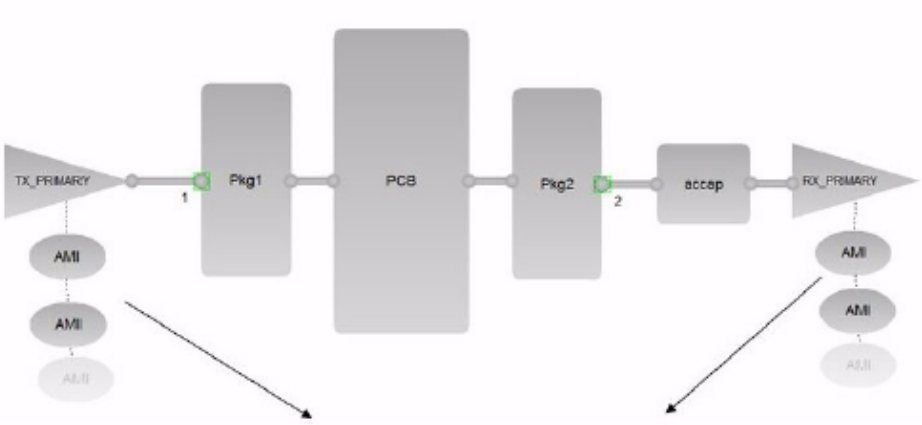
The Sigrity SystemSI – Serial Link Analysis includes several SERDES equalizer behavior models. All of these models use the Algorithmic Model Interface (AMI). This chapter describes using the Sigrity AMI models.

## 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).




Allows Flexible Modeling and Debugging
This example shows a Cascaded Equalizer Model with a stand-alone cdr

## 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 | Type | Values | Comment |
|---|---|---|---|
| res | integer | 64 32 128 | Resolution for recovery. |
| cdr_off | integer | 0 0 1 | 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 | Type | Values | Comment |
|---|---|---|---|
| profile | string | profile.txt | Transfer function table in either frequency 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 | Type | Values | Comment |
|---|---|---|---|
| dbl | float | 6 6 15 | Initial dB loss of filter. |
| f0 | range | 1e9 0.5e9 3e9 | Cutoff frequency. |
| adapt_on | integer | 1 0 1 | Adaptation ON/OFF switch. |
| adapt_cyc_latency_transitions | integer | 256 | Adaptation cycle in UI. |
| adapt_freq_factor | integer | 128 | The quantized tap coefficients are output to a text file qffecoeff.txt. |
| cdr | string | cdr dfe off | Integrated CDR, CDR together with DFE or neither one enabled. |
| cdr_off | integer | 0 0 1 | 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. |

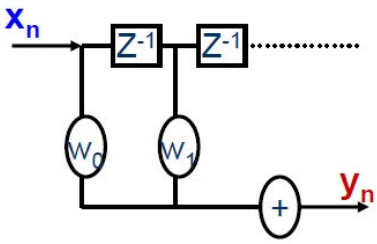
### 3.2.4 AMIFFE Configuration

Feed Forward Equalization is represented mathematically as:

$$y_n = \Sigma\, w_i * x_i$$

$x_n$ - input

$Y_n$ - output

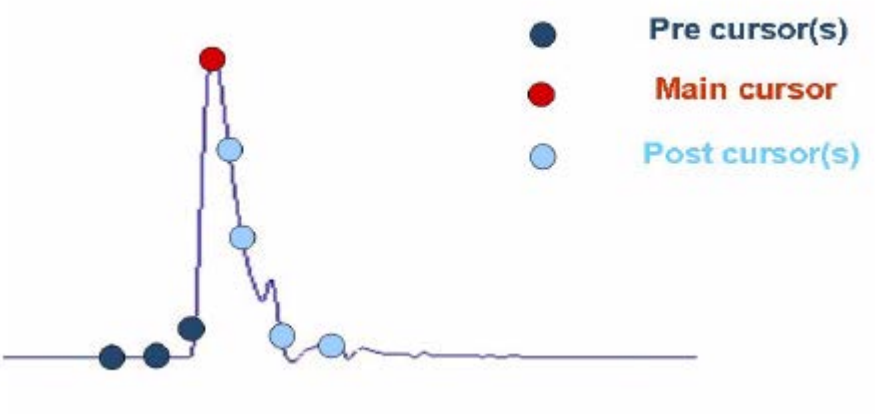



### 3.2.4.1 AMIFFE Description

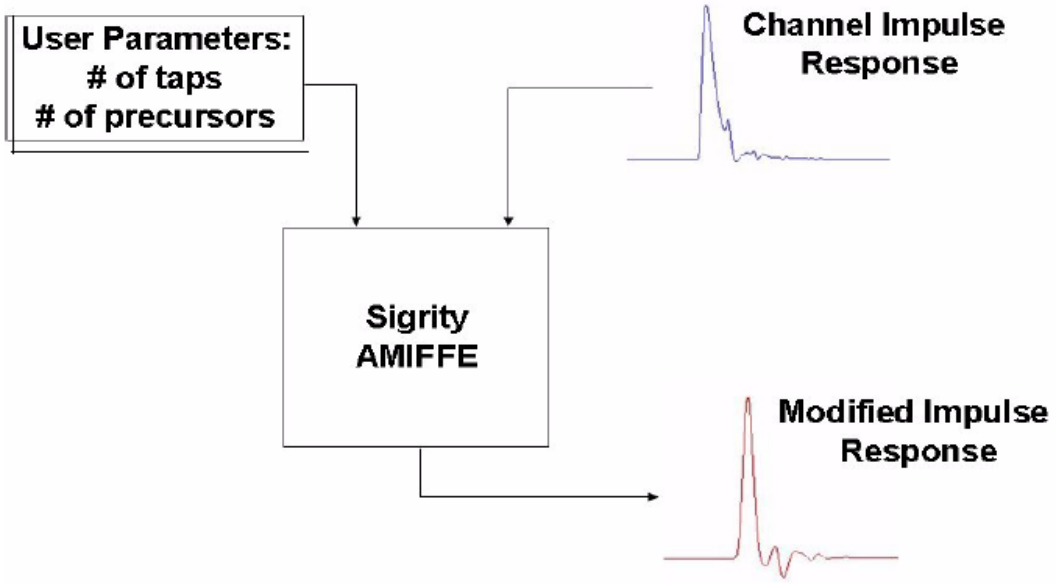
AMIFFE is generally applied at the **Tx** side. It can be applied to the **Rx**. They are mathematically equivalent. The filter is specified by:

- **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 | Type | Values | Comment |
|---|---|---|---|
| fwd | Integer | 2 1 128 | Forward Taps. Includes main cursor. |
| pre | Integer | 1 0 5 | Precursor taps. |
| coeffout | Integer | ffcoeff.txt | The tap coefficients are output to the text file **ffcoeff.txt**. |
| UserTapsFile | String | nil | User-supplied Tap Coefficient from a file. By default it is set to **nil** meaning no user-supplied Tap Coefficients. If the user supplies the coefficients no Optimization 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 | 0 0 1 | Setting = 1 forces sum of normalized Tap Coefficients equal to 1. |
| qcoeffout | String | qffcoeff.txt | The quantized Tap Coefficients are output to the text file **qffcoeff.txt**. |
| offset | Float | 0 -0.5 1.5 | Tweaks the automatic error minimization algorithm for Tap Coefficient synthesis. |
| OptimizerPulse | Integer | 0 0 1 | Tweaks the FFE coefficient generation algorithm. Setting this parameter may improve the coefficients. |
| refine_coeff | Integer | 0 0 1 | Setting = 1 gives an additional stage of non-linear Tap Optimization. Can improve over standar MMSE algorithm. Can use user-supplied starting coefficients. |
| 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:

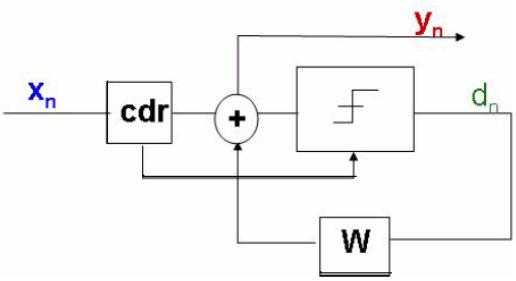
$$y_n = X_n + \Sigma\, w_i * d_i$$

$y_n$ - output

$x_n$ - input

$d_i$ - previous $i^{th}$ decision

$w_i$ - $i^{th}$ 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 pre-cursor 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:

$$w_i(n + 1) = W_i(n) + u * E * d_m(n)$$

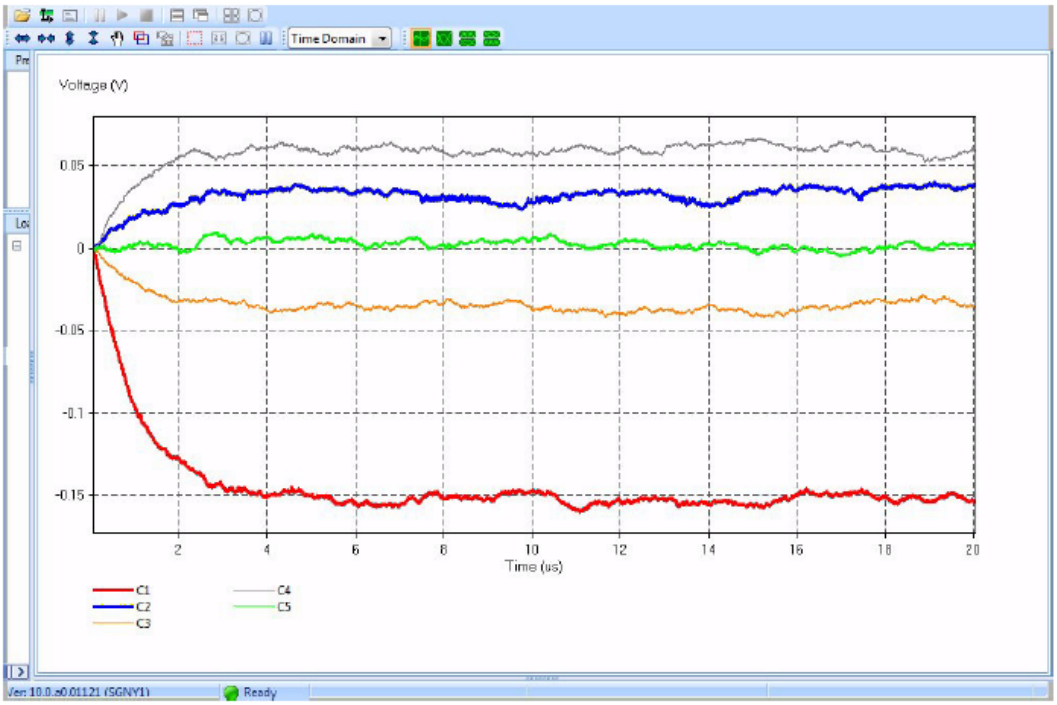
$W_i$ − $i^{th}$ coefficient

$d_i$ − $i^{th}$ symbol

E − error

u − proportionality factor

### 3.2.5.3 DFE Coefficient Dynamic Adaptation Example

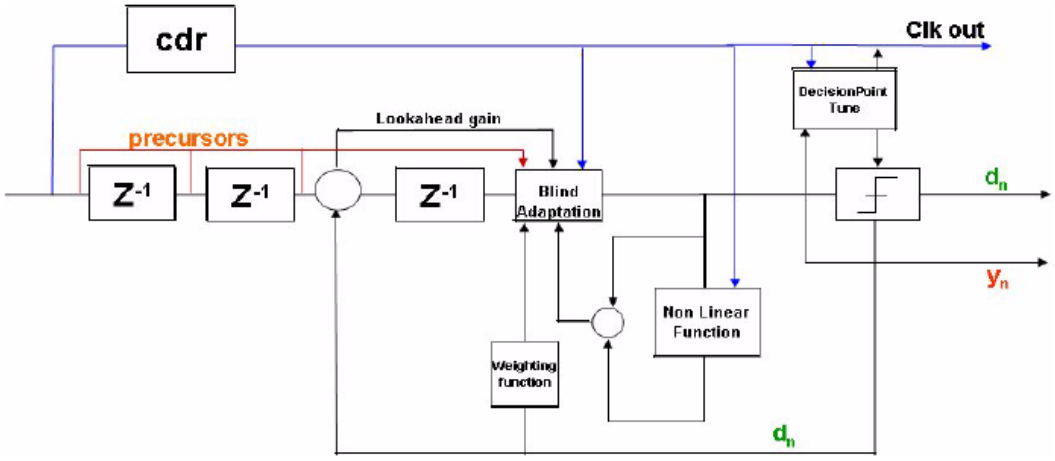


#### 3.2.5.4 AMIDFE2 User Parameters

| Parameter | Type | Values[1] | 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 | Type | Values[1] | 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 | 0 0 10 | Number of precursor Taps. Default = 0. Not as effective as dedicated pre-emphasis in transmitter. |
| lookahead | Integer | 0 0 1 | 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 **lookahead** 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 inactive. -1/+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

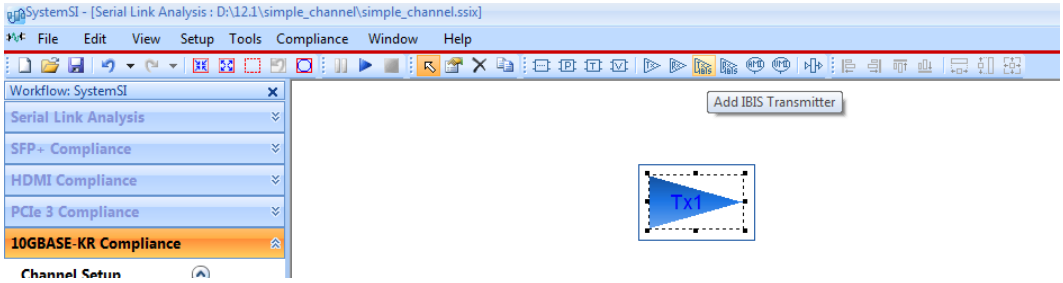
Use these capabilities to further investigate channel performance and optimize the design.

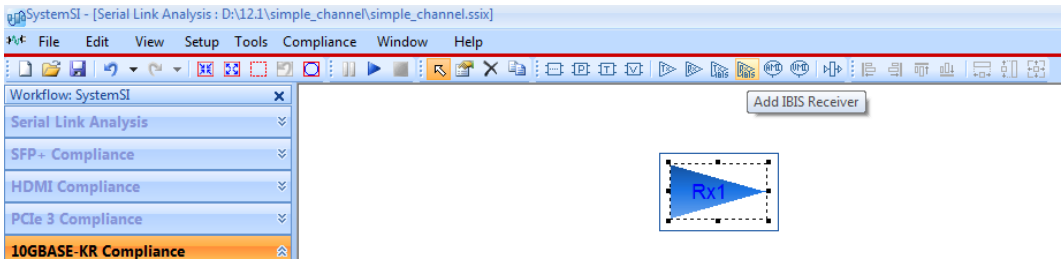
## 4.1 IBIS Transmitter and Receiver

### 4.1.1 Add IBIS Transmitter and Receiver

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

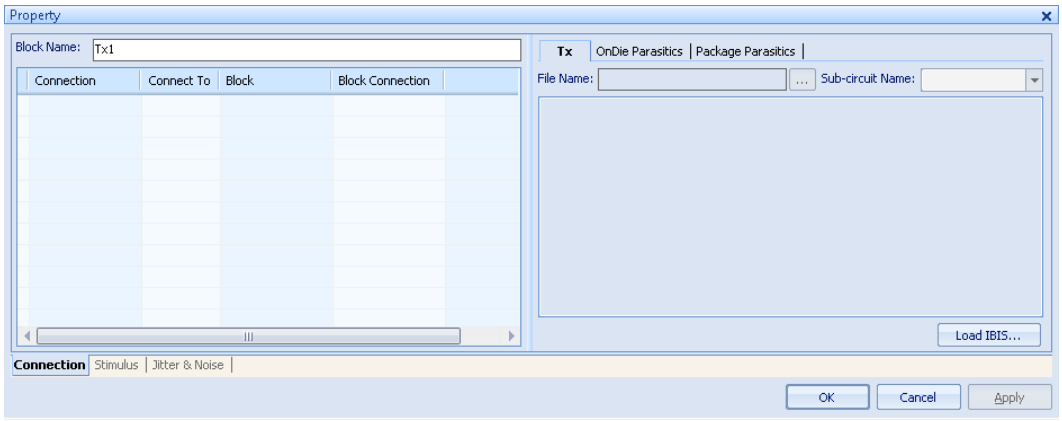
   The **Tx1** or **Rx1** block is added.



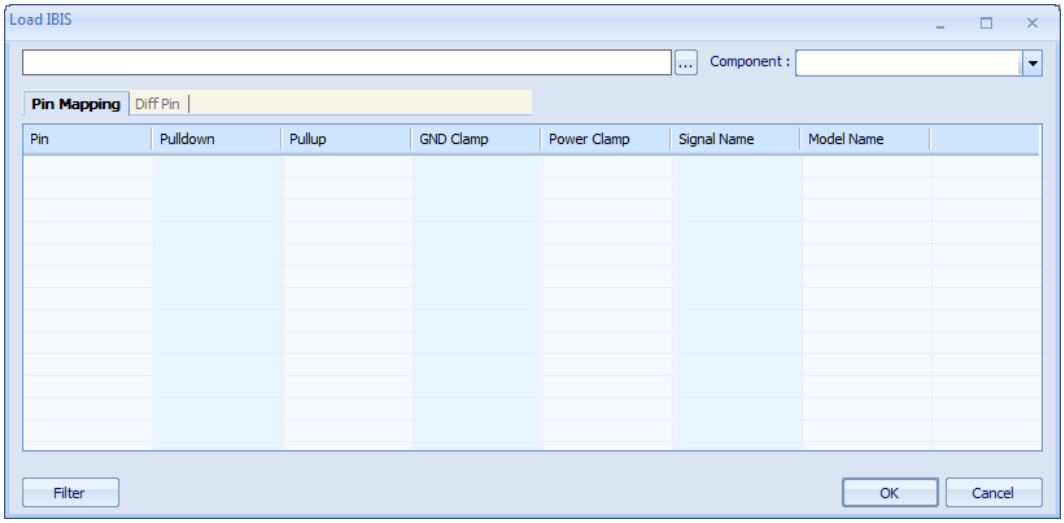




### 4.1.2 Load an IBIS File

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

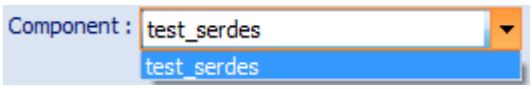



2. Click the **Load IBIS…** button. The **Load IBIS** window opens.




3. Click the  button to load an IBIS file.

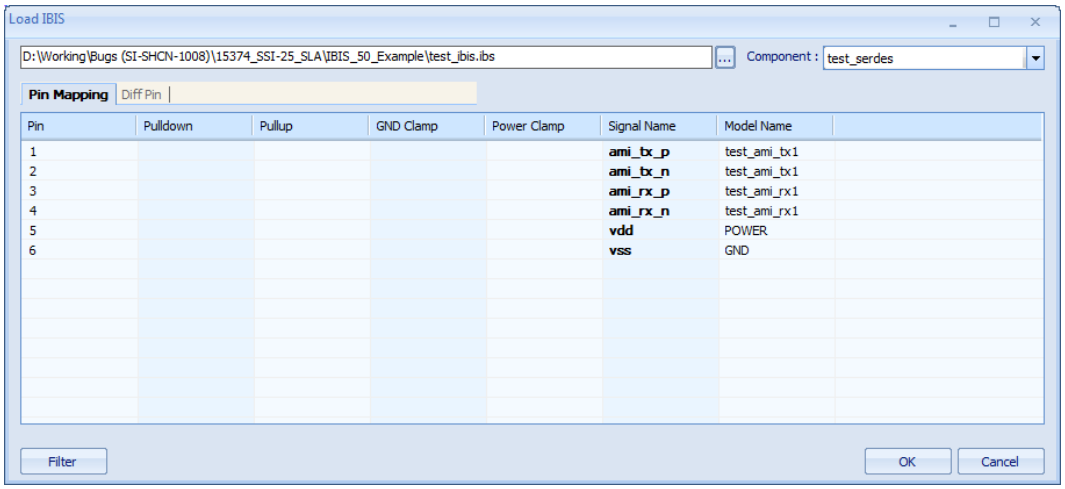
4. Choose from the **Component** drop-down list to select a component for the IBIS Tx block or IBIS Rx block.



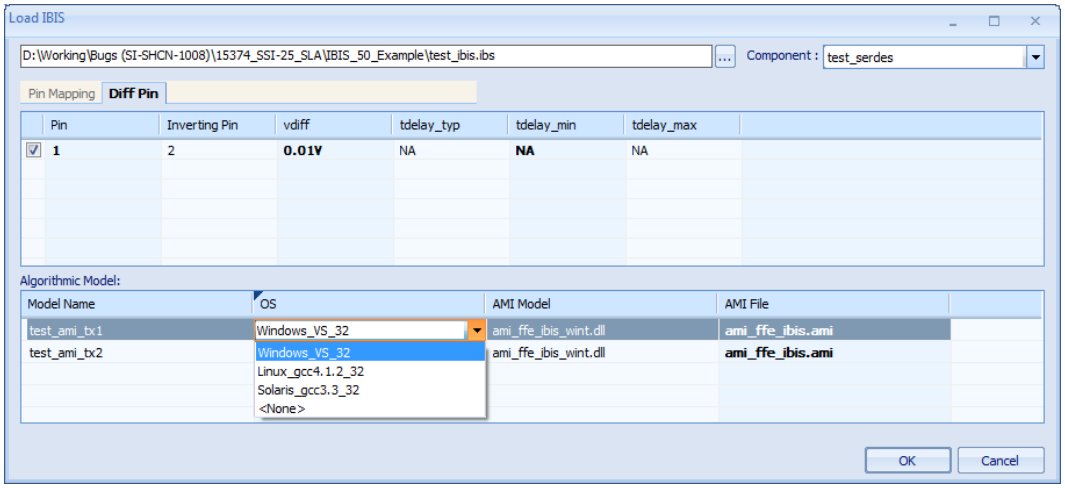

The **Pin Mapping** tab is shown.

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.

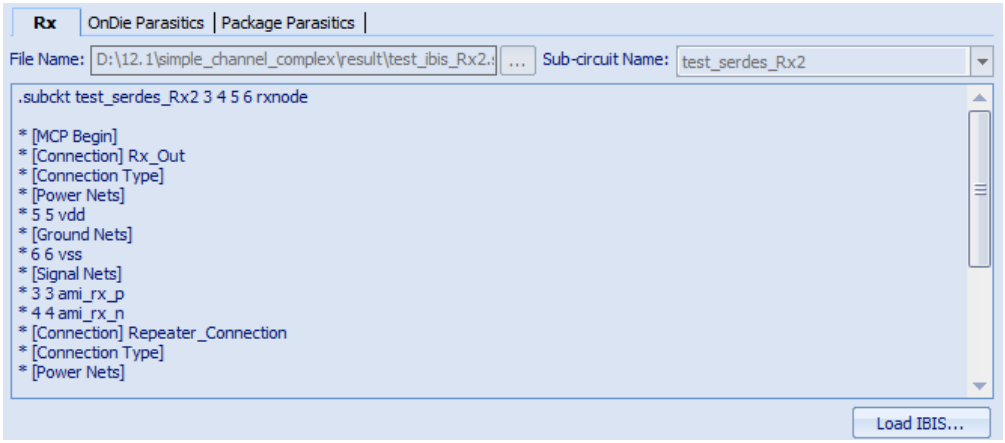



5. Click the **Diff Pin** tab, and select a diff pin pair.




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

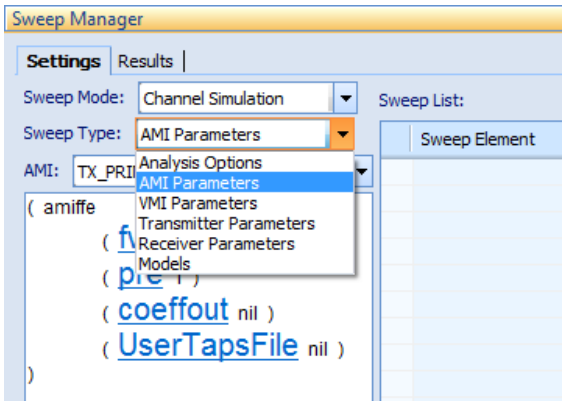
If the IO model is selected for the simulation, the selected AMI model will be used. The `.sp` file for the selected IBIS component will be automatically created and assigned to the IBIS block.



## 4.2 Sweep Manager

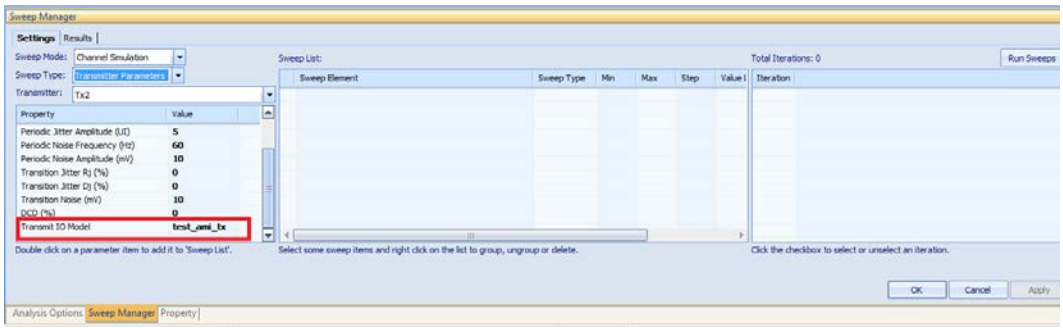
The exercise uses one of the single channel templates. The types of sweeps are explained and in different case examples.

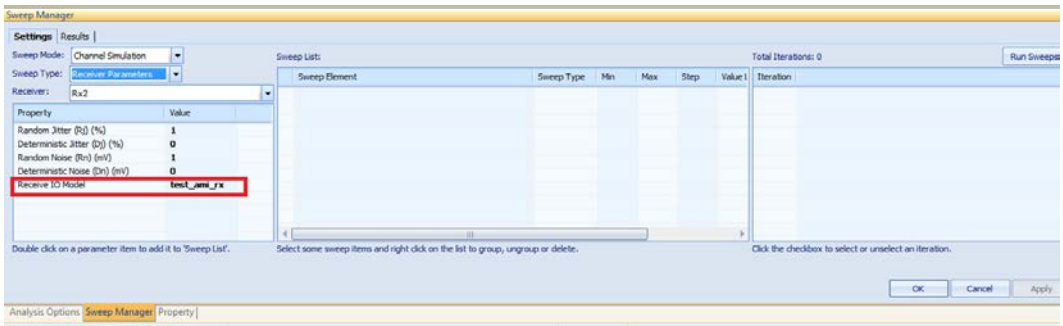
The **Sweep Manager** supports six types of sweeps.




- **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.



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

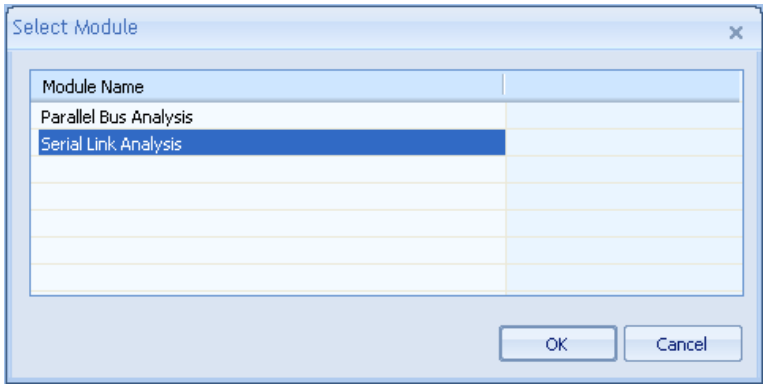
### 4.2.1 Launch Single Channel Complex Template

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

   or

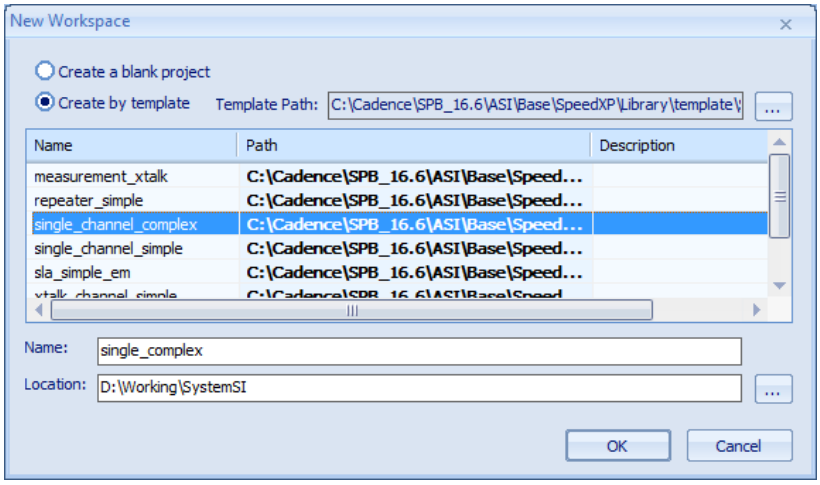
   in the Main toolbar, click the **New** button.

   The **Select Module** dialog opens.

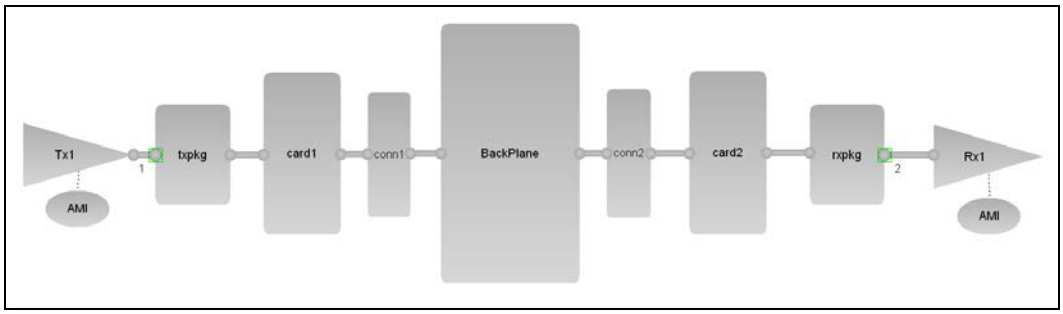



3. Select **Serial Link Analysis**.
4. Click **OK**.

   The **New Workspace** dialog (Single Channel Template) opens.




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

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

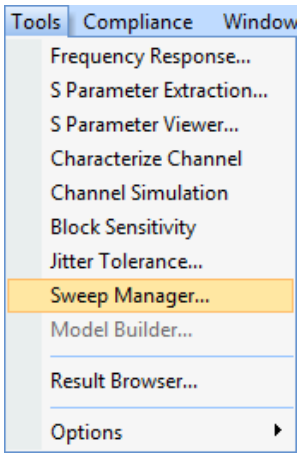
#### 4.2.1.1 Block Models

Each block contains the following models.

| Component | Circuit model |
|---|---|
| **Tx** | 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

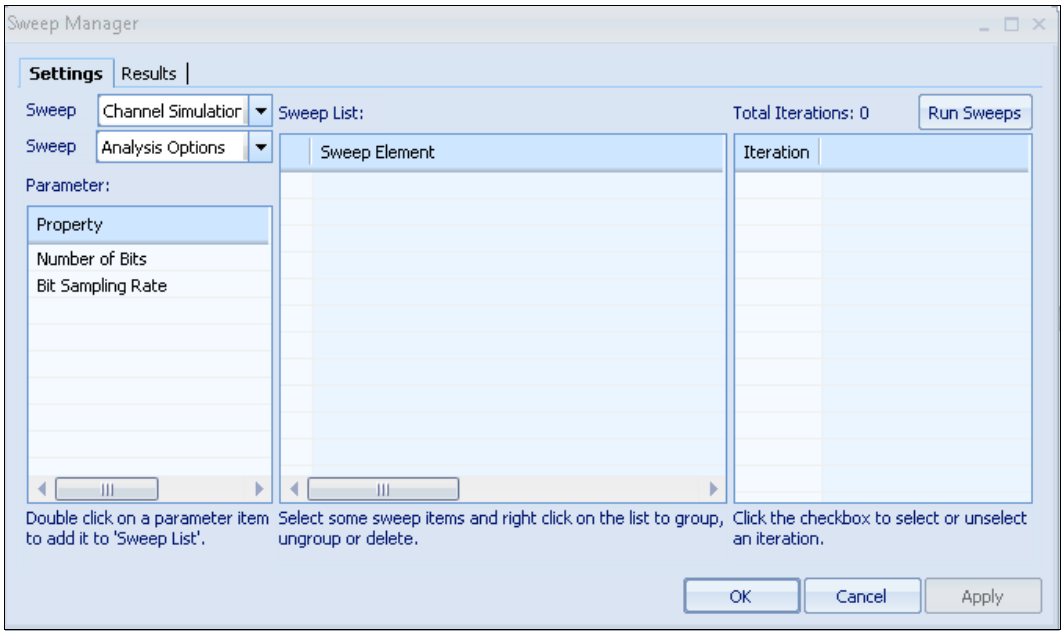
- Start in the **Tools** menu.




- Click **Sweep Manager…**.

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

- Parameter
- Sweep List
- Iteration




#### 4.2.2.1 Sweep Mode

The two options supported are:

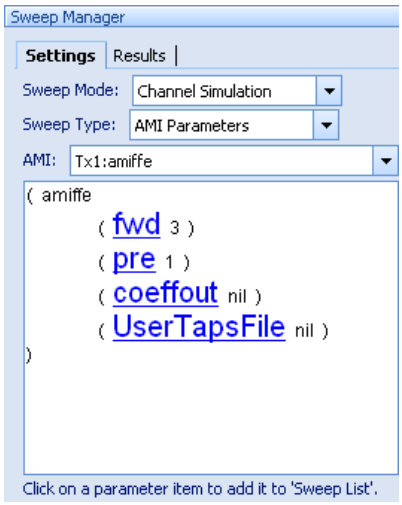
- Channel Simulation
- Frequency Response

#### 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**.

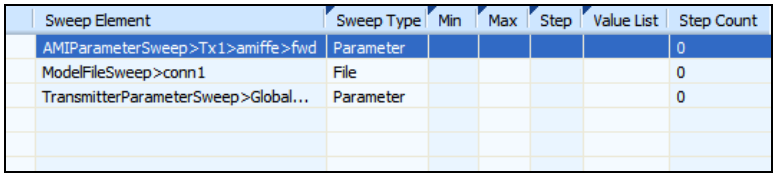
In the example shown, the sweep name appears in the **Sweep List** if you click on **fwd**.




### 4.2.2.3 Sweep List

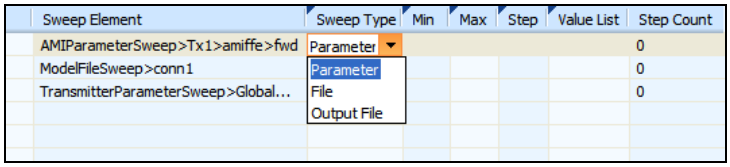
To choose parameter values, select the Sweep item from the Sweep List and go to the next section. The **Sweep List** pane contains these items:

- **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 Type**

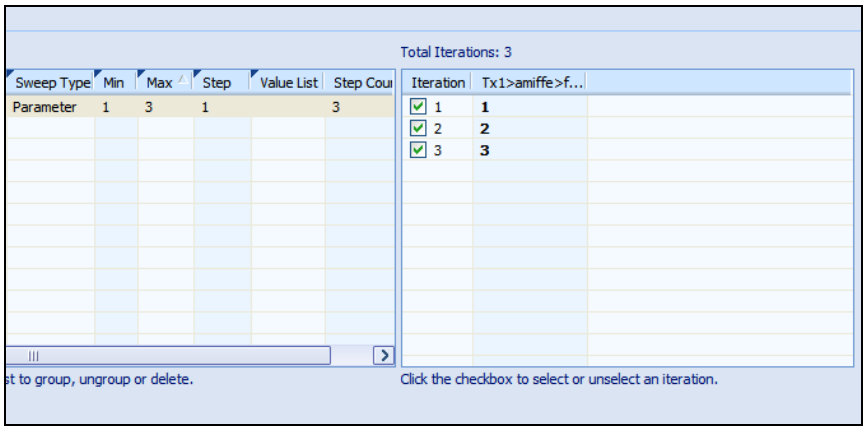
The default **Sweep Type** is dependent on the parameter chosen. In this example, **Parameter** is the default.




### 4.2.2.4 Total Iterations

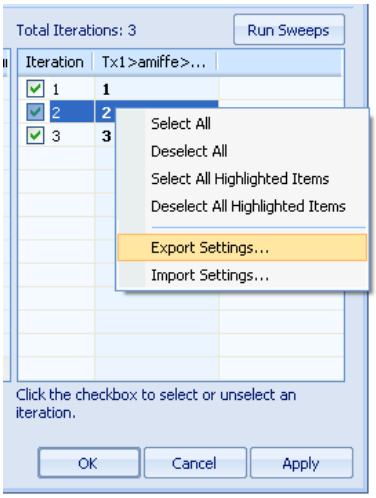
**Total Iterations** are the total number of simulations to be preformed.

In this example, 3 Iterations will be performed.




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

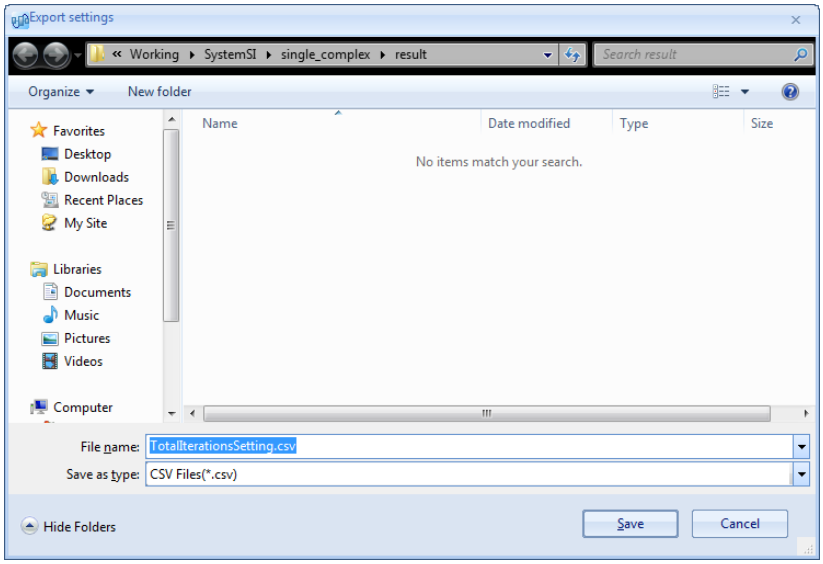
Two options **Export Settings...** and **Import Settings...** are available in the pop-up menu list.




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

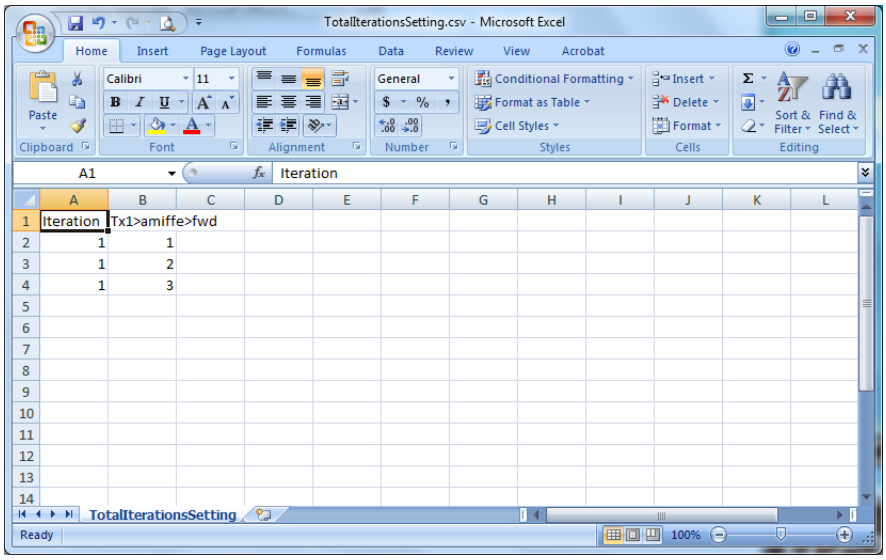
2. Click **Export Settings...**.

The **Export settings** window opens.



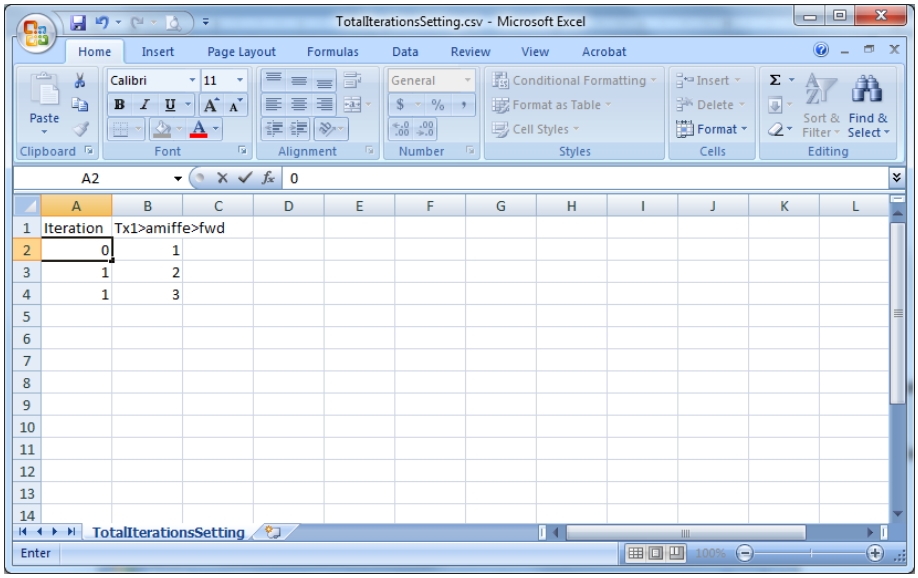
3. Click **Save** to save a csv file listing all the possible iterations.

4. Open the saved csv file.

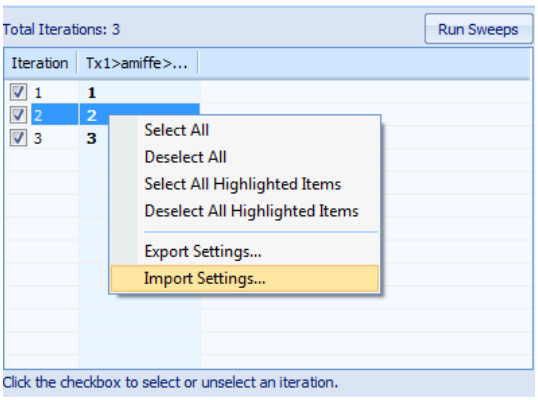



- The number **1** in the **Iteration** column indicates this iteration is enabled in the **Total Iterations** spreadsheet
- The number **0** in the **Iteration** column indicates this iteration is disabled in the **Total Iterations** spreadsheet

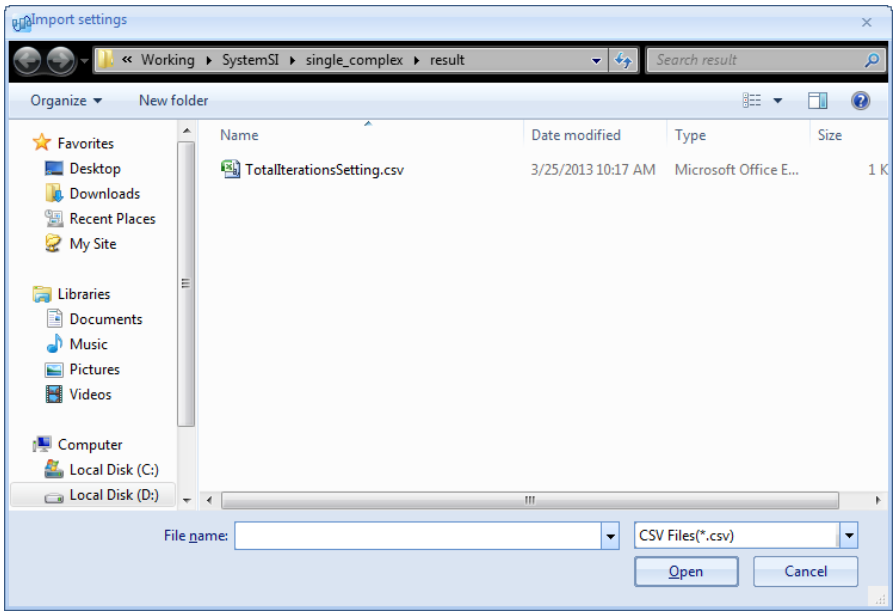
5. Change the **Iteration** value of the first row to **0** as follows, and save the file.




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

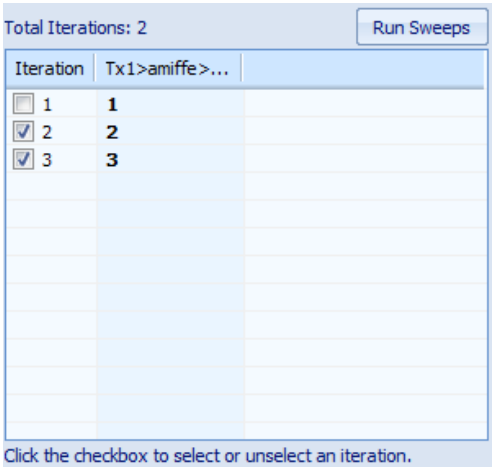



The **Import settings** window opens.




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

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

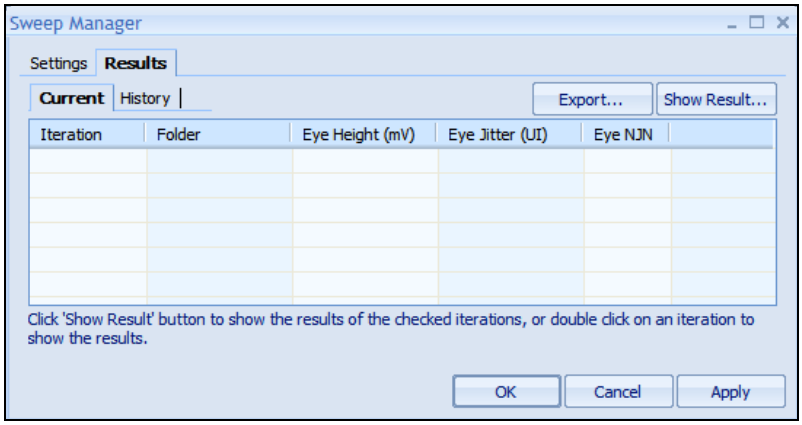



The first row of **Iteration** is disabled.

| NOTE! | The .csv file to be imported must match the content of the **Total Iterations** spreadsheet; otherwise an error message will display to show its failure. |
|---|---|

#### 4.2.2.5 Results

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




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

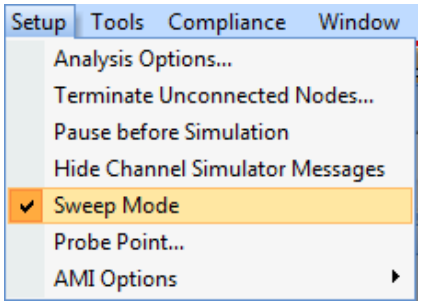
.

### 4.2.3 Running a Single Sweep

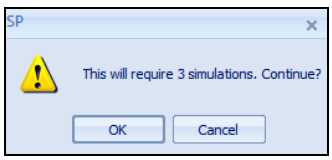
Set up the Simulation Sweep Options and run the sweep using the example in ***Section 4.3.1 Launch Single Channel Complex Template***.

#### 4.2.3.1 Analysis Options

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




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

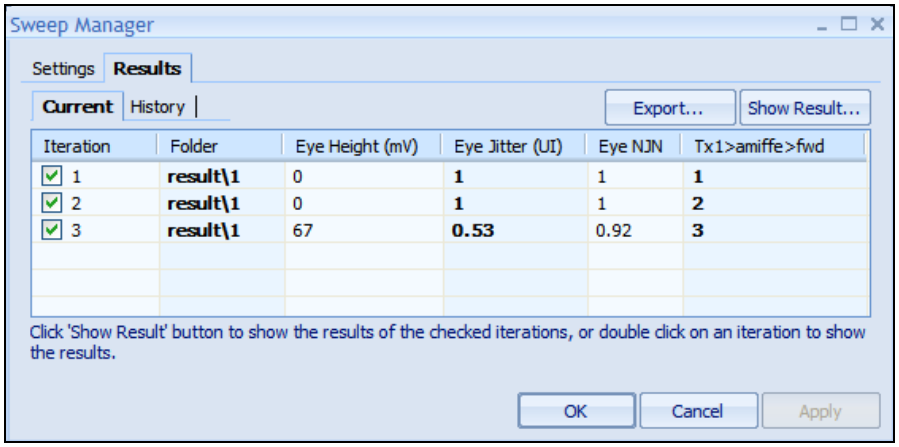



#### 4.2.3.2 View the Results

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

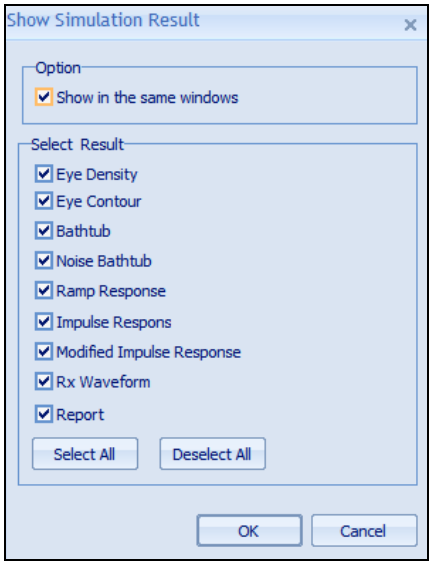
- Measured **Eye Height (mV)**

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

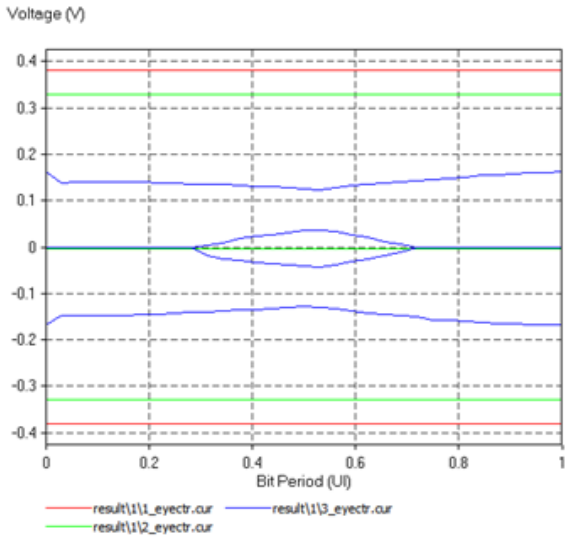



Two settings for the **fwd** parameter do not produce any eye opening.

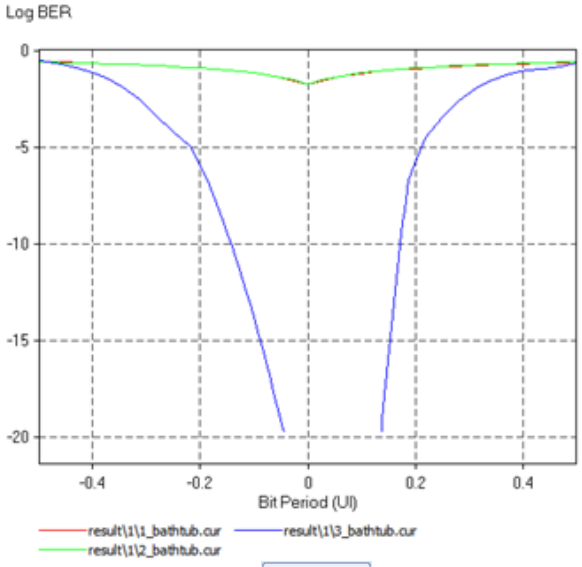
1. Select one or more iterations.
2. Click Show Result... to view the curves. The Show Simulation Result dialog appears.

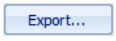



3. Click **OK**. The **Eye Contour** results appear.

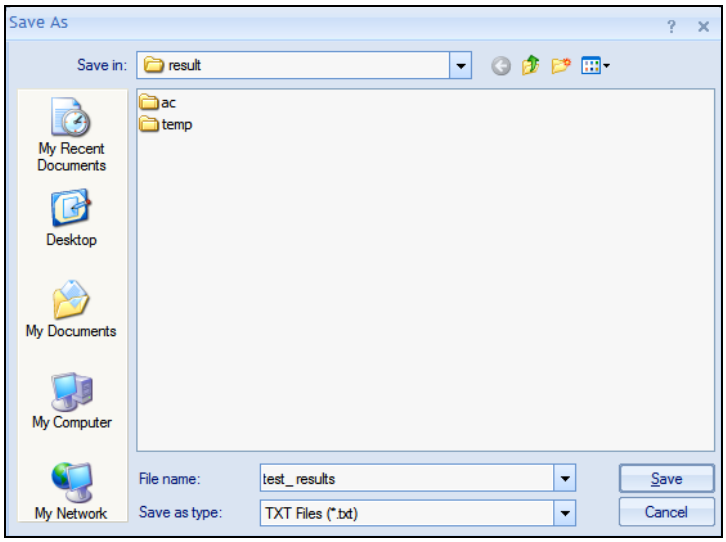



The **Bathtub** cureves for all the iterations appear.




4. Click the **Export** button  in the **Sweep Manager** interface to export the results.

The **Save As** dialog opens.




### 4.2.4 Run Multiple Sweeps

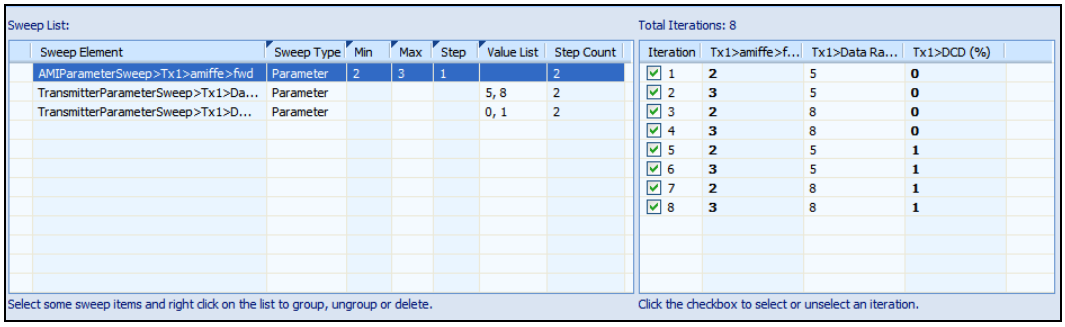
This section describes how to sweep multiple parameters and across categories to show the sweep capability.The parameters and the values are to be set:

**fwd** of amiffe: 2, 3

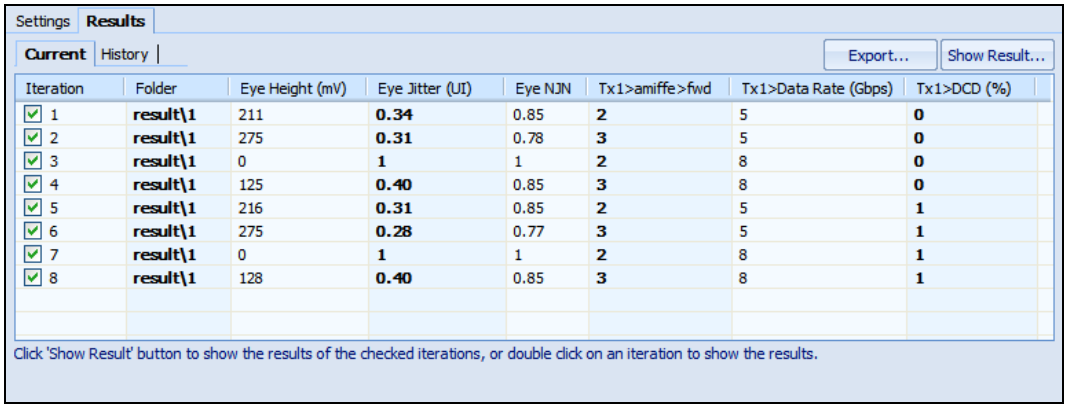
**Data Rate** of Tx1 (Gbps): 5, 8

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

There are a total of 8 combinations.




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.



## 4.2.5 Other Sweep Features

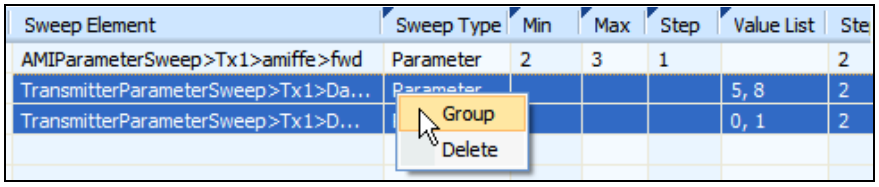
The **Sweep Manager** can group parameters and help select Sweep Models.

### 4.2.5.1 Grouping Parameters

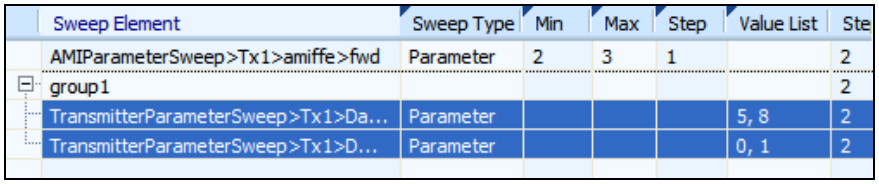
You can group two or more parameters to reduce the number of iterations.

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**.




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




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**:

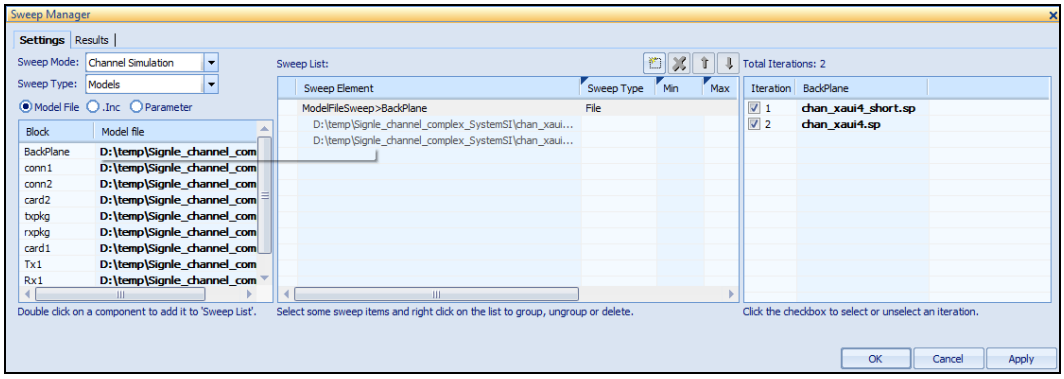
- **.Inc** sweep – Sweep multiple circuit models if the original model is called within a `.include` statement (such as **conn1** and **conn2** blocks).
- **Model File** sweepSweep different parameters defined in the `.sp` files (such as **c_comp** for the **Tx1** block).

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]`.

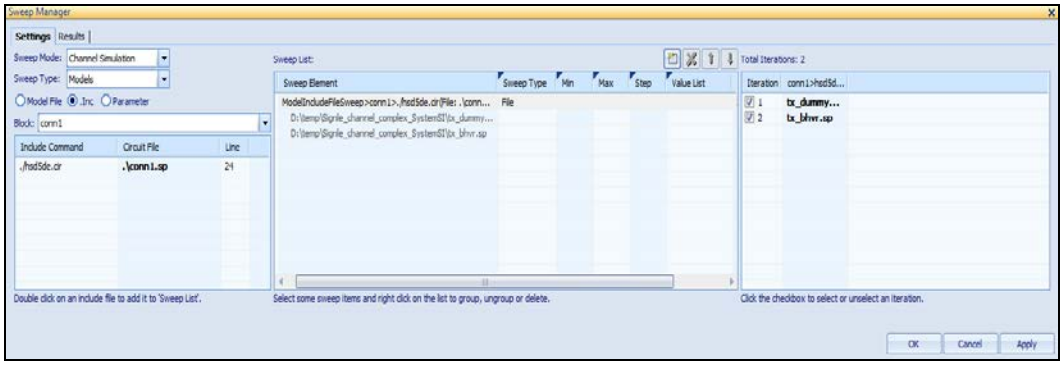
Click the **New** button  to add model files.



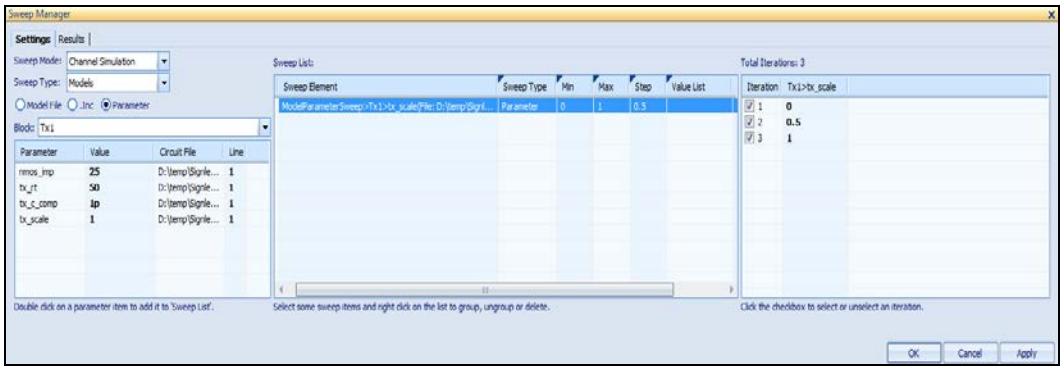

**.Inc Sweep**

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.




**Parameter Sweep**

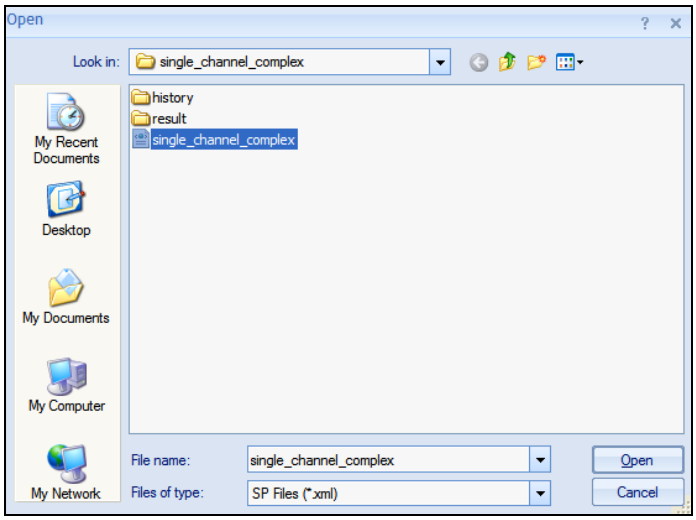



## 4.3 S-Parameter Extraction

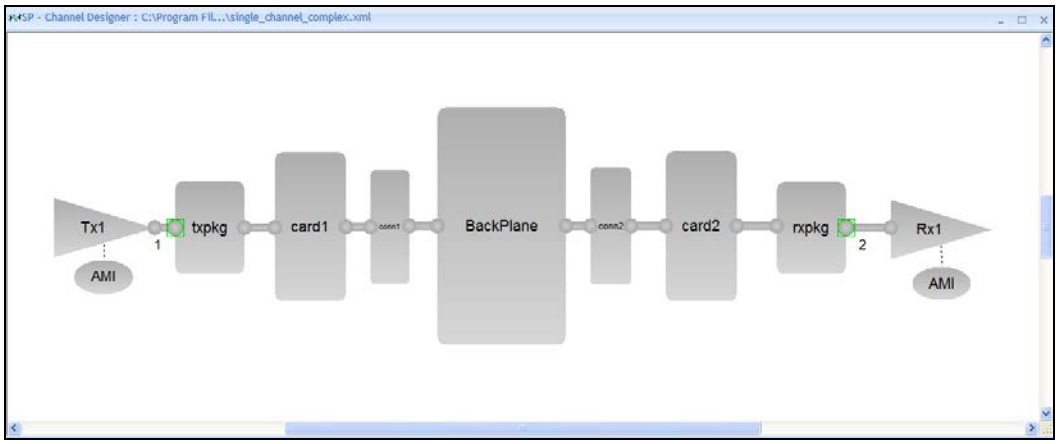
### 4.3.1 Launch Single Channel Complex Template

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

Click **Open** in the **File** menu to open the **`single_channel_complex`** Template.




The **`single_channel_complex`** template workplace is shown below.




The **Single Channel Complex** template contains the following blocks:

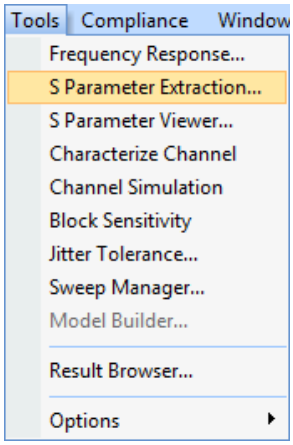
- A Primary Transmitter (**Tx1**)
- A Primary Receiver (**Rx1**)
- Two Packages (**txpkg** and **rxpkg**)
- Two Add-in Cards (**card1** and **card2**)
- Two Connectors (**conn1** and **conn2**)
- A Backplane (**BackPlane**)
- Two **AMI** Models

## 4.3.2 S-Parameter Extraction

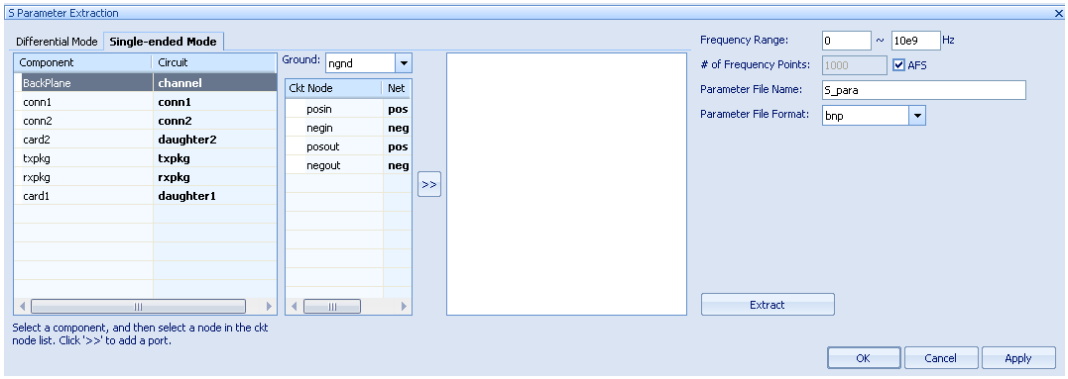
SystemSI - Serial Link Analysis supports **S-Parameter Extraction** for both single-ended nets and differential nets.

Select

Tools > S Parameter Extraction….

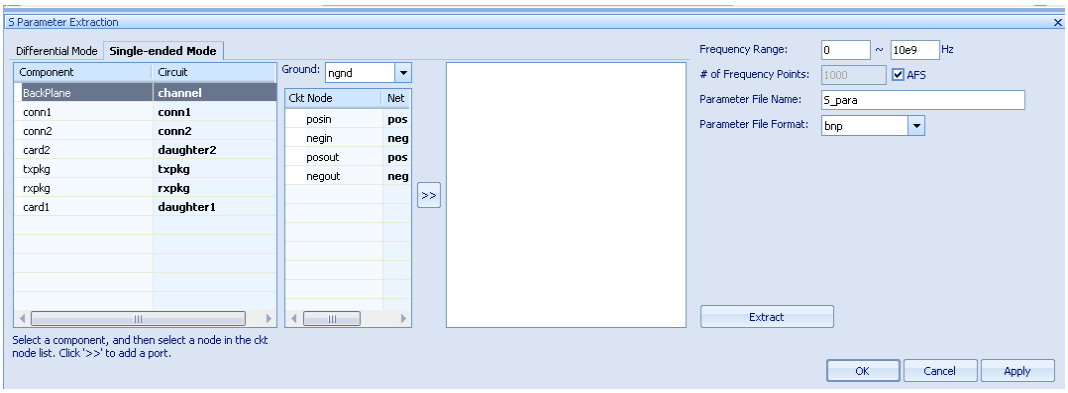



The **S Parameter Extraction** window opens.

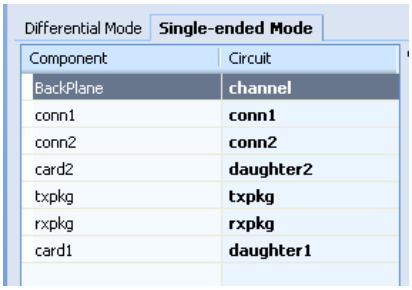



### 4.3.2.1 Single-ended Mode

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

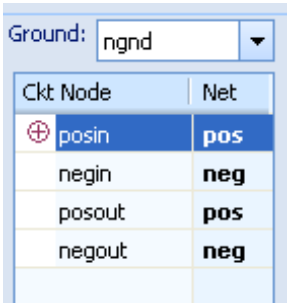



2. Select the **BackPlane** component to extract S Parameter.

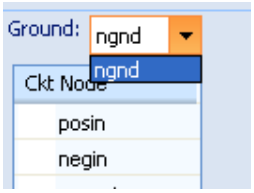



3. Set up ports.

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

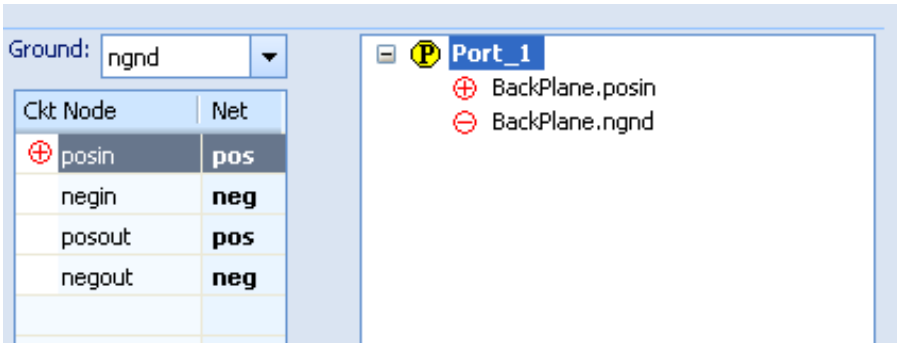



3.2 Choose **ngnd** as Ground node.



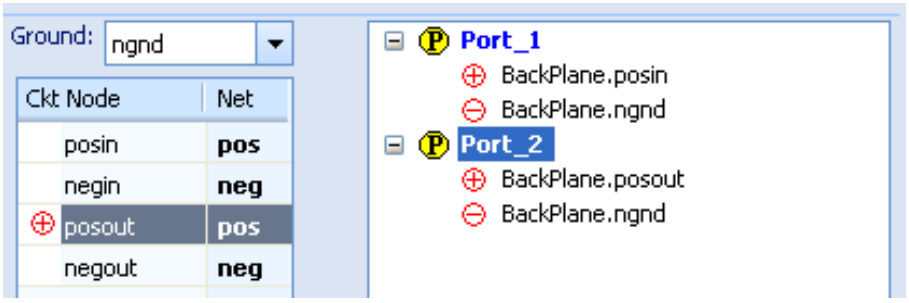

3.3 Click the  button.

Settings for **Port_1** are completed as the following window shows.



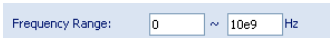

3.4 Repeat Step 3.1 to 3.3 to define the circuit node **posout**.

Settings for **Port_2** are completed as the following window shows.




4. Set up Parameters.

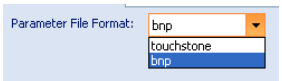
4.1 Define **Frequency Range**.




4.2 Define **No. of Frequency Points**.

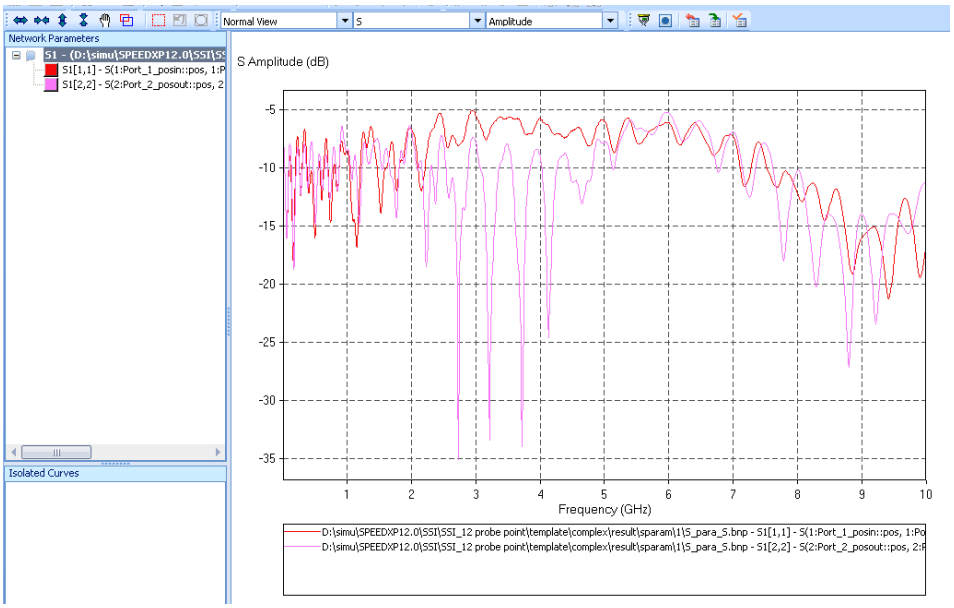

4.3 Input **Parameter File Name**. For example: S_para.


4.4 Select **Parameter File Format**: bnp.




5. Click the **Extract** button.

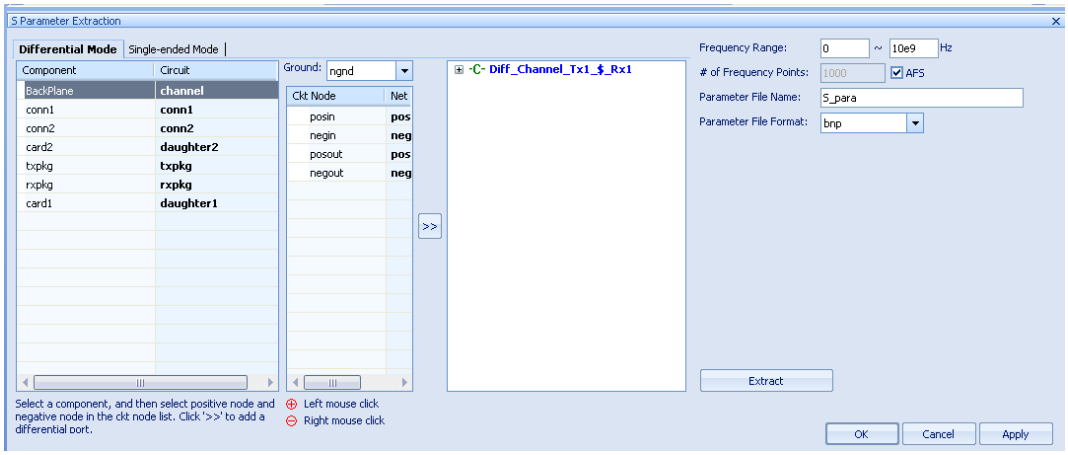
S Parameter curves for **Port_1** and **Port_2** show as the following figure.



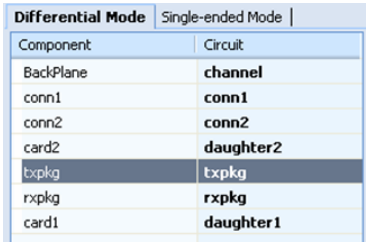

S Parameter document is automatically generated in the folder of <working folder path>\result\sparam\.

### 4.3.2.2 Differential Mode

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

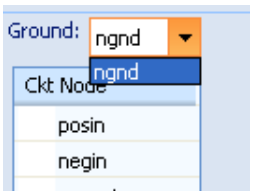



2. Select the **txpkg** component to extract S Parameter.



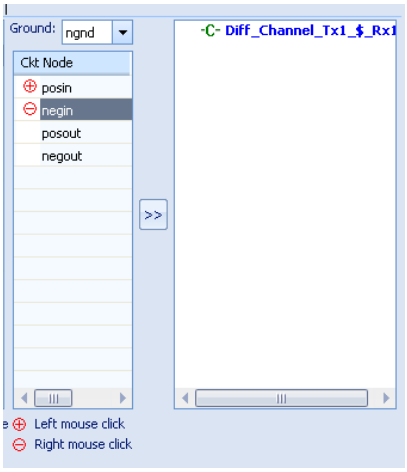

3. Set up ports.

   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.

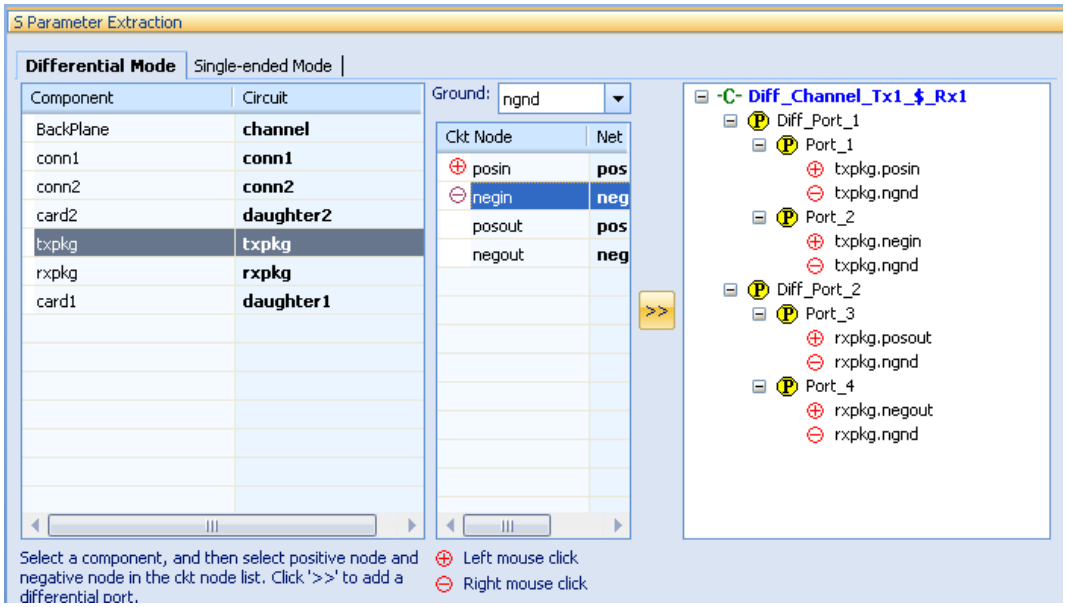



3.4 Click the  button.

The port **Diff_Port_1** is generated automatically.

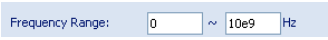
3.5 Repeat Step 3.1 to 3.3 to define the **rxpkg** component (define **posout** as positive node and **negout** as negative node).

The port **Diff_Port_2** will be generated automatically.




4. Set up parameters.

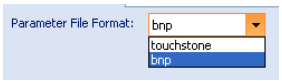
4.1 Define **Frequency Range**.



4.2 Define **No. of Frequency Points**.

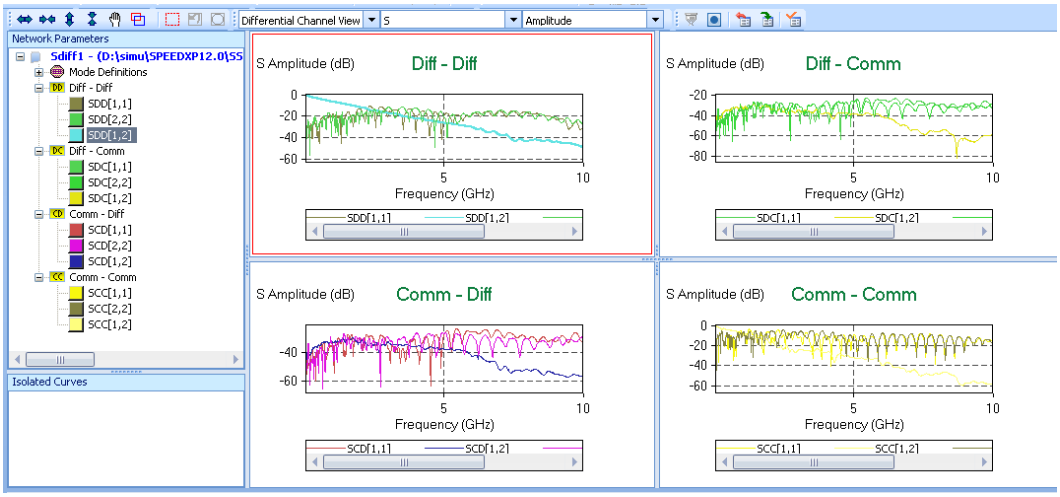

4.3 Input **Parameter File Name**. For example: S_para.


4.4 Select **Parameter File Format**: bnp.




5. Click the **Extract** button.

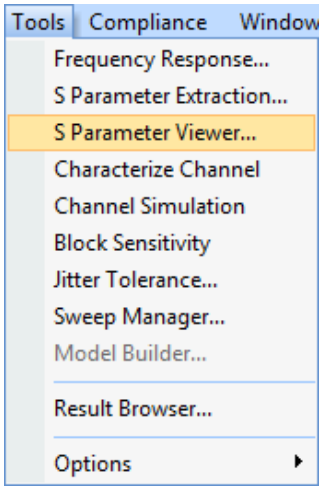
S Parameter curves show as the following figure.



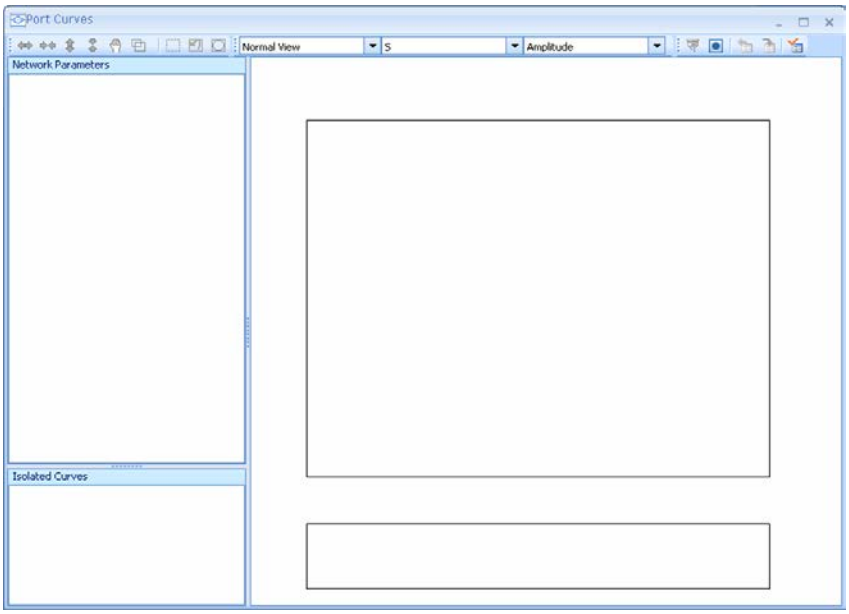

## 4.3.3 S-Parameter View

1. Select

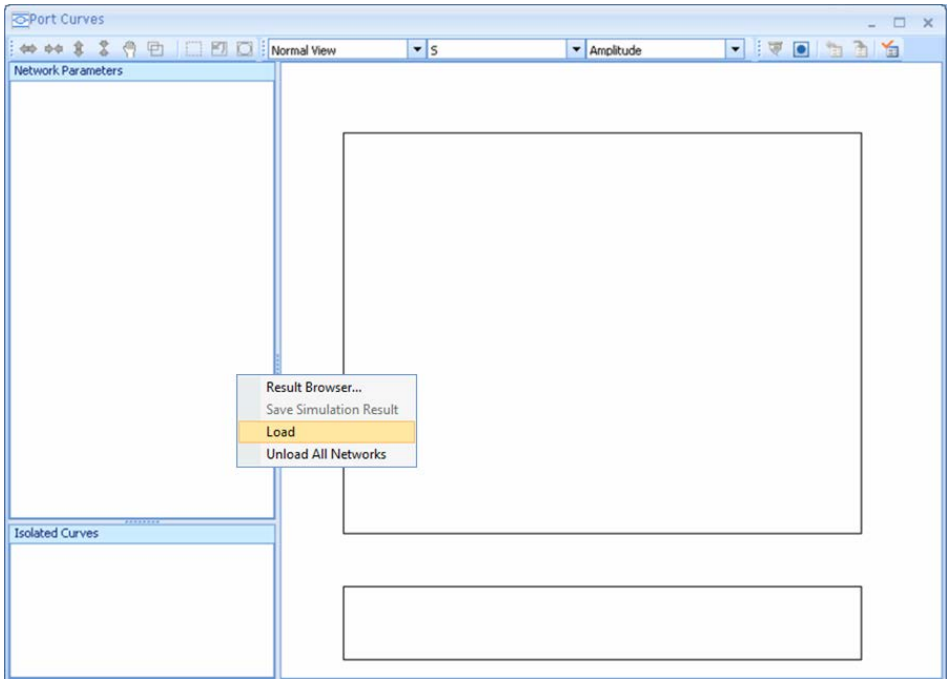
Tools > S Parameter Viewer.



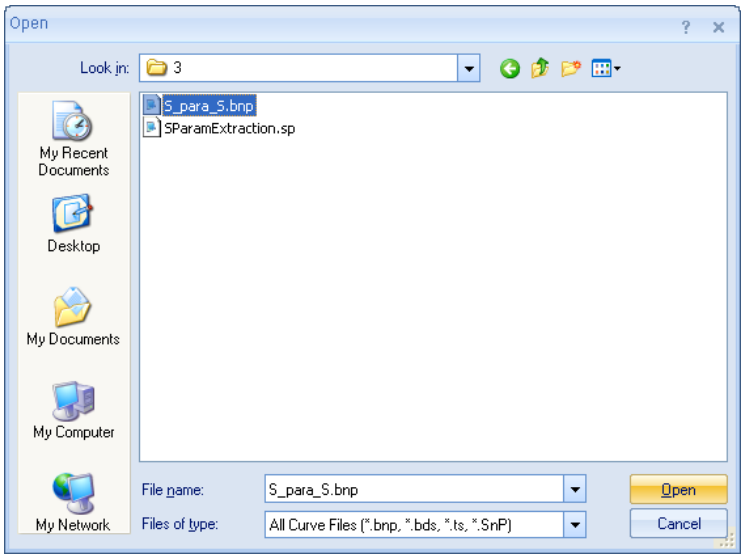

The **Port Curves** window opens.




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

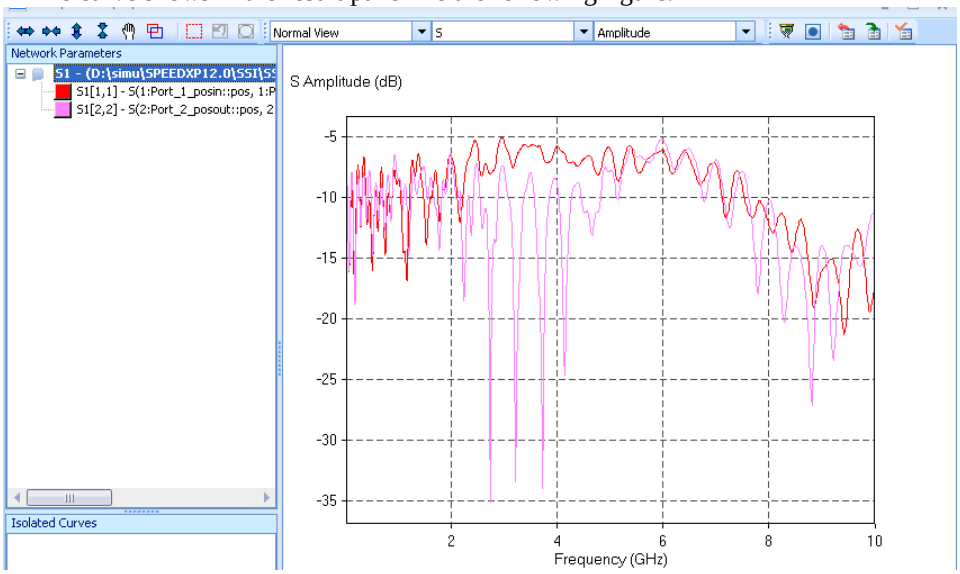



The **Open** window opens.




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.

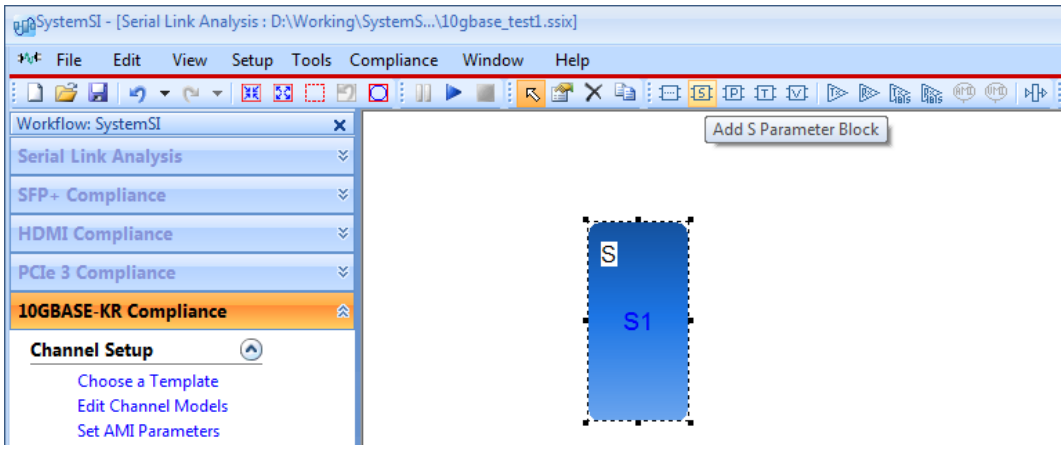



## 4.4 S-Parameter Wrapping and BBS Integration

### 4.4.1 Add S Parameter Block

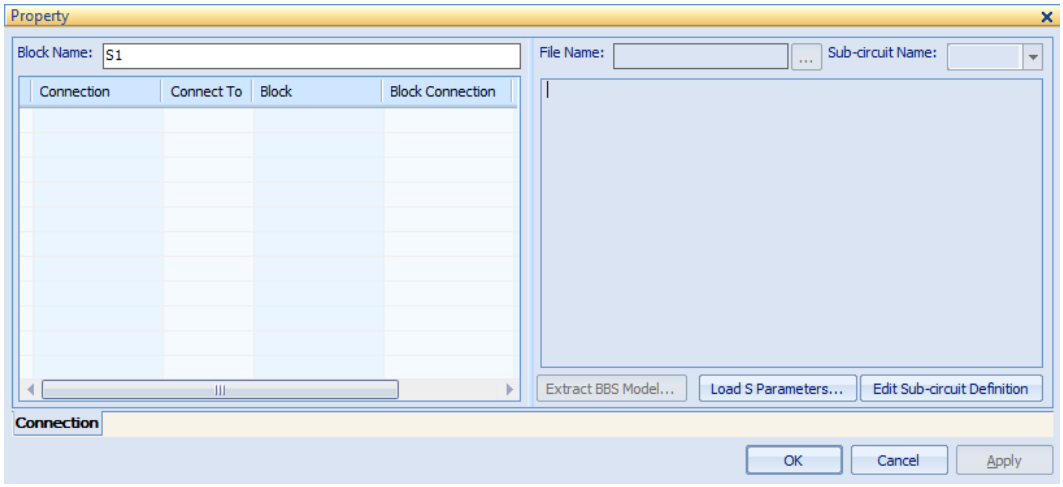
1. Click the **Add S Parameter Block** icon on the tool bar, and click in the **Layout** window.

   The S Parameter block **S1** is added.




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

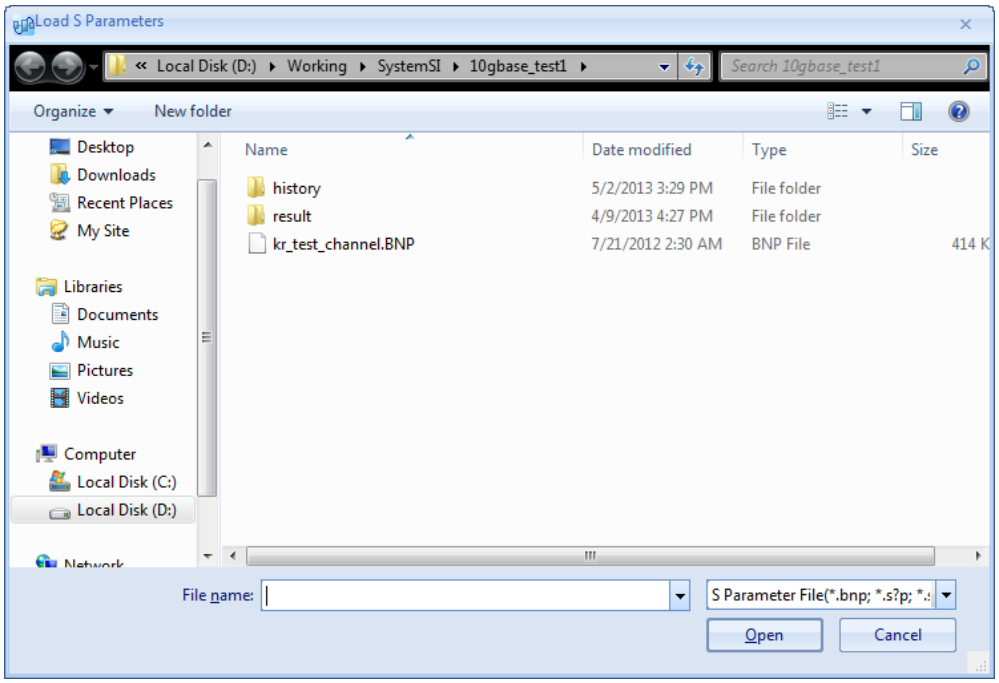
   The **Property** pane opens.




### 4.4.2 Load S Parameter File

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

   The **Load S Parameters** window opens.

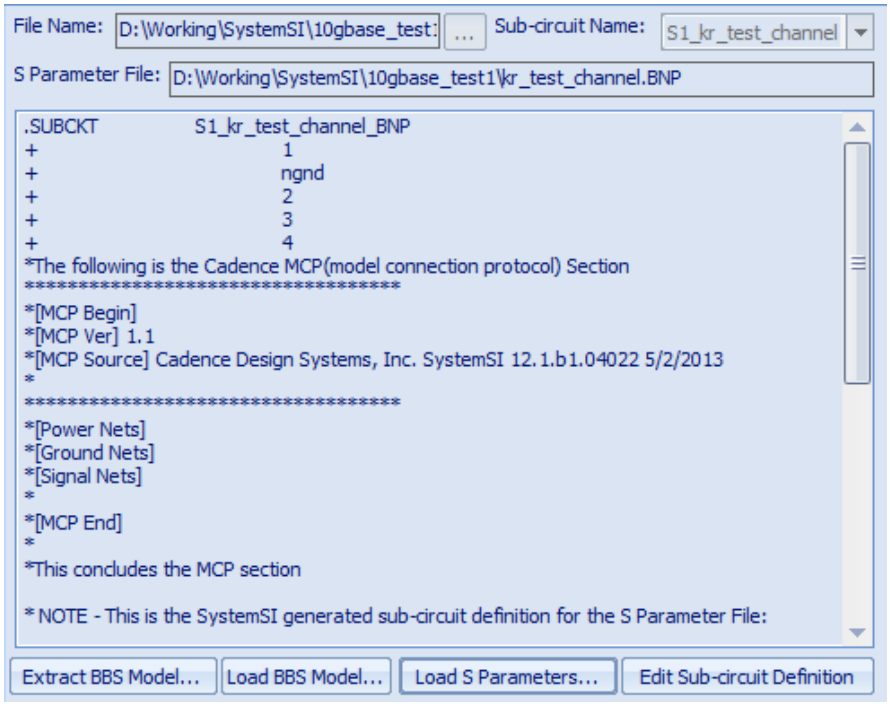



2. Load the S Parameter file (BNP or Touchstone) to the S Parameter block.

   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




   - 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

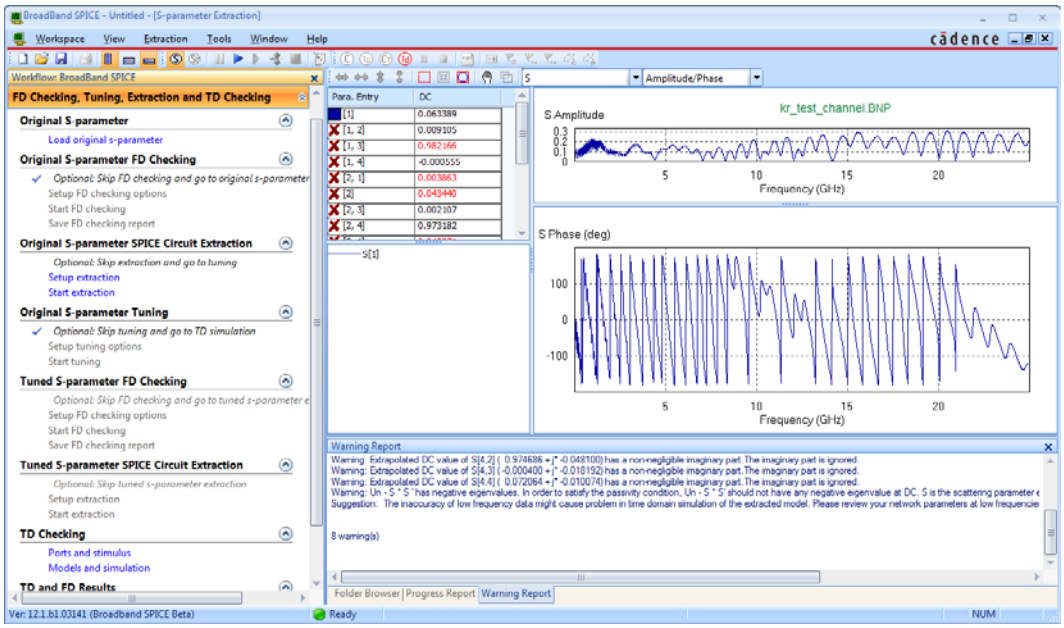
- For the Touchstone file, or the BNP file which does not have the MCP information, you are expected to manually add the MCP through the MCP Editor

| 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.

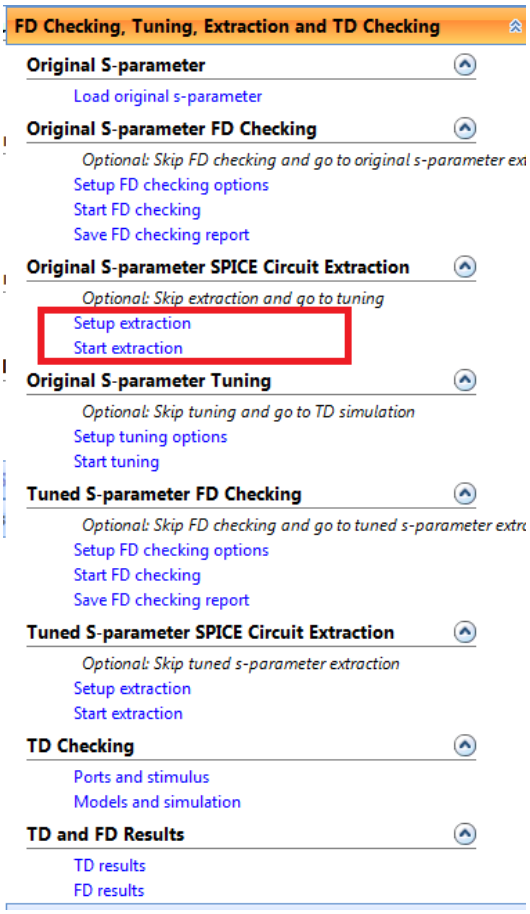
   The BroadbandSPICE application is launched.



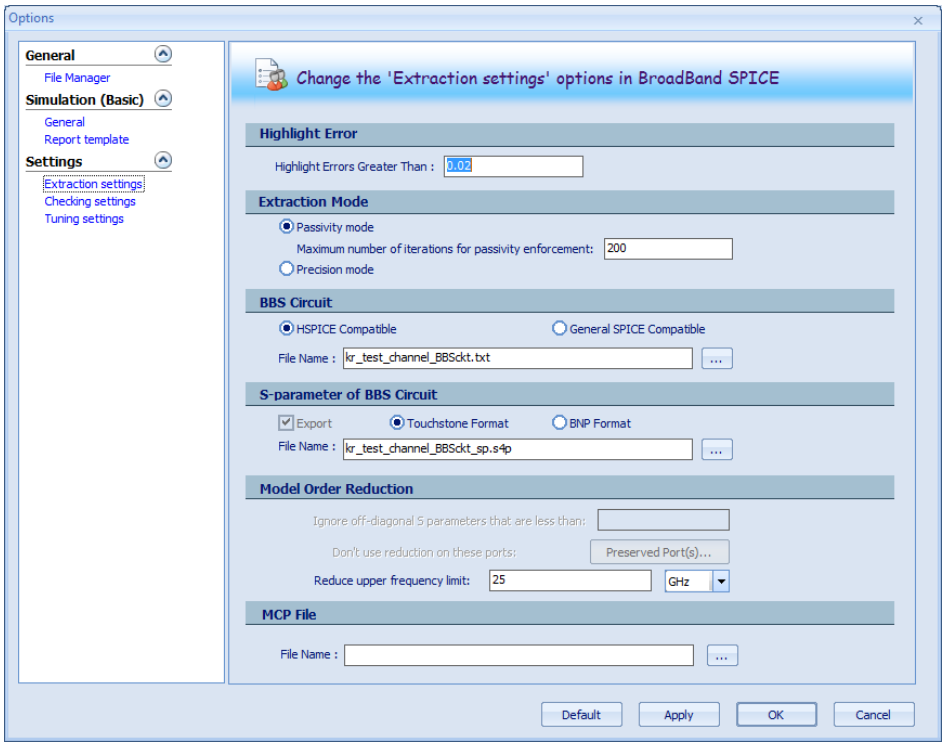

2. Click the **FD Checking, Tuning, Extraction and TD Checking** workflow to check and tune the S Parameters.

   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.

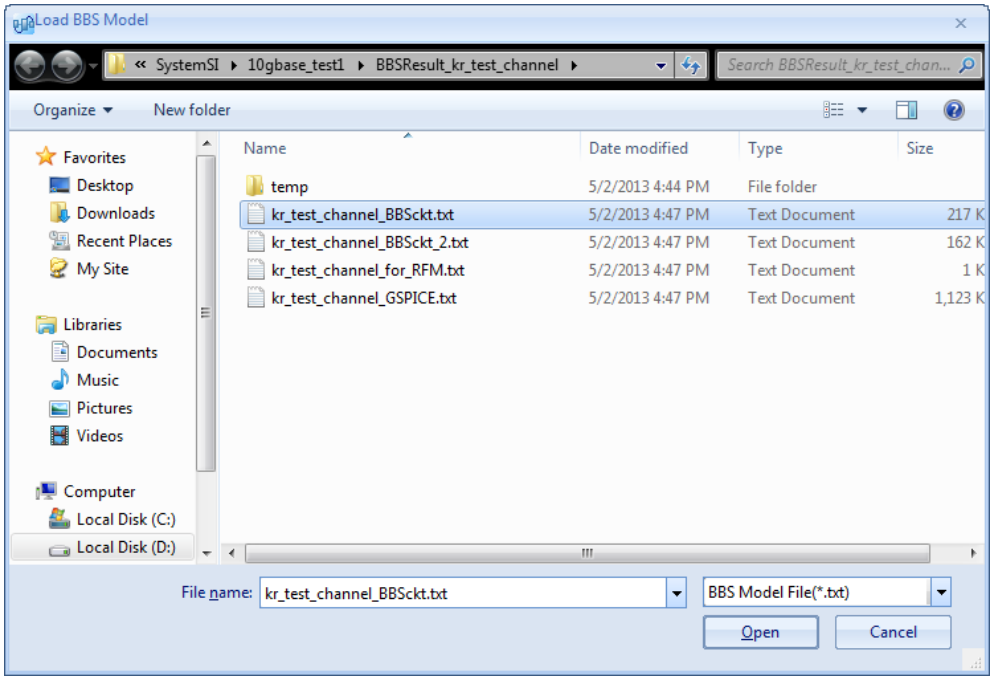



The **Options** window opens.



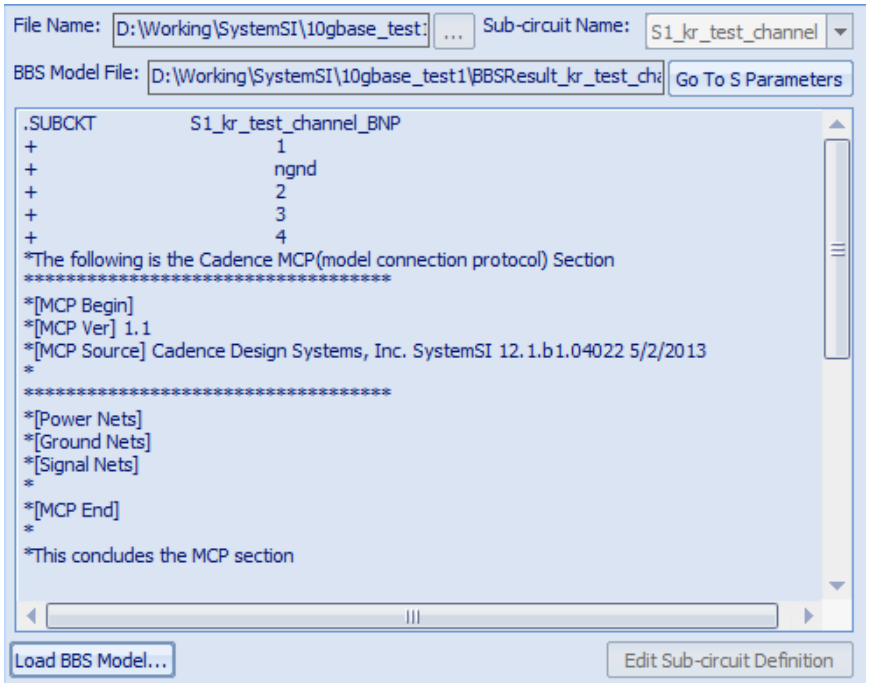
4. Set up the extraction settings as desired.
5. Click **Start extraction** in the workflow to extract the BBS model.

   If the BBS model extraction is successful, a .txt file will be automatically generated.
6. Click the **Load BBS Model…** button to load the generated BBS model **kr_test_channel_BBSckt.txt**.




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



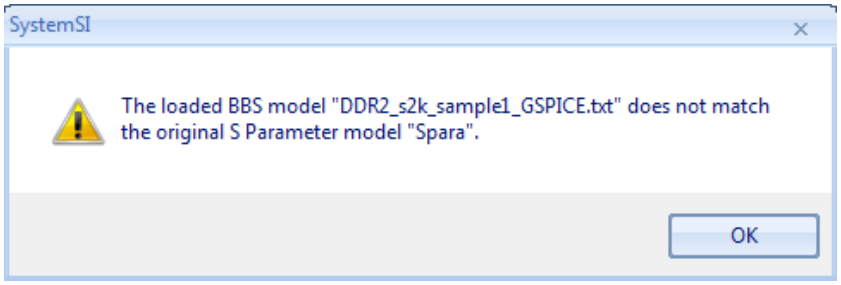
| NOTE! | Editing of .sp file for the BBS model is NOT recommended. |
|---|---|

### 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.

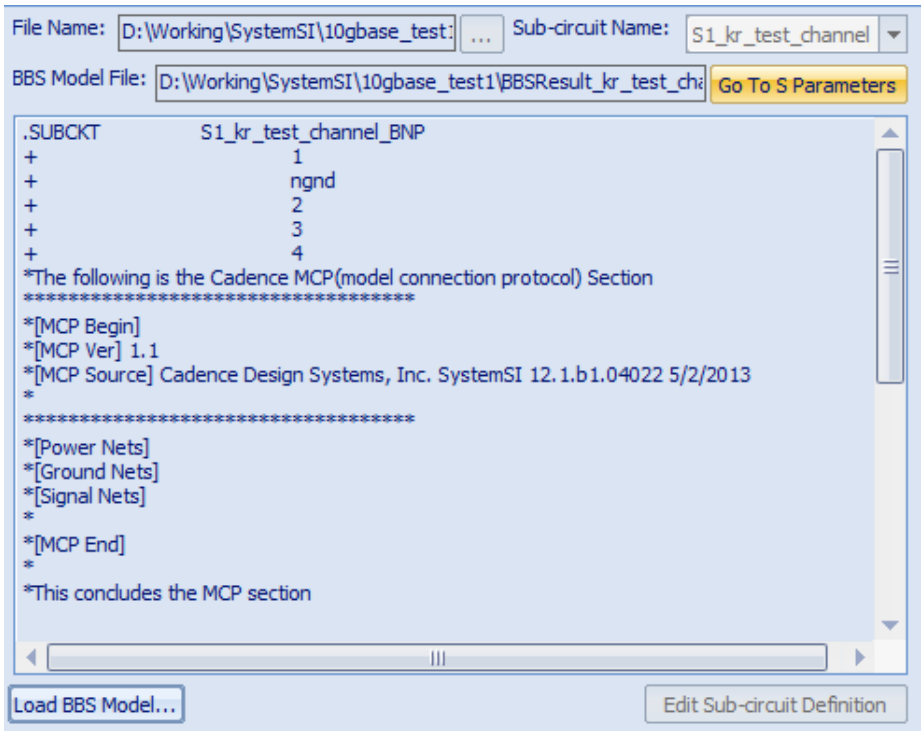
Please refer to **Step 6** in ***Section 4.4.3 Extract the BBS Model*** for details.

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

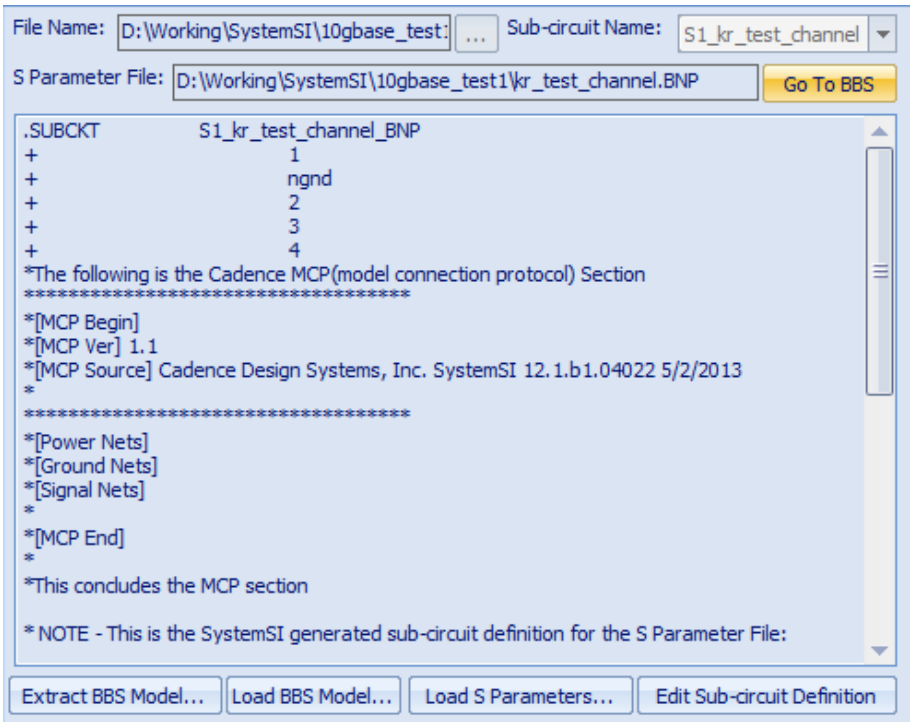



### 4.4.5 Switch the Models

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




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




| | |
|---|---|
| **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

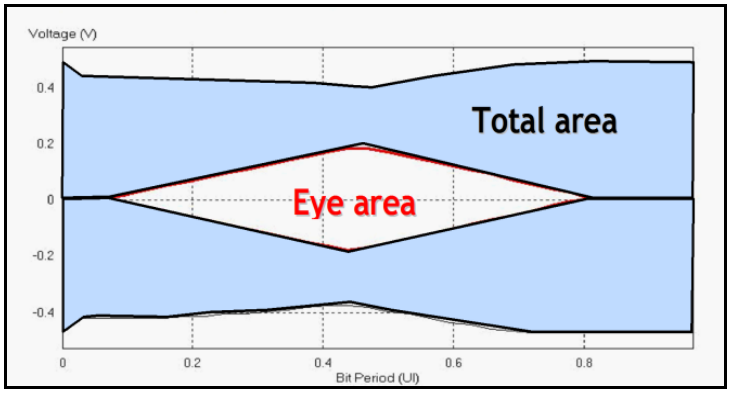
The **Block Sensitivity** feature uses a novel eye-area-based normalized Jitter and Noise (NJN) metric. The goal is to find out the effect of each system component on Jitter and Noise.

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.

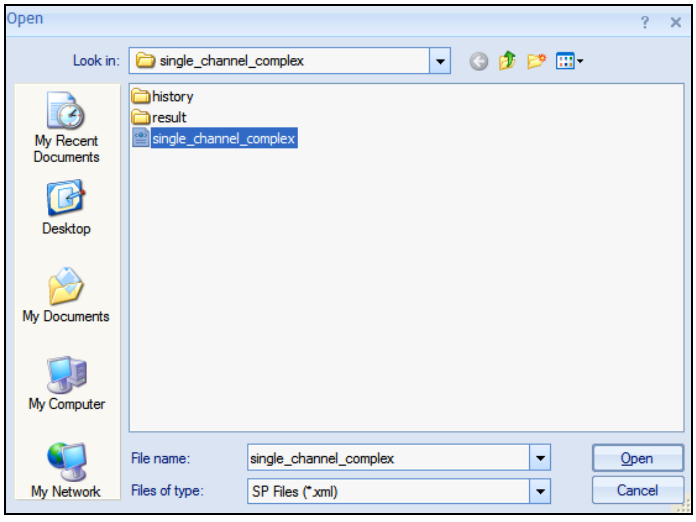
A non-zero eye opening is required to run the **Block Sensitivity** feature at any data rate and get meaningful results.

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

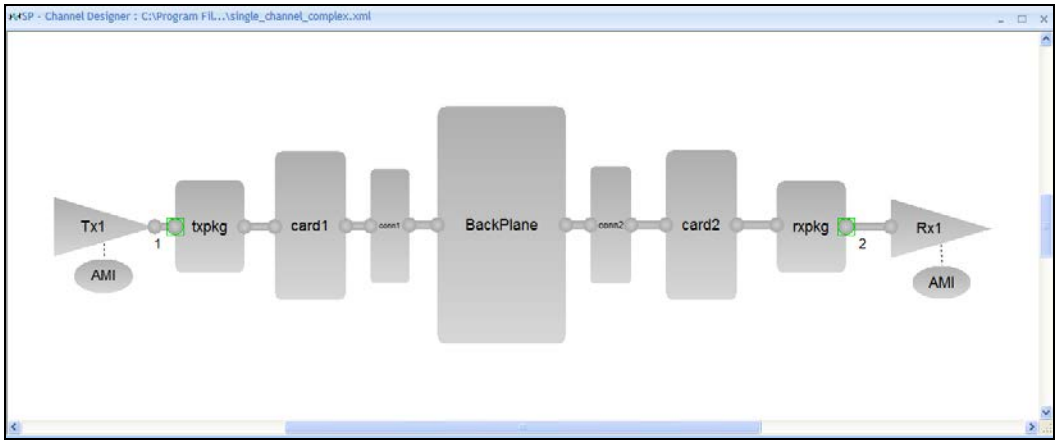
### 4.5.2 Launch Single Channel Complex Template

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

Click **Open** in the **File** menu to open the **`single_channel_complex`** Template.




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.

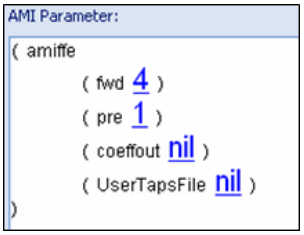
We are using the results of the single sweep in the *Sweep Manager* section.

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

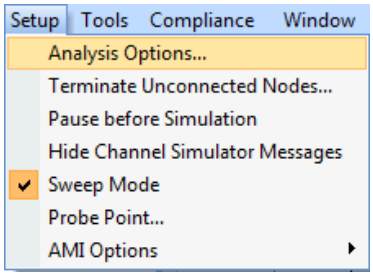
This setting is necessary to get any eye opening.

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

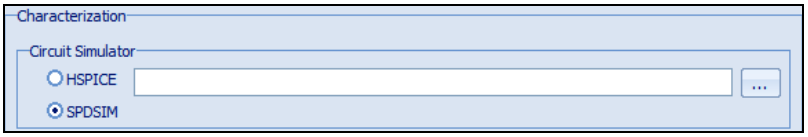

2. Change the default **fwd** parameter to 4.



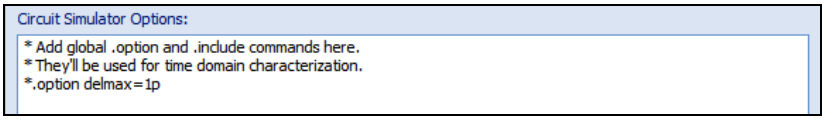
3. In the **Setup** menu, click **Analysis Options…**.




Circuit Simulator




Circuit Simulator Options




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

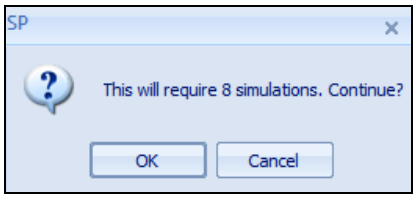
### 4.5.4 Start Block Sensitivity

In the **Tools** menu, click

Block Sensitivity.

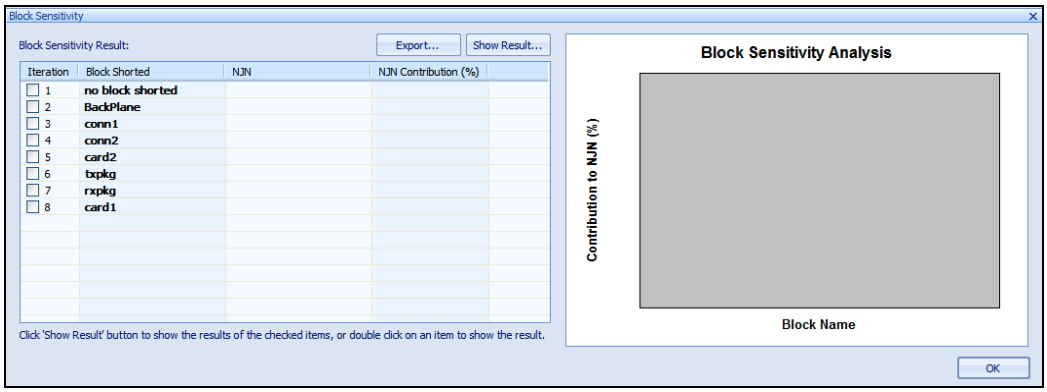
Once you start **Block Sensitivity**, the tool immediately prompts you to confirm the number of simulations to be run.

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



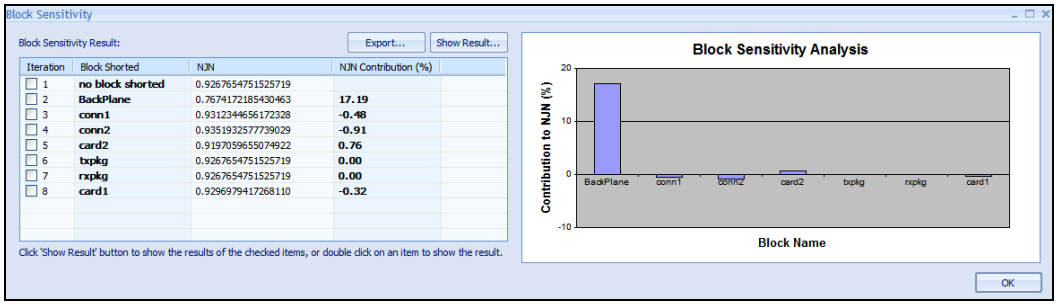

A net name must be associated with each node in the channel. If not, the Block Sensitivity will not work. You can assign Net names from the **Block Property** panes or from the **MCP Editor**.

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




## 4.5.5 View and Analyze the Results

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




### 4.5.5.1 Block Sensitivity Result Pane

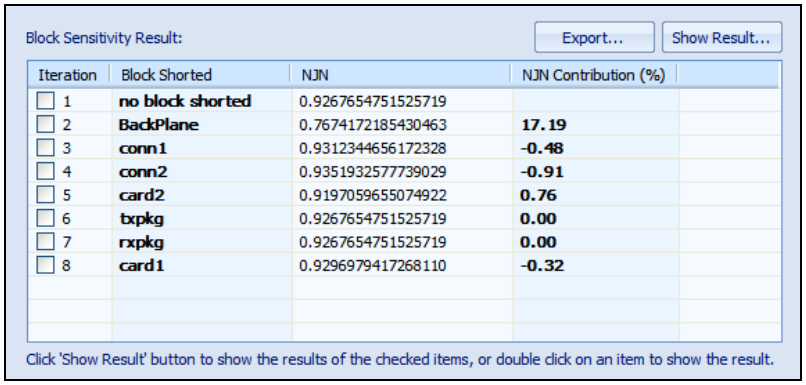
The **Block Sensitivity Result** pane has four columns:

- **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).

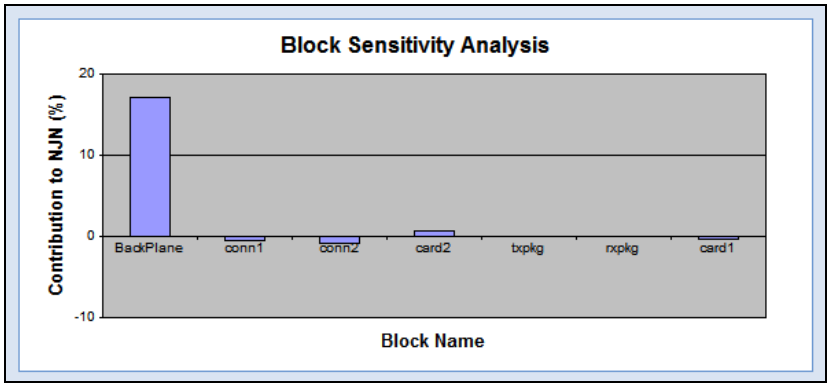
- **NJN Contribution** - Ratio of the NJN of the shorted block to the NJN when no blocks are shorted. Obtained from the formula:

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



### 4.5.5.2 Block Sensitivity Analysis

The Block Sensitivity Analysis chart displays NJN Contribution results.



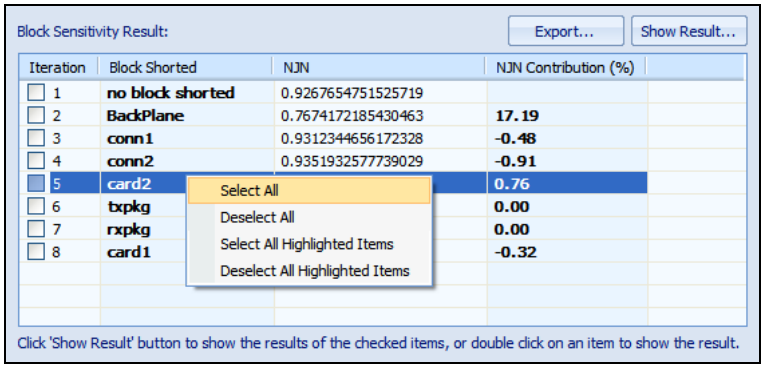

Click on the column header **NJN Contribution (%)** to sort the results in ascending or descending order.

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

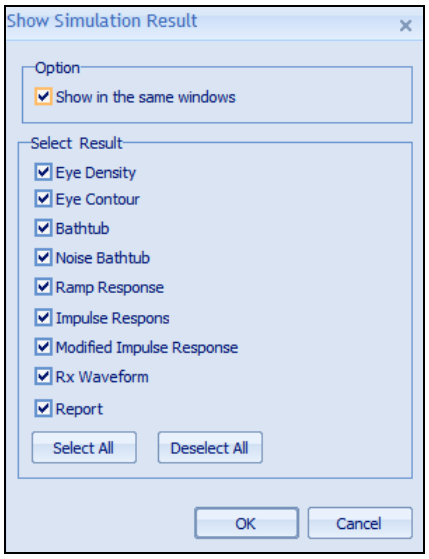
Multiple contributions are negative, which indicates that the eye gets worse when the component is shorted. It is difficult to predict what will happen when components are shorted.
The **NJN Contribution** of **txpkg** and **rxpkg** blocks is 0% because these blocks already have dummy models in the default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

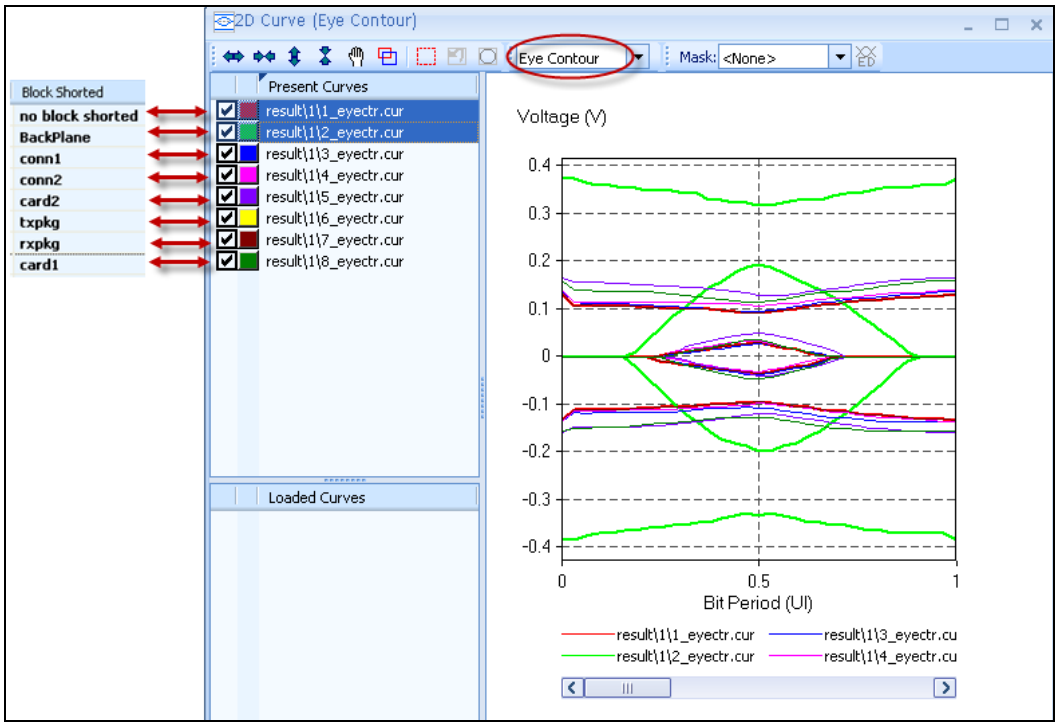



4. The **Show Simulation Result** window opens.

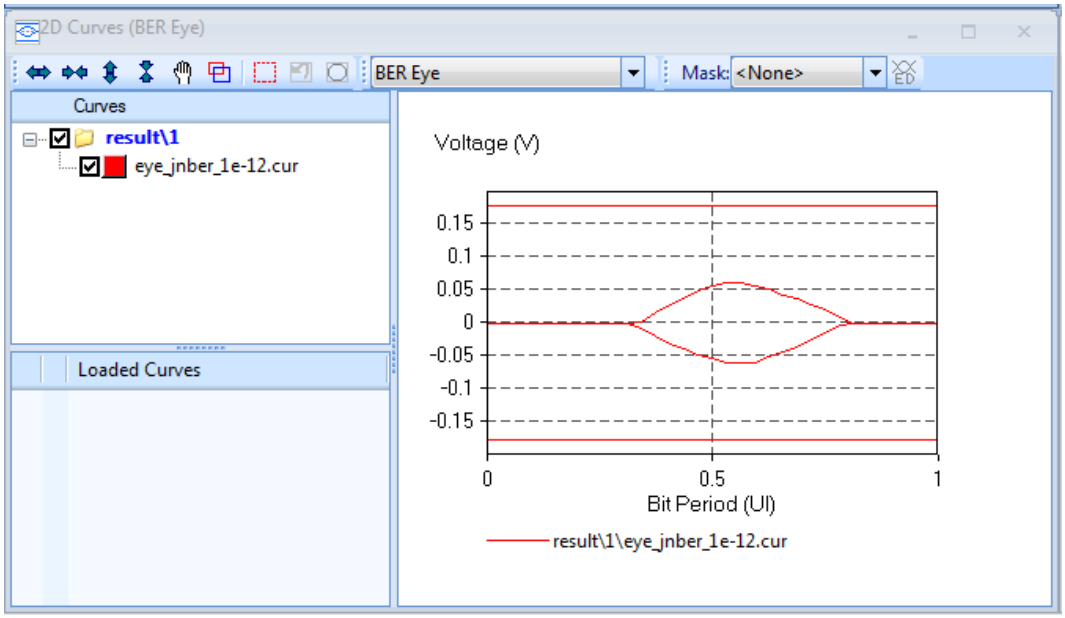



See the following pages for more curves.

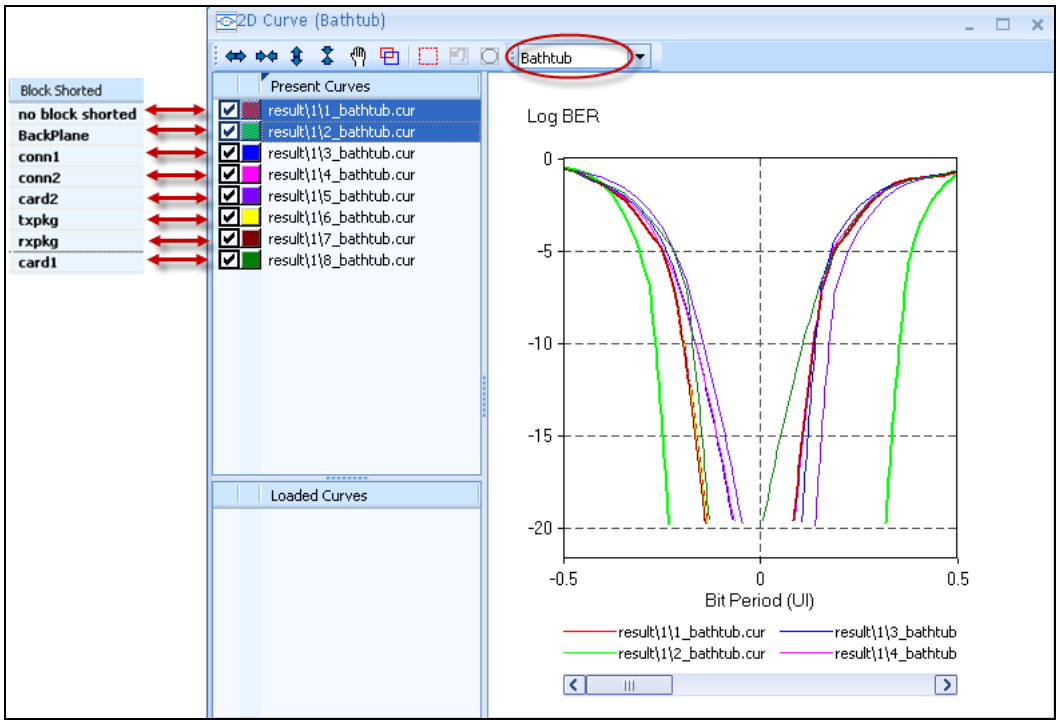
#### 4.5.5.2.2 2D Curve (Eye Contour)



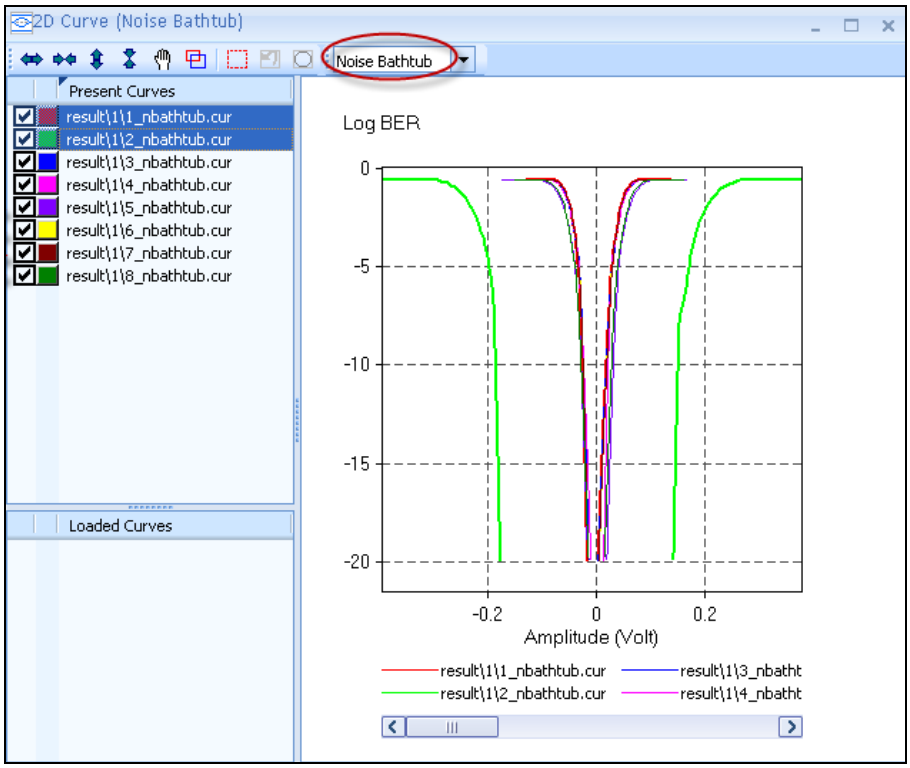

If **Both time and voltage** is checked in the **Analysis Options** window, a new curve for the JN_BER_Eye will be generated and displayed as **BER Eye**.



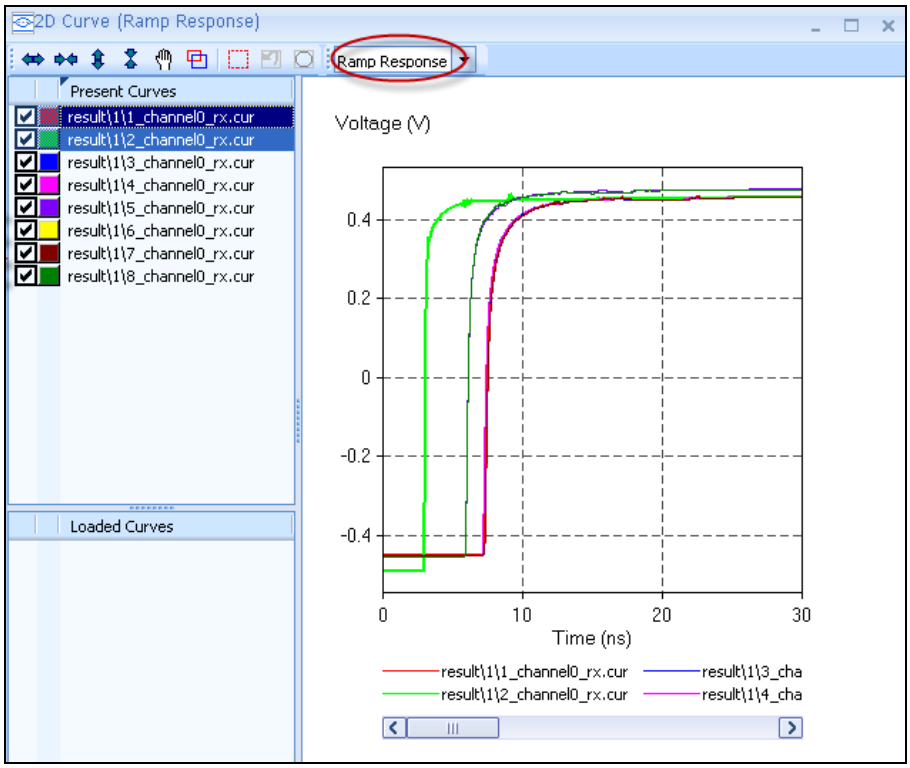

#### 4.5.5.2.3 2D Curve (Bathtub)


2D Curve (Bathtub)
Bathtub
Block Shorted
no block shorted
BackPlane
conn1
conn2
card2
txpkg
rxpkg
card1
Present Curves
result\1\1_bathtub.cur
result\1\2_bathtub.cur
result\1\3_bathtub.cur
result\1\4_bathtub.cur
result\1\5_bathtub.cur
result\1\6_bathtub.cur
result\1\7_bathtub.cur
result\1\8_bathtub.cur
Loaded Curves
Log BER
0
-5
-10
-15
-20
-0.5
0
0.5
Bit Period (UI)
result\1\1_bathtub.cur
result\1\3_bathtub
result\1\2_bathtub.cur
result\1\4_bathtub

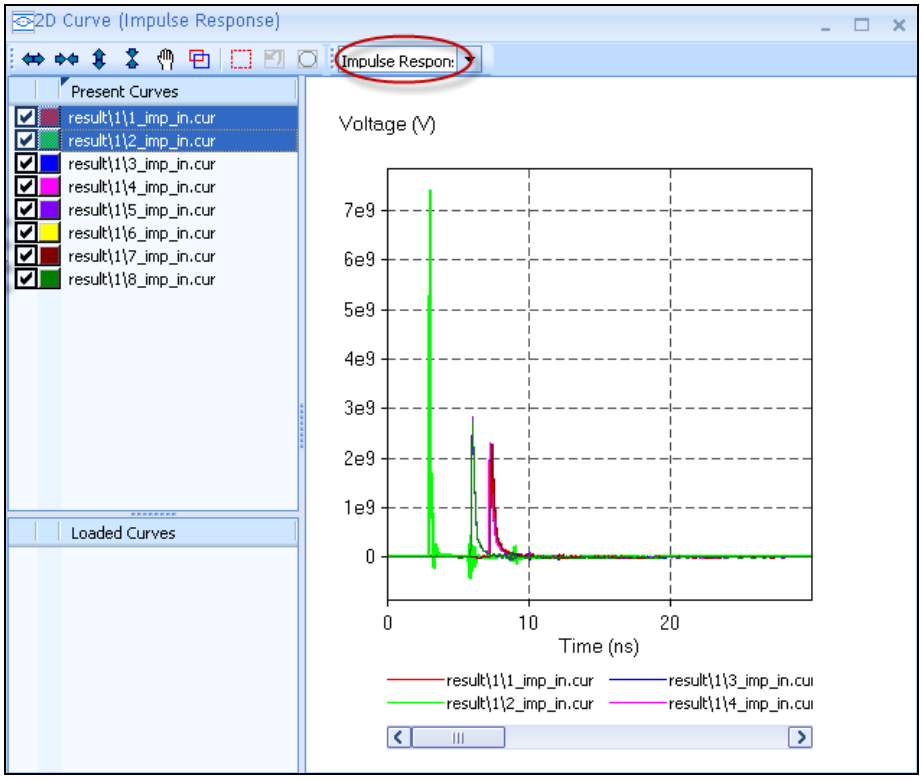
#### 4.5.5.2.4 2D Curve (Noise Bathtub)



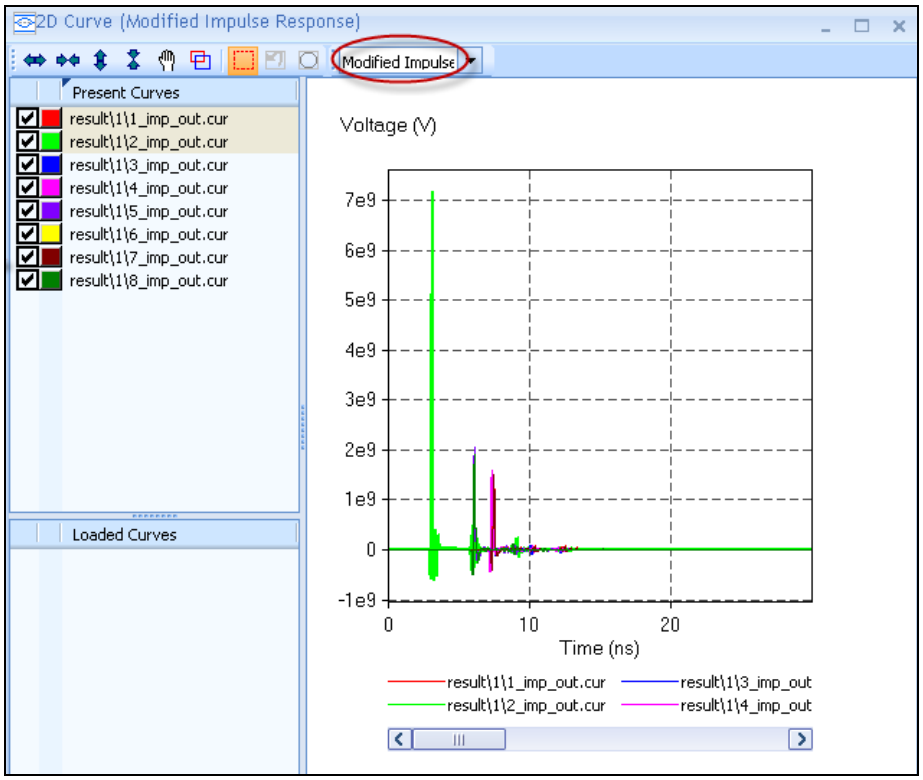

#### 4.5.5.2.5 2D Curve (Ramp Response)



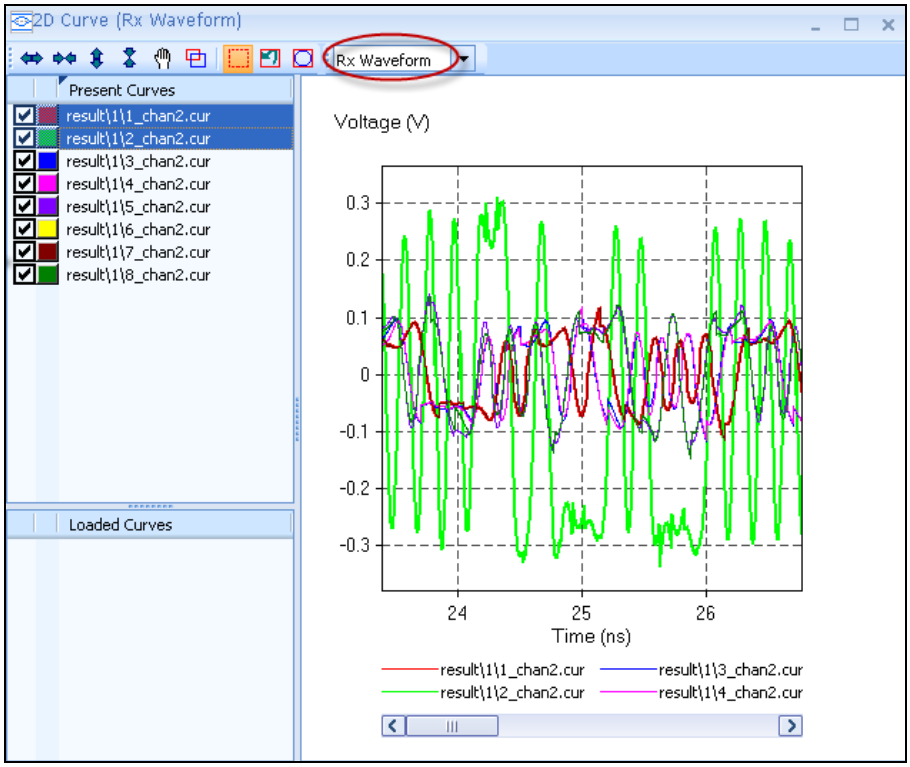
#### 4.5.5.2.6 2D Curve (Impulse Response)



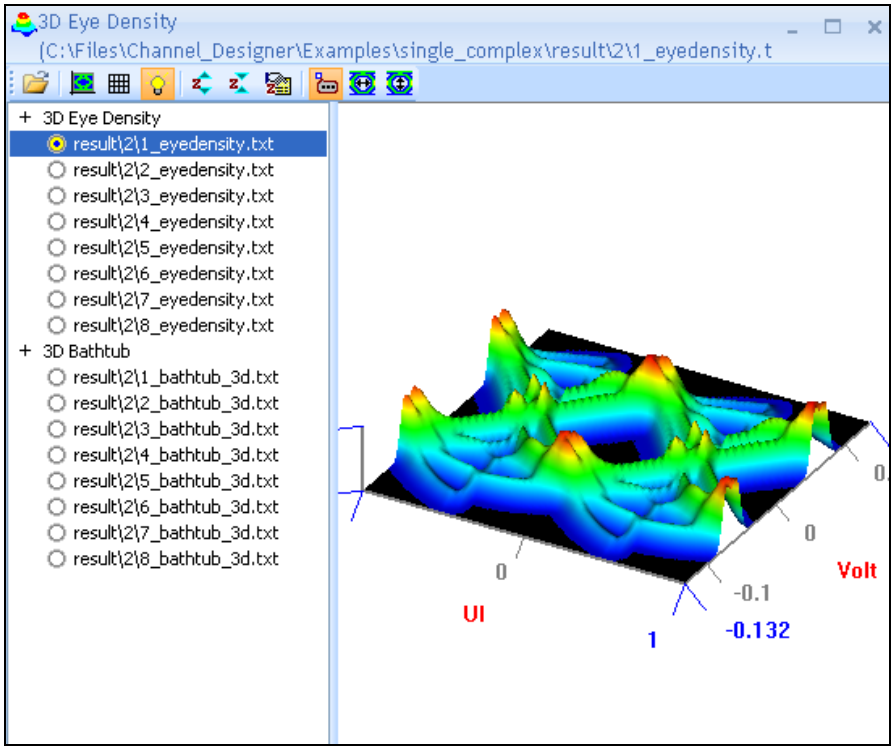

#### 4.5.5.2.7 2D Curve (Impulse Response)



#### 4.5.5.2.8 2D Curve (Rx Waveform)

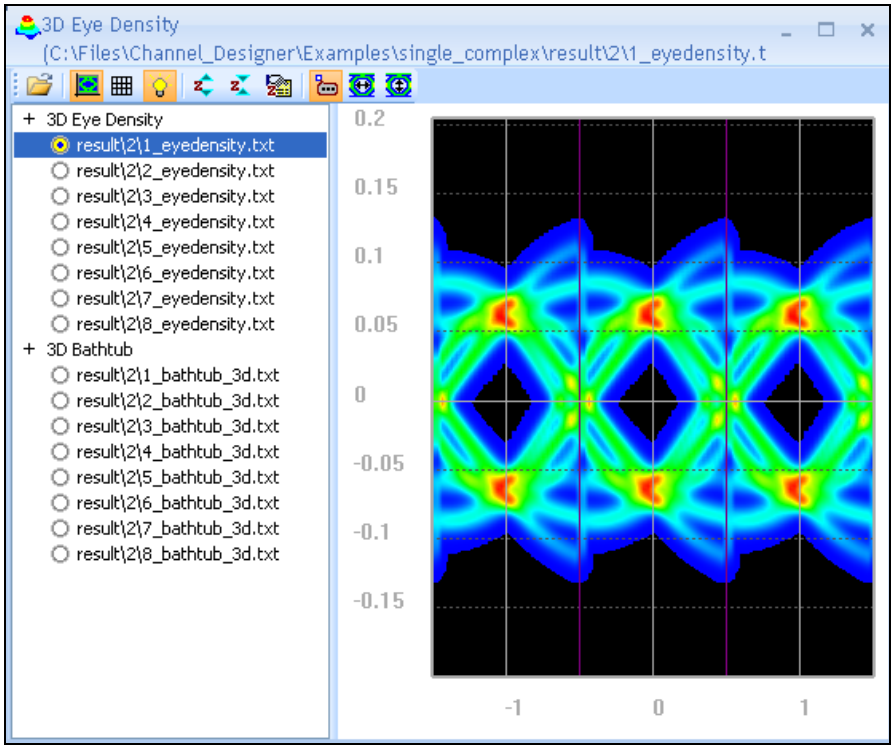



#### 4.5.5.2.9 3D Eye Density




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

#### 4.5.5.2.10 3D Eye Density



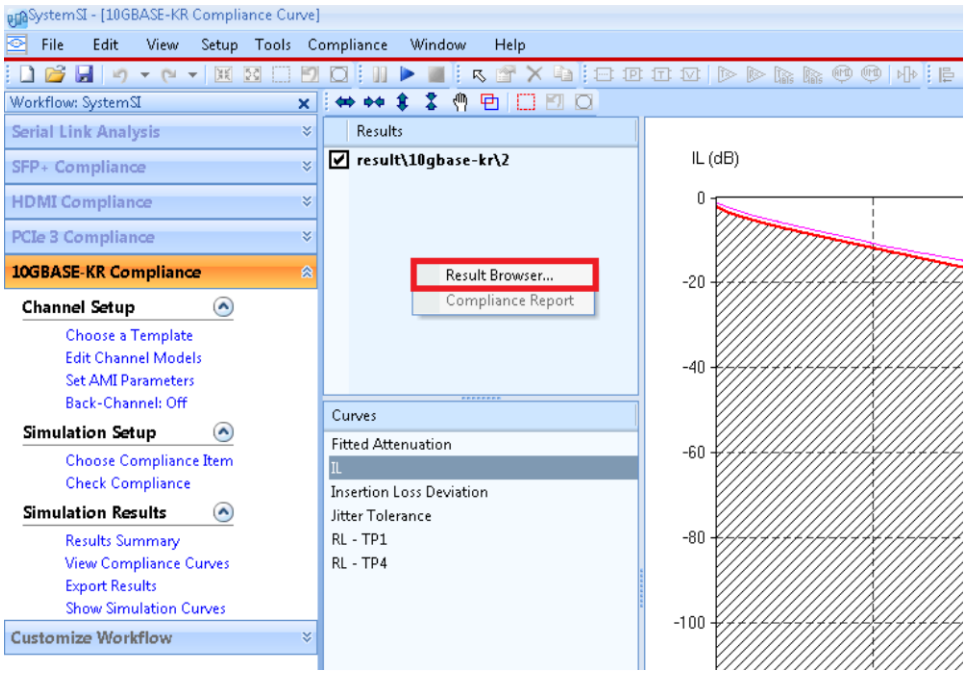

You can scroll the middle mouse button to zoom in and out, and you can also click  to add measurement cursors.

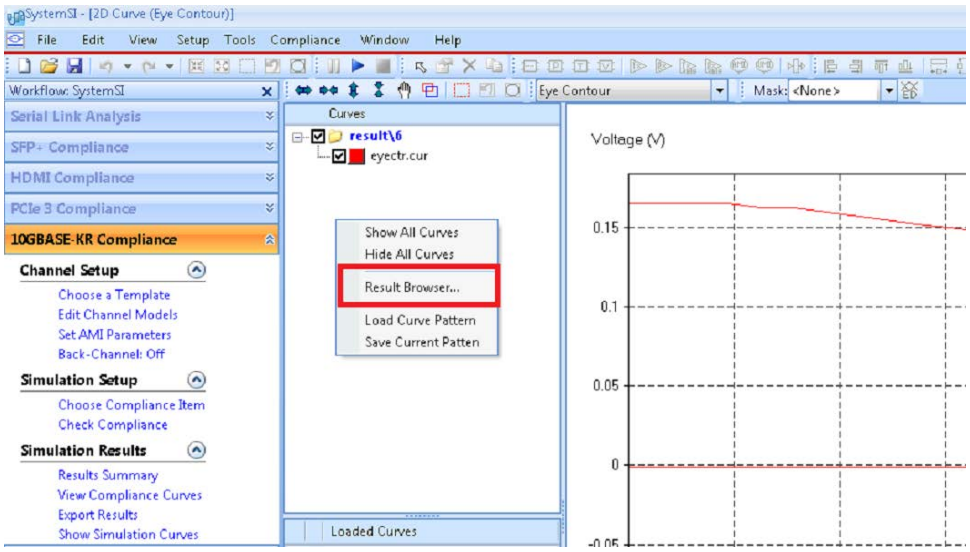
# 4.6 Result Browser and 2D Curve Presentation

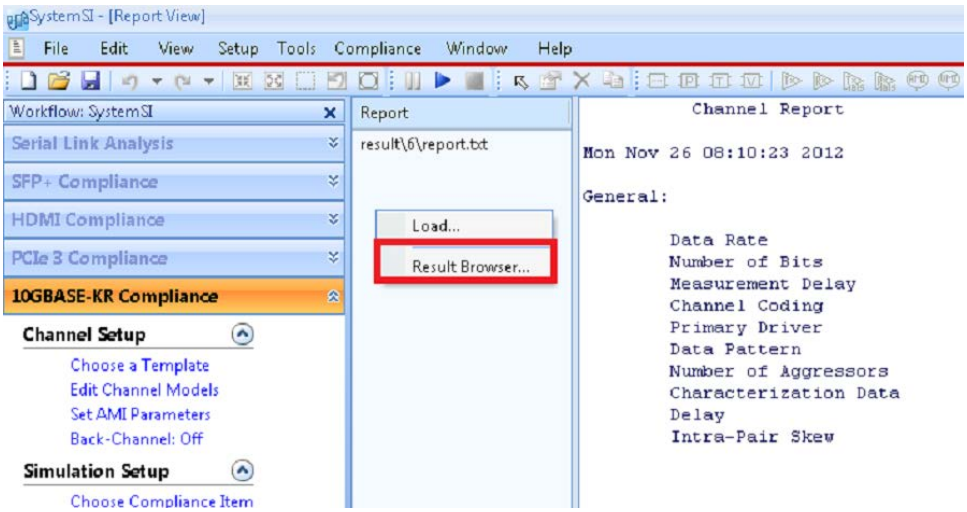
## 4.6.1 Result Browser

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.

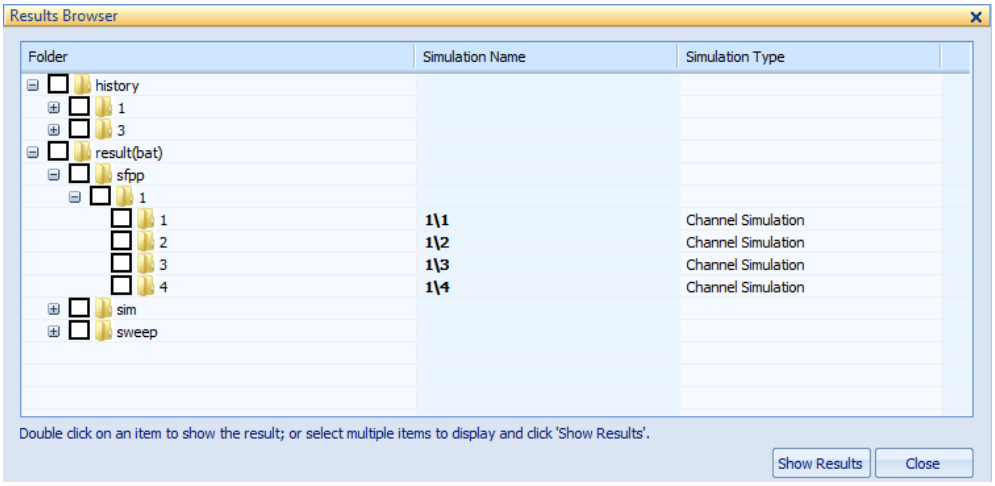
You can right-click in the curve panel, and choose **Result Browser…** from the pop-up menu to view the results. As shown in the following examples:


SystemSI - [10GBASE-KR Compliance Curve]
File Edit View Setup Tools Compliance Window Help
Workflow: SystemSI
Serial Link Analysis
SFP+ Compliance
HDMI Compliance
PCIe 3 Compliance
10GBASE-KR Compliance
Channel Setup
Choose a Template
Edit Channel Models
Set AMI Parameters
Back-Channel: Off
Simulation Setup
Choose Compliance Item
Check Compliance
Simulation Results
Results Summary
View Compliance Curves
Export Results
Show Simulation Curves
Customize Workflow
Results
result\10gbase-kr\2
Result Browser...
Compliance Report
Curves
Fitted Attenuation
IL
Insertion Loss Deviation
Jitter Tolerance
RL - TP1
RL - TP4
IL (dB)
0
-20
-40
-60
-80
-100


SystemSI - [2D Curve (Eye Contour)]
File Edit View Setup Tools Compliance Window Help
Workflow: SystemSI
Serial Link Analysis
SFP+ Compliance
HDMI Compliance
PCIe 3 Compliance
10GBASE-KR Compliance
Channel Setup
Choose a Template
Edit Channel Models
Set AMI Parameters
Back-Channel: Off
Simulation Setup
Choose Compliance Item
Check Compliance
Simulation Results
Results Summary
View Compliance Curves
Export Results
Show Simulation Curves
Eye Contour
Mask: <None>
Curves
result\6
eyectr.cur
Show All Curves
Hide All Curves
Result Browser...
Load Curve Pattern
Save Current Patten
Loaded Curves
Voltage (V)
0.15
0.1
0.05
0

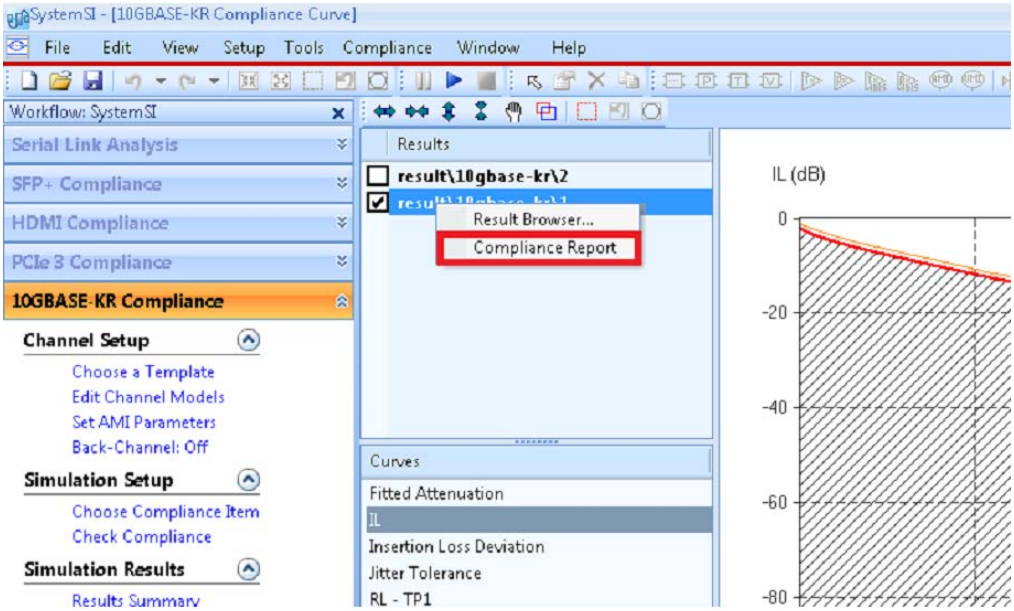



With this option, all the previous results under the **result**, **history** and **result(bat)** folders can be browsed and displayed from various curve windows.

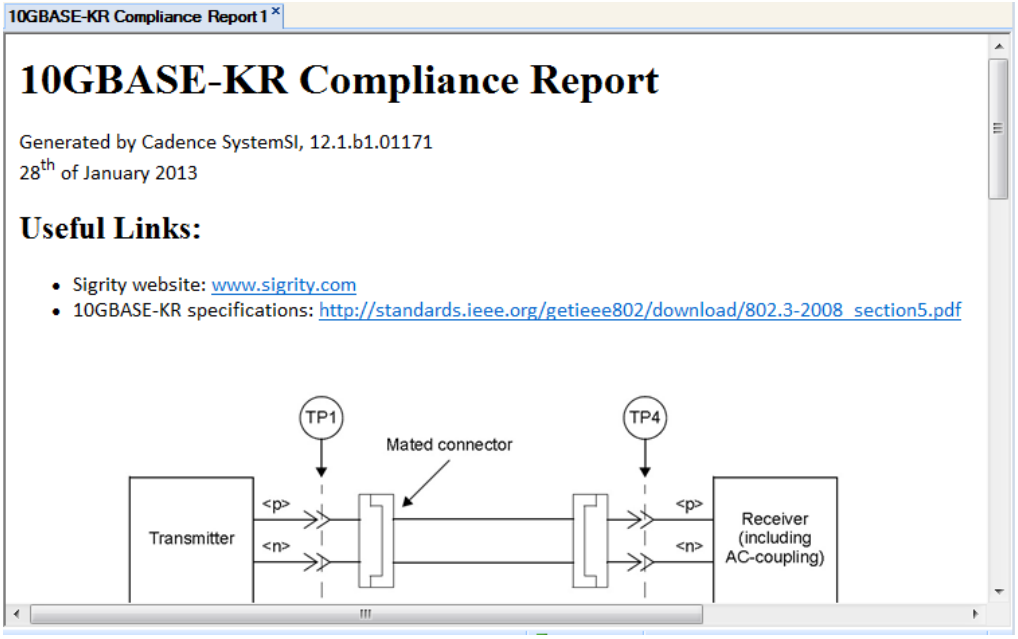



## 4.6.2 Compliance Curve Window

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

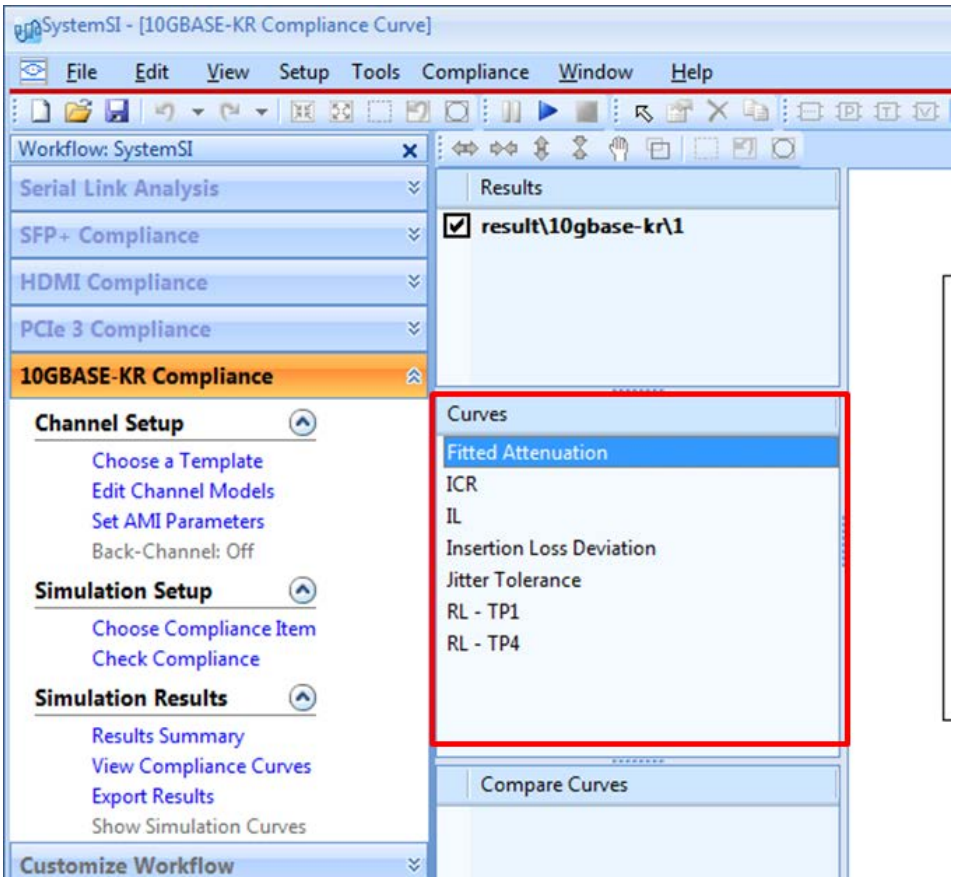



The previous generated Compliance report will be displayed.



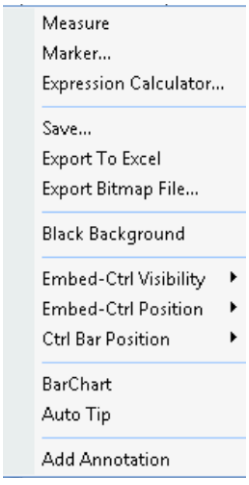

This works for all four SLA Compliance kits.

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.




You can perform the following operations in this menu.

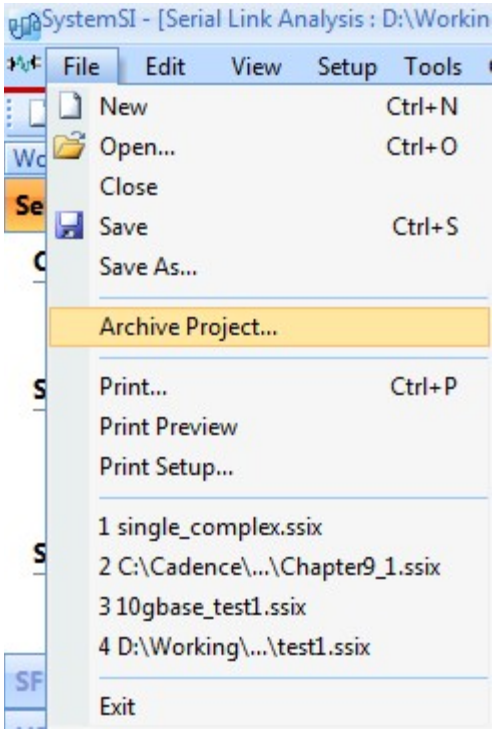
- **Measure** — Toggle the horizontal and vertical measure lines.
- **Marker** — Toggle the horizontal and vertical marker lines.
- **Expression Calculator**—Setup and calculation the expression.
- **Save**—Save the curve.

- **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.


By default,

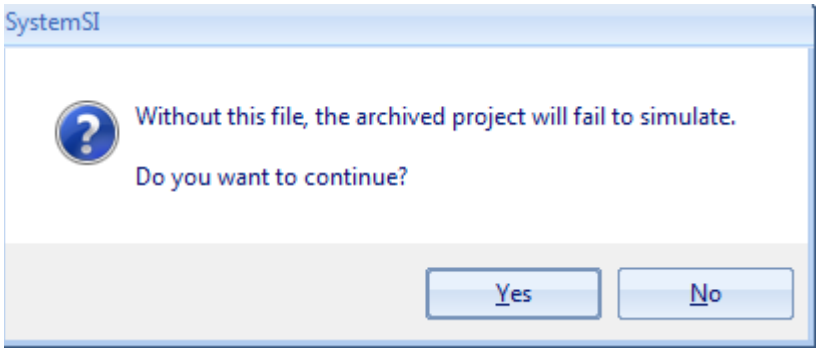
- The location of the original project is selected for the copied project location
- **<original_project_name>_copy** is set for the copied project name

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

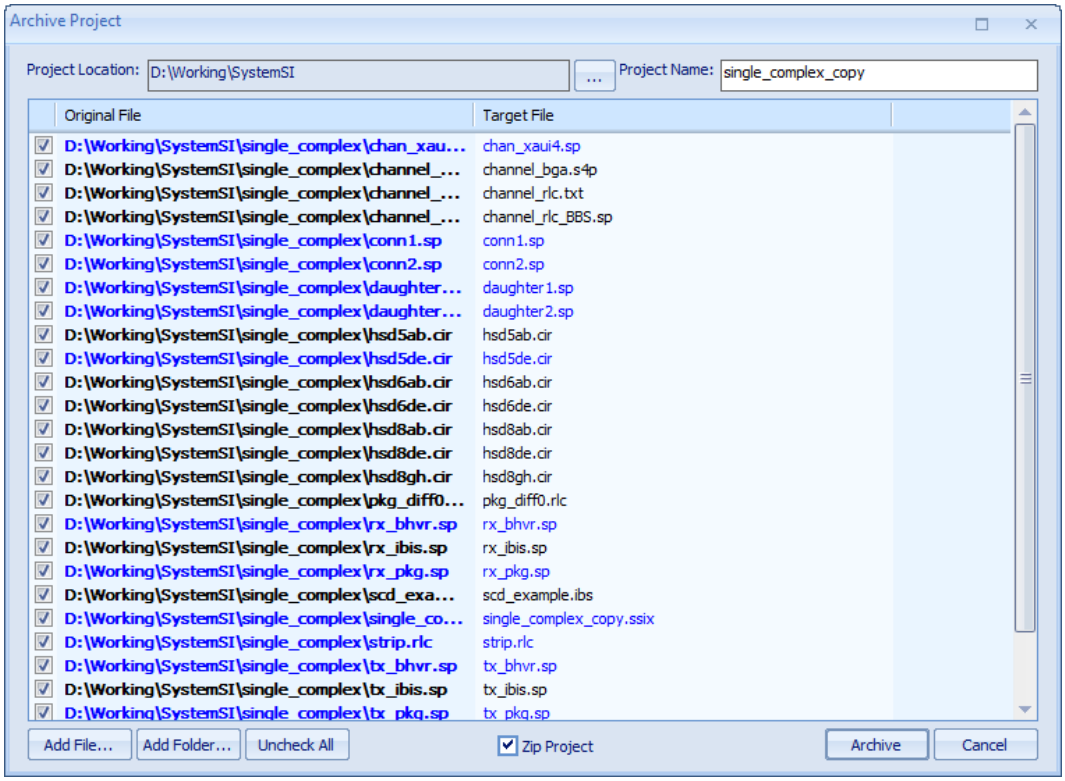
   By default,

   - 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

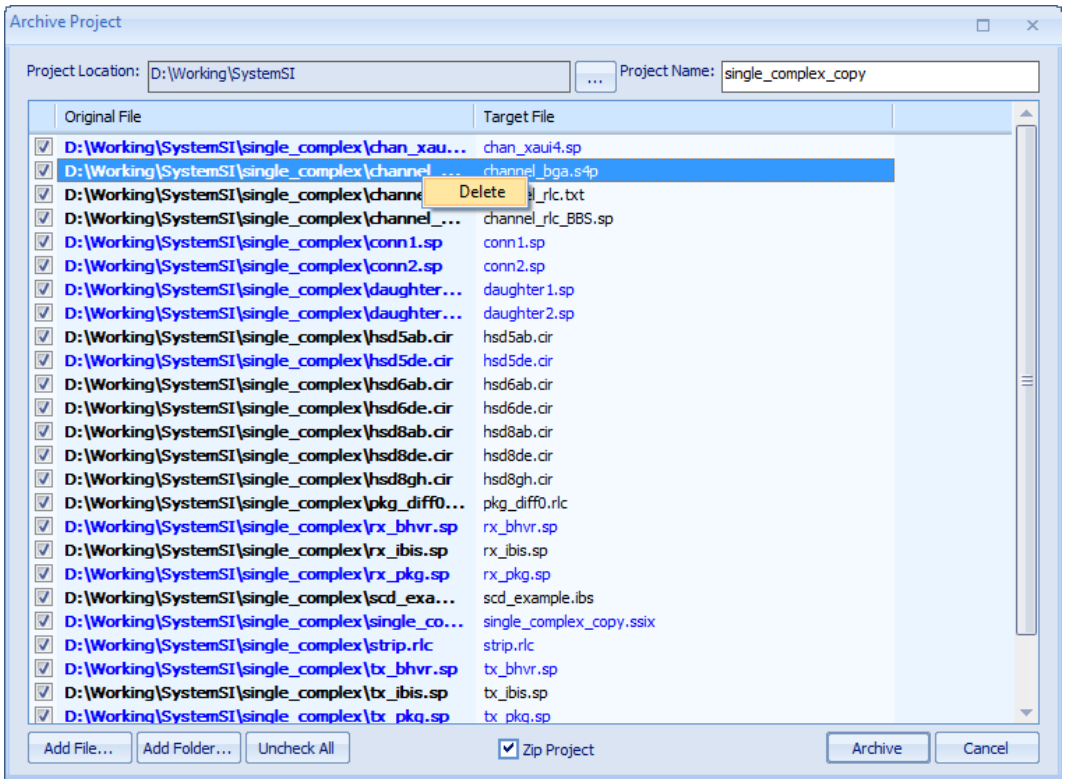
   If you want to uncheck a file used by the original project, the following message opens.



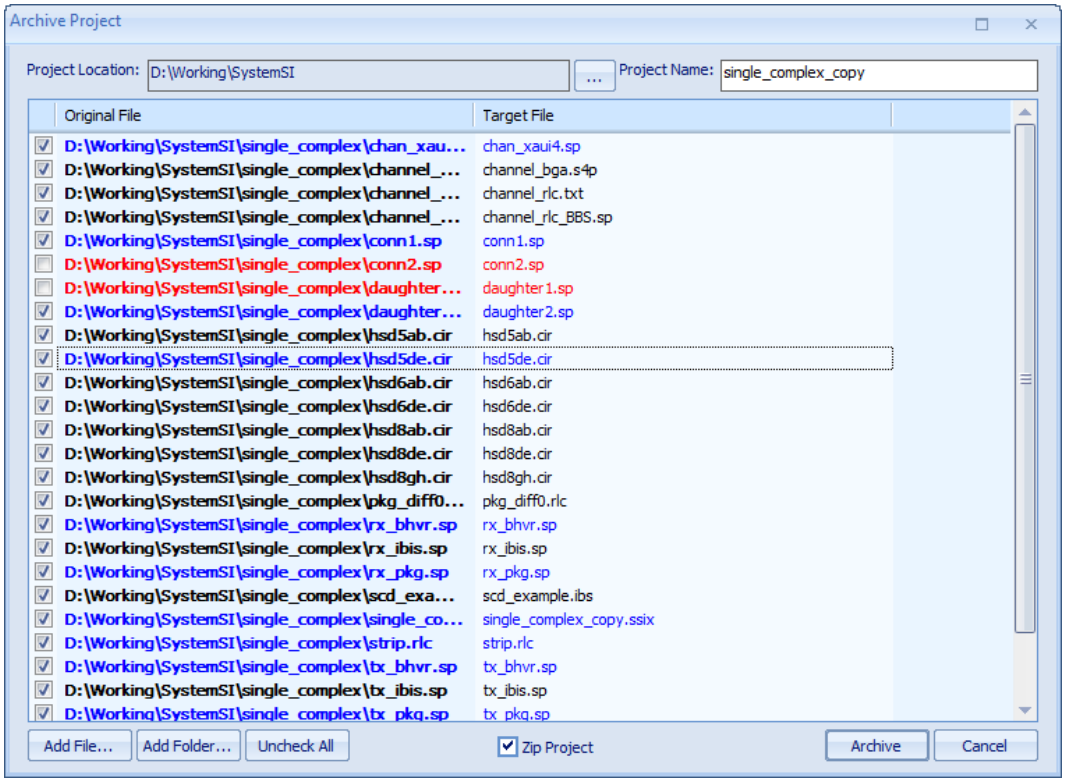

4. To add additional file, click the **Add File…** button.
5. To add additional folder, click the **Add Folder…** button.
   - The files checked and used by the original project are highlighted in blue




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




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

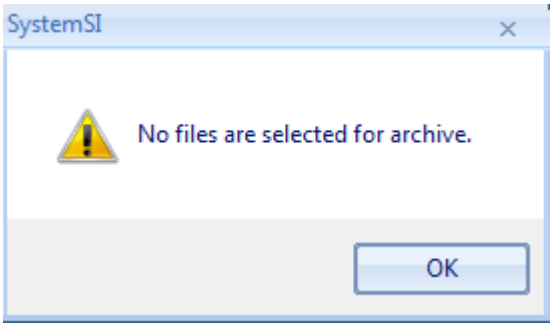


| NOTE! | If there are any files highlighted in red, the copied project will fail to simulate. |
|---|---|

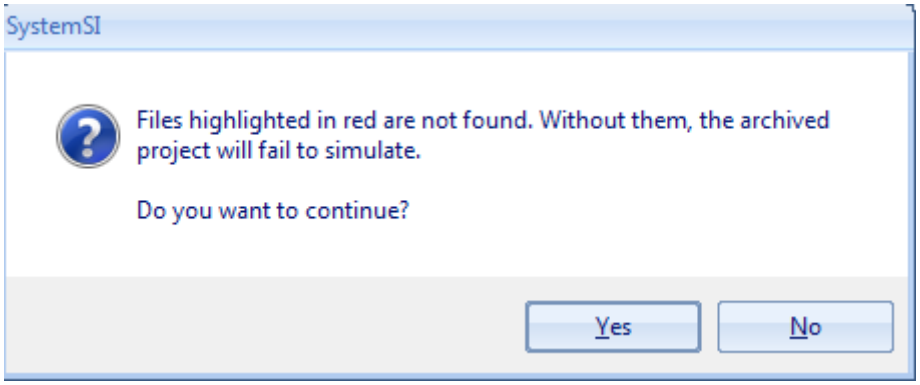
6. Click the **Archive** button to generate the copied project.
   - All the checked files will be copied to the copied project
   - If the **Zip Project** option is checked, a zipped project file will be created. Otherwise, a new project folder will be created for the copied project

   The following messages may appear while archiving:

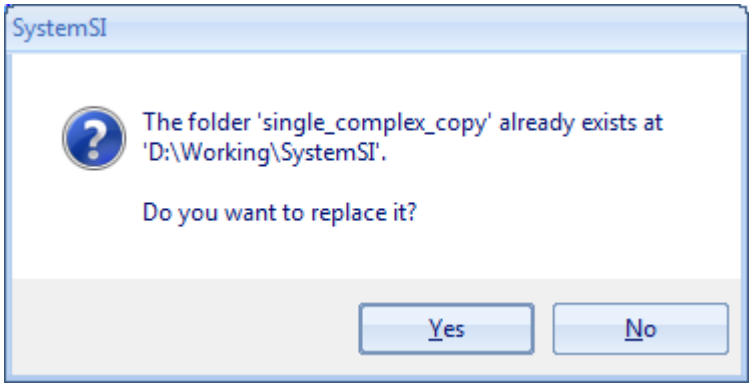
   - If no files are checked, the following message opens

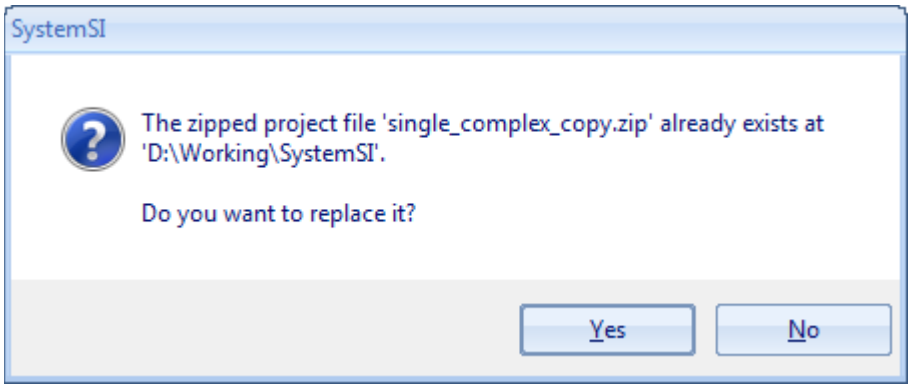



   - If the file used by the original project does not exist, the following message opens

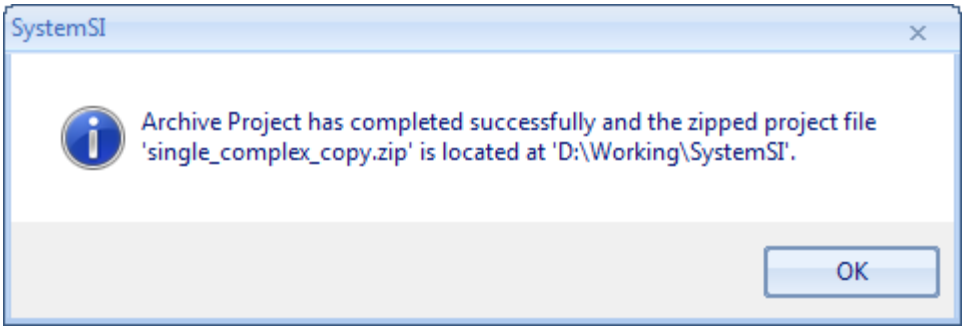


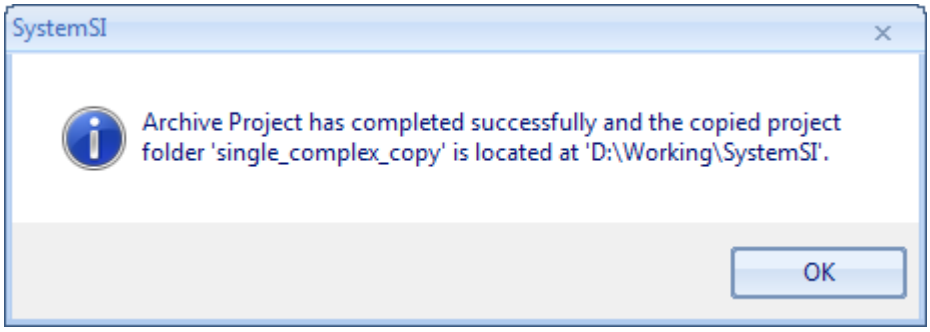

   - If the designated project folder already has the archived project folder or zipped file, the following messages open






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



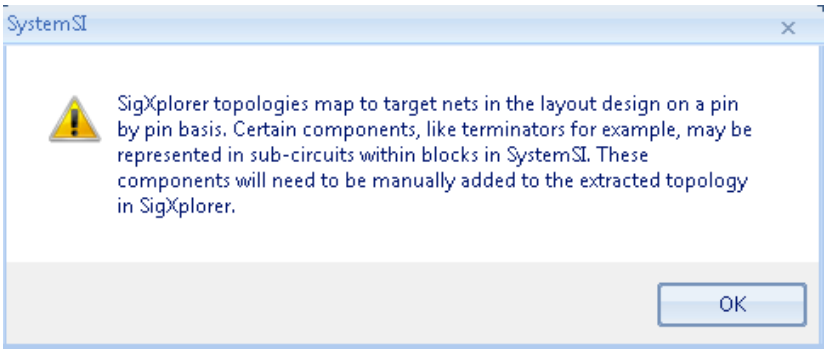

## 4.8 Integration with Allegro Signal Explorer

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**.

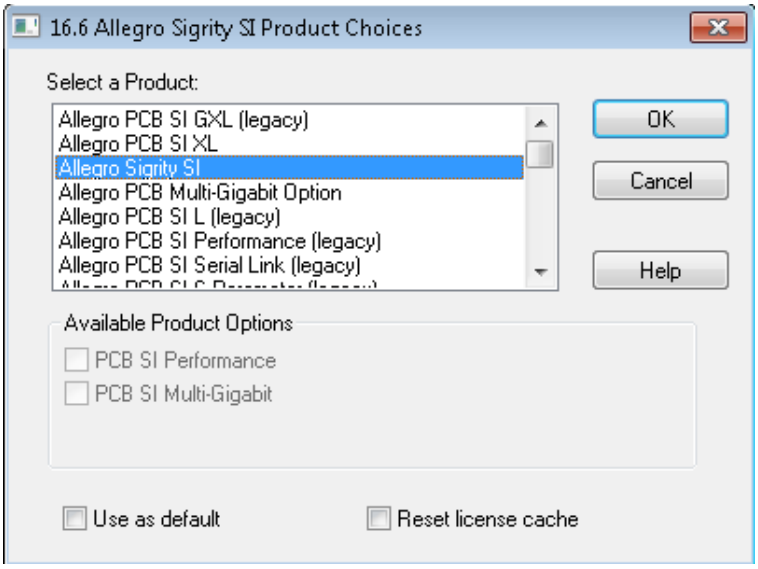
   Following message box is displayed.



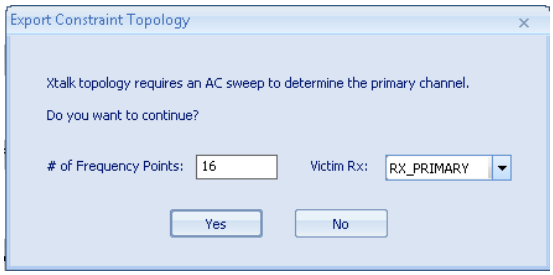

2. Select **OK**, to close the message box.

   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**.




   The exported topology is displayed in Allegro SigXplorer.

| | |
|---|---|
| **NOTE!** | For projects with cross-talk topologies, before the start of the export process, the Export Constraint Topology dialog box is displayed.<br>  <br><br>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 **Yes**. |

# 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:

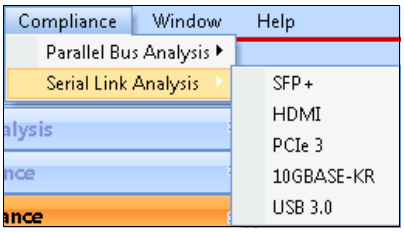
- Compliance Workflows
- SFP+ Compliance
- HDMI Compliance
- PCIe Compliance
- 10GBASE-KR Compliance
- USB 3.0 Compliance
- MIPI Compliance

## 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.

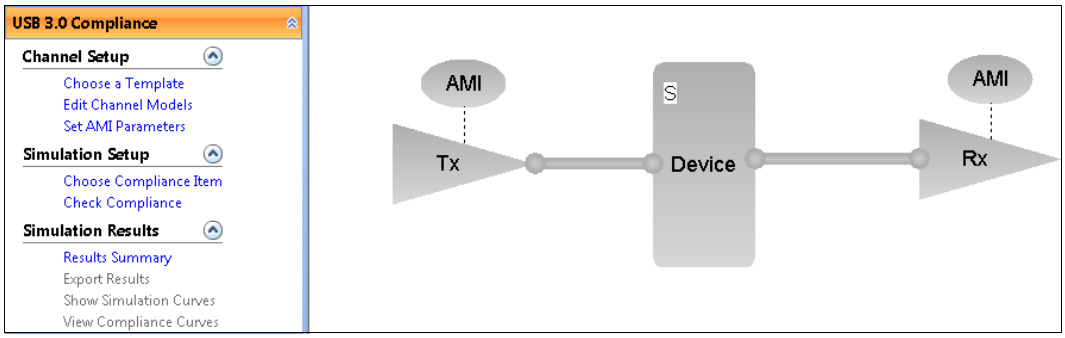



3. To display a compliance workflow, select the compliance from the sub-menu.

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.



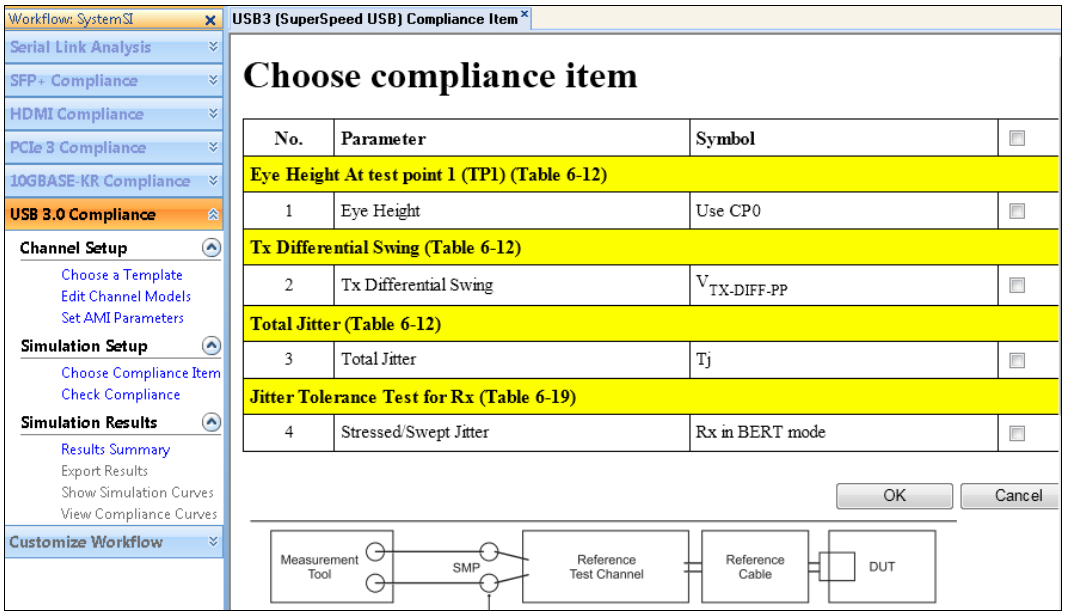

Selecting a template and associating appropriate models to the template blocks are common tasks to be performed for all compliance kits. Depending on the compliance selected, you may need to perform an extra step of setting AMI Parameters.

### 5.1.2 Compliance Checks

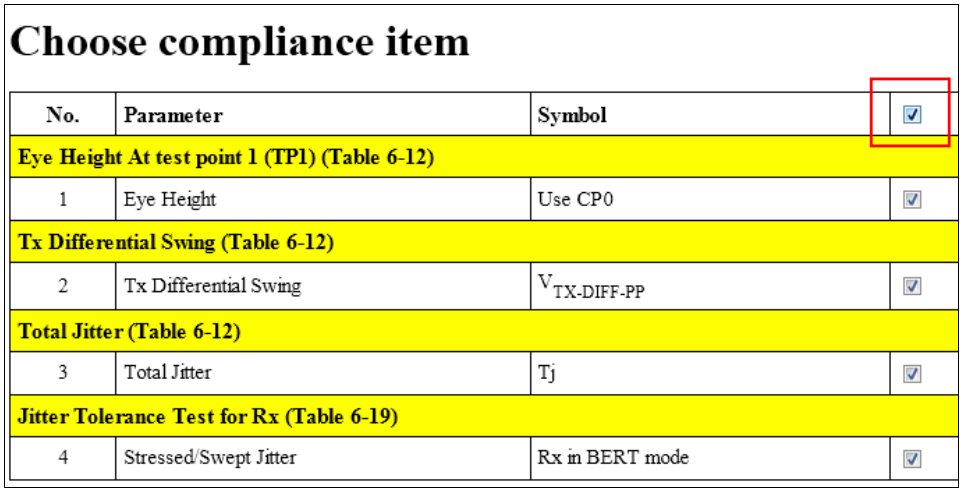
After you have specified the models associated with each block, you can run the compliance checks.

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.



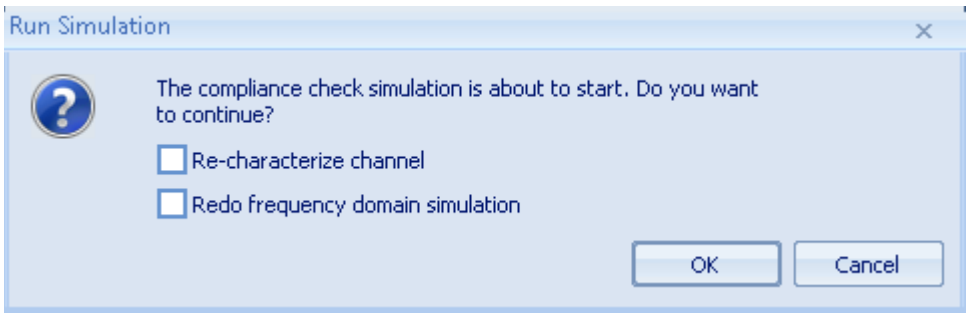

2. You can either select all options by selecting the check box in the first row, or can select individual options by selecting the corresponding check boxes in the last column.




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.



- ***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***

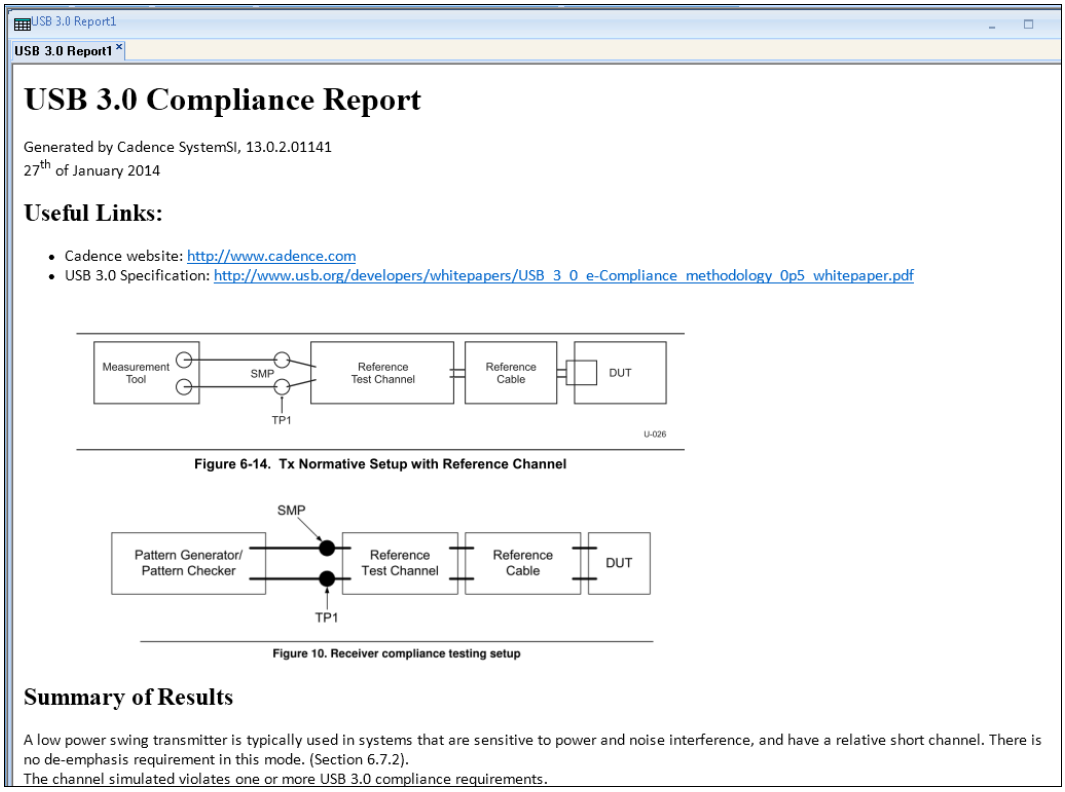
  Select this option if you want to run the frequency domain simulation (This is same as signal connectivity check in other Sigrity tools). This option has no impact on the channel simulation results, and can be ignored.

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.

**Eye Height At test point 1 (TP1)**

| Parameters | Symbol | Min | Max | Units | Simulation Results | Pass/Fail |
|---|---|---|---|---|---|---|
| Eye Height | | 0.1 | 1.2 | V | 0.144 | Pass |

**Tx Differential Swing**

| Parameters | Symbol | Min | Max | Units | Simulation Results | Pass/Fail |
|---|---|---|---|---|---|---|
| Tx Differential Swing | $V_{TX-DIFF-PP}$ | 0.8 | 1.2 | V | 1.122 | Pass |

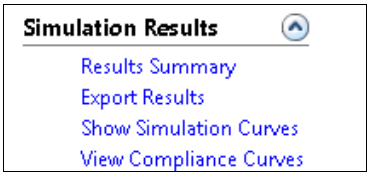
**Total Jitter**

| Parameters | Symbol | Min | Max | Units | Simulation Results | Pass/Fail |
|---|---|---|---|---|---|---|
| Total Jitter | Tj | | 0.66 | UI | 0.670 | Fail |

**Jitter Tolerance Test for Rx**

| Parameters | Symbol | Min | Max | Units | Simulation Results | Pass/Fail |
|---|---|---|---|---|---|---|

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

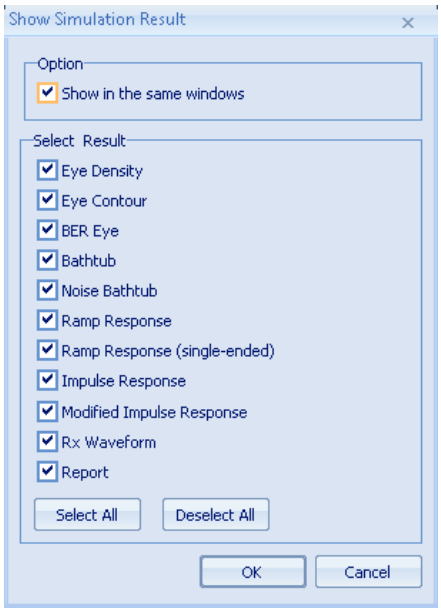



## Export Results

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

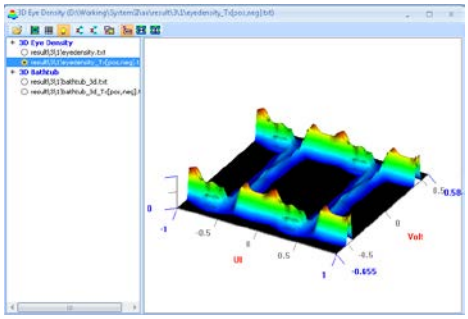
## Show Simulation Curves

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

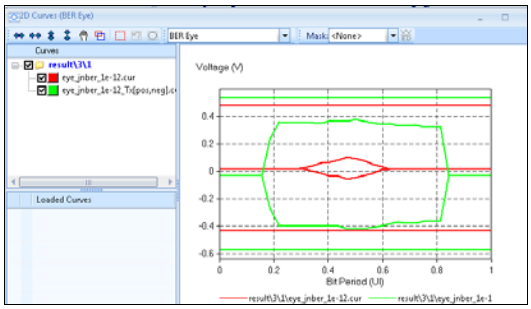



- Accept the default selection and click OK.

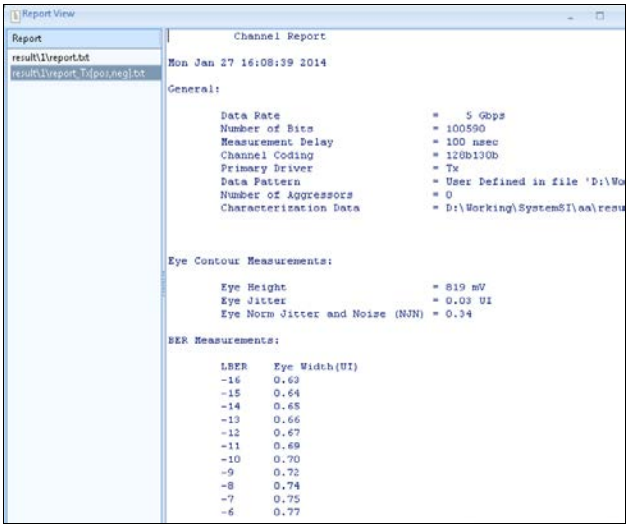
Simulation curves are displayed in 3 windows; 2D curves window, 3D Curves, and report.



*Eye Density*



*BER EYE*




*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.

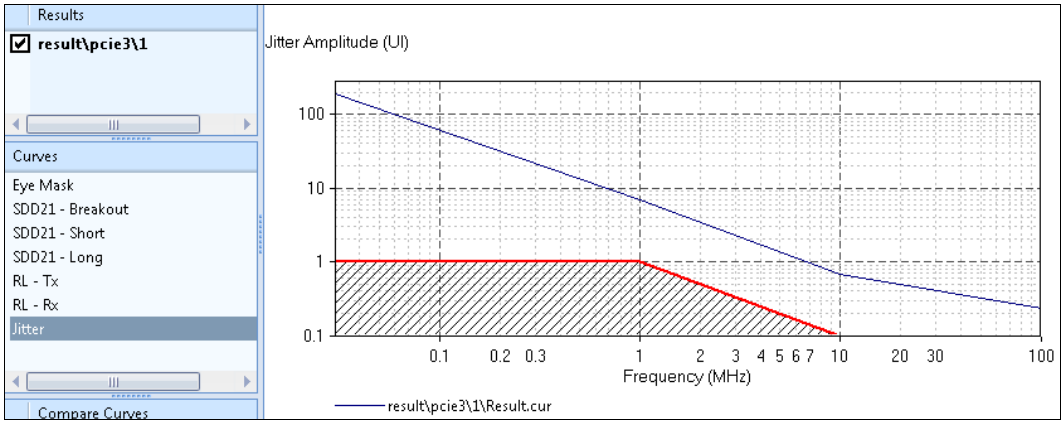



*Figure 5-1 PCIe 3 Compliance Curves*

| 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..

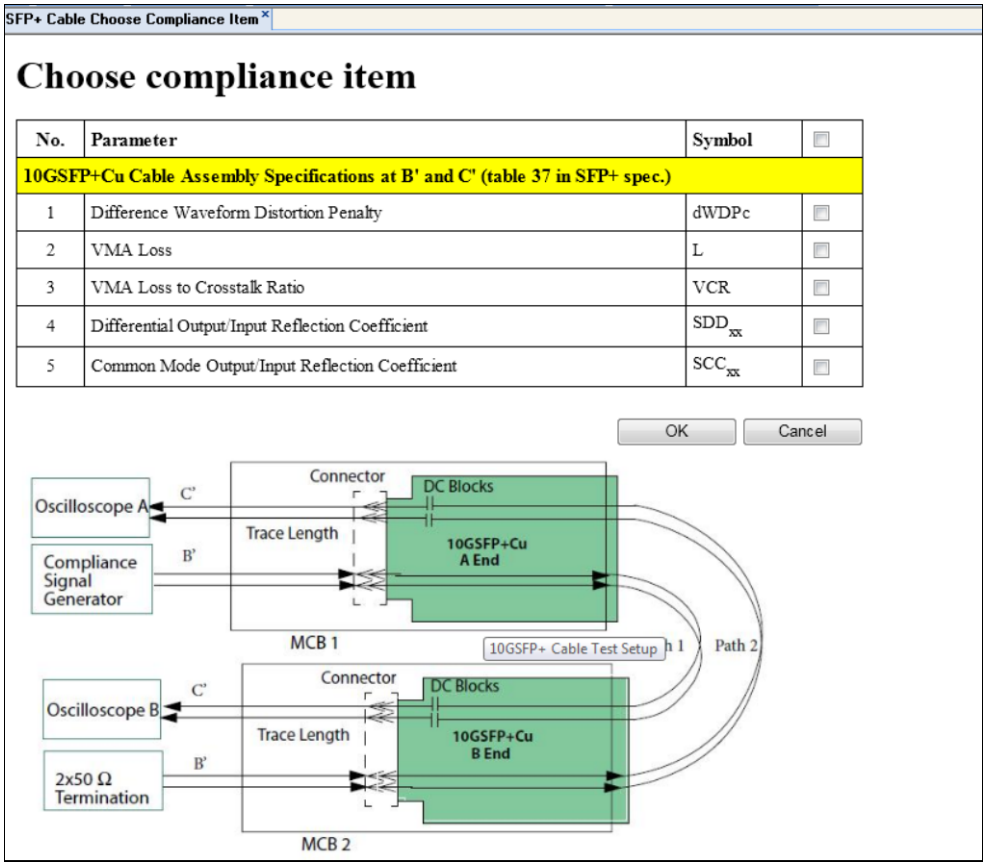
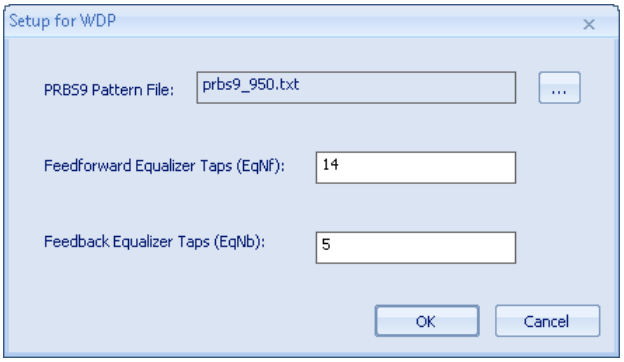



*Figure 5-2: SFP+ Compliance Checks for Cables*

| No. | Parameter | Symbol | |
|---|---|---|---|
| **Host Transmitter Output Specification at B (table 11 in SFP+ spec.)** | | | |
| 1 | Termination Mismatch at 1MHz | $DZ_M$ | ☐ |
| 2 | Differential Output S-parameter | SDD22 | ☑ |
| | Common Mode Output S-parameter | SCC22 | |
| **Host Transmitter Output Jitter and Eye Mask Specification at B (table 12 in SFP+ spec.)** | | | |
| 3 | Signal Rise/Fall Time (20% to 80%) | Tr, Tf | ☐ |
| 4 | Total Jitter | TJ | ☐ |
| 5 | Data Dependent Jitter | DDJ | ☐ |
| | Data Dependent Pulse Width Shrinkage | DDPWS | |
| 6 | Uncorrelated Jitter | UI | ☐ |
| 7 | Transmitter Qsq | Qsq | ☐ |
| 8 | Eye Mask | Eye Mask | ☐ |
| **Host Receiver Input Specification at C (table 13 in SFP+ spec.)** | | | |
| 9 | Differential Input S-parameter | SDD11 | ☐ |
| | Reflected Differential to Common Mode Conversion | SCD11 | |
| **SFI Channel Transfer Recommendation (table 25 in SFP+ spec.)** | | | |
| 10 | Channel Transfer from chip pad to point B or C | SDD21 | ☐ |
| **Additional 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:




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 | $V_{swing}$ | ☑ |
| 2 | Single-ended high level output voltage | $V_H$ | ☑ |
| 3 | Single-ended low level output voltage | $V_L$ | ☑ |
| **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 | | ☑ |
| 9 | Eye Mask | | ☑ |
| **Engineering Target** | | | |
| 10 | Differential Insertion Loss | SDD21 | ☑ |
| 11 | Differential Impedance | $Z_{DIFF}$ | ☐ |
| 12 | Single-ended Impedance | $Z_{SE}$ | ☐ |
| 13 | Intra-Pair Skew at Source Connector, max | | ☑ |
| 14 | Inter-Pair Skew at Source Connector, max | | ☑ |
| 15 | NEXT / FEXT | | ☑ |

*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 | ☐ |
|---|---|---|---|
| **Channel Tolerancing Eye Mask Values (table 4-27 in PCI Express Base spec.)** | | | |
| 1 | Eye Height | $V_{RX-CH-EH}$ | ☐ |
| 2 | Eye Width at Zero Crossing | $T_{RX-CH-EW}$ | ☐ |
| 3 | Peak EH Offset from UI Center | $T_{RX-DS-OFFSET}$ | ☐ |
| 4 | Range for DFE $d_1$ Coefficient | $V_{RX-DFE-COEFF}$ | ☐ |
| 5 | Eye Mask | | ☐ |
| **Differential Insertion Loss (figure 4-66 in PCI Express Base spec.)** | | | |
| 6 | Insertion Loss | SDD21 | ☐ |
| **Differential Return Loss (figure 4-56 in PCI Express Base spec.)** | | | |
| 7 | Tx Return Loss | RL - Tx | ☐ |
| 8 | Rx Return Loss | RL - Rx | ☐ |
| **Stressed/Swept Jitter Test (figure 4-74 in PCI Express Base spec.)** | | | |
| 9 | Stressed/Swept Jitter | | ☐ |

*Figure 5-5PCIe 3 Compliance 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.

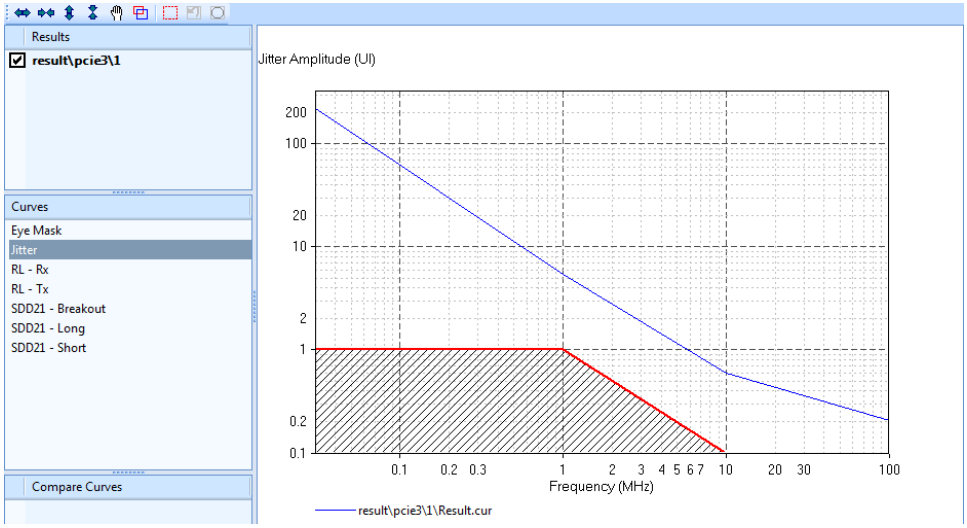
| | |
|---|---|
| **NOTE!** | A PCIe 3 specific jitter mask is shipped with SystemSI. For PCIe 3 compliance kit, load the **Pcie3_jtolmask.txt** mask file, before running the compliance checks. |

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 |

- Select the Jitter link in the report, to view the jitter tolerance simulation plot.



## 5.5 10GBASE-KR Compliance

| No. | Parameter | Symbol | ☑ |
|---|---|---|---|
| **Interference Tolerance (Annex 69A of the 10GBASE-KR spec.)** | | | |
| 1 | Interference Tolerance | | ☑ |
| **Interconnect Characteristics (Annex 69B of the 10GBASE-KR spec.)** | | | |
| 2 | Skew between P and N Side of Thru Diff Pair | | ☑ |
| 3 | Insertion Loss | IL | ☑ |
| 4 | Fitted Attenuation | A | ☑ |
| 5 | Insertion Loss Deviation | ILD | ☑ |
| 6 | Tx Return Loss at TP4 | RL - TP4 | ☑ |
| 7 | Rx Return Loss at TP1 | RL - TP1 | ☑ |
| 8 | Ratio of Insertion Loss to Crosstalk between TP1 and TP4 | ICR | ☑ |

*Figure 5-6 10GBASE-KR Compliance Checks*

### *Jitter Tolerance*

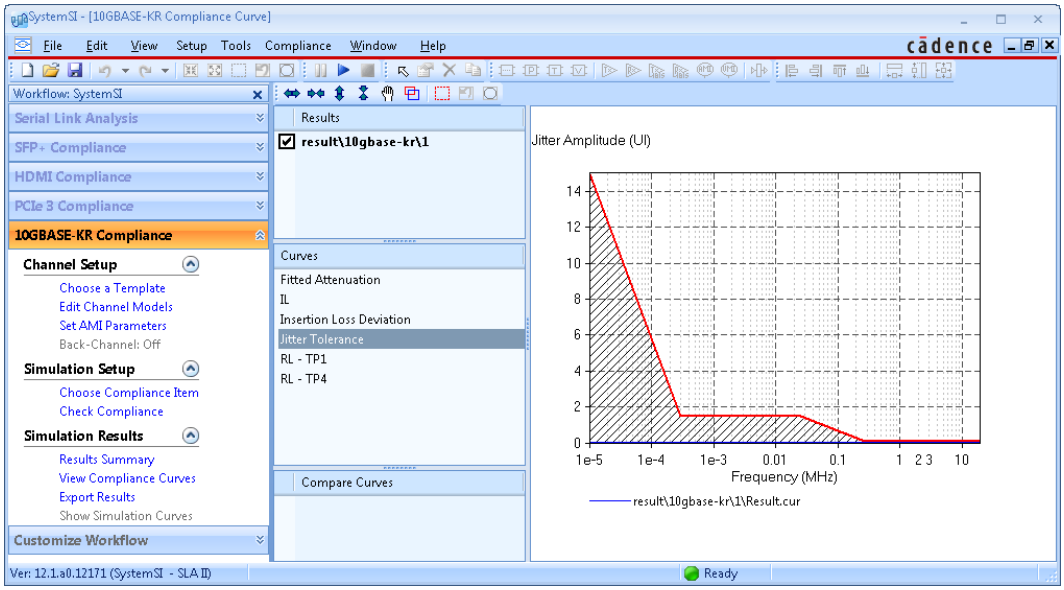
To run the Jitter Tolerance analysis for **10GBASE**-KR compliance, select **The Interference Tolerance** compliance check.

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 Height At test point 1 (TP1) (Table 6-12)** | | | |
| 1 | Eye Height | Use CP0 | ☐ |
| **Tx Differential Swing (Table 6-12)** | | | |
| 2 | Tx Differential Swing | $V_{TX-DIFF-PP}$ | ☐ |
| **Total Jitter (Table 6-12)** | | | |
| 3 | Total Jitter | Tj | ☐ |
| **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.
- 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

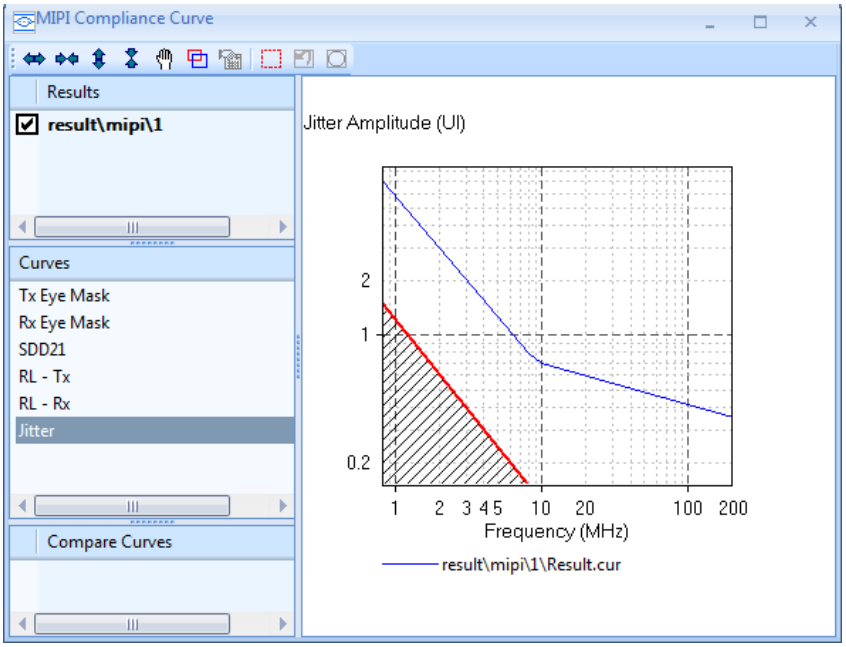
The MIPI standard is targeted for short range (less than one meter) mobile applications that require low pin count and low power consumption. SystemSI MIPI compliance kit supports high speed MIPI serial links.

| No. | Parameter | Symbol | ☑ |
|---|---|---|---|
| **TX Eye Mask Values (Table 16 in M_Phy spec version 3.0)** | | | |
| 1 | Eye Height | $V_{DIF\_AC\_HS\_G3\_TX}$(>80 mv) | ☑ |
| 2 | Eye Width at Zero Crossing | $T_{EYE\_HS\_G3\_TX}$(>0.55 UI) | ☑ |
| 3 | Eye Mask | | ☑ |
| **Channel Tolerancing Eye Mask Values(Table 21 in M_Phy spec version 3.0)** | | | |
| 4 | Eye Height | $V_{DIF\_AC\_HS\_G3\_RX}$(>80 mv) | ☑ |
| 5 | Eye Width at Zero Crossing | $T_{EYE\_HS\_G3\_RX}$(>0.58 UI) | ☑ |
| 6 | Eye Mask | | ☑ |
| **Differential Return Loss(Figure 31 and 43 in M_Phy spec version 3.0)** | | | |
| 7 | Tx Return Loss | RL - Tx | ☑ |
| 8 | Rx Return Loss | RL - Rx | ☑ |
| **Stressed/Swept Jitter Test(Figure 44 in M_Phy spec 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>
```

\\ for the frequency response

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

\\ 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>
```

\\ 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>
```

\\ 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>
```

\\ 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>
```

\\ for the 10gbase-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>
```

\\ 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. |
|---|---|

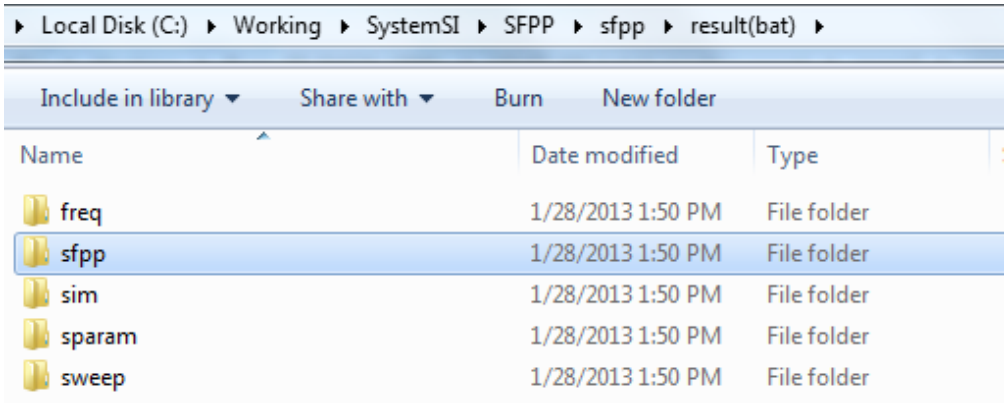
2. Double-click the `.bat` file to run it.

## A.3 Result Folder for Batch Mode

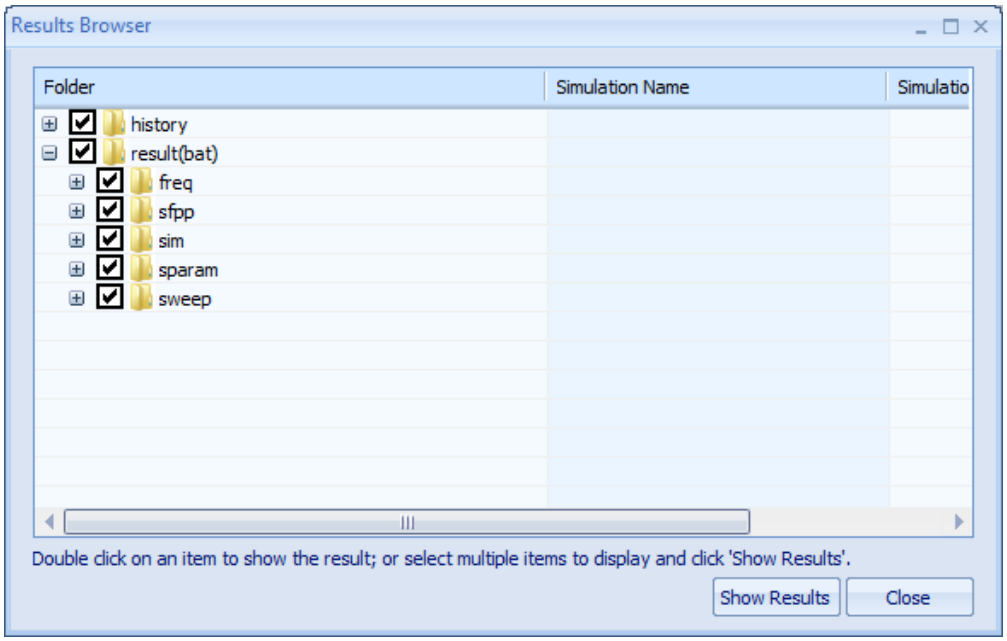
A new sub-folder **result(bat)** will be added for all batch mode results. Each batch mode simulation has its own sub-folder under the **result(bat)** folder.

Example

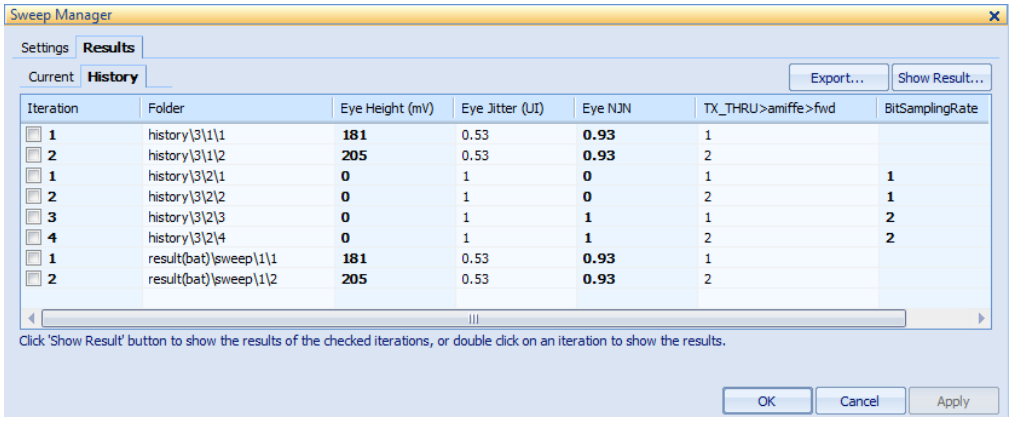
For a SLA workspace, each common result is located in its responding sub-folder. The compliance result is in a folder named with the compliance type.




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.



## 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