

SigXplorer Command Reference

Product Version 23.1
September 2023

© 2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

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

Allegro SigXplorer contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

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

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

All other trademarks are the property of their respective holders.

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

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

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Cadence is committed to using respectful language in our code and communications. We are also active in

the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

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

Contents

<u>File Menu Commands</u>	11
<u>File – New</u>	11
<u>File – Open</u>	12
<u>File – Example</u>	13
<u>Procedure</u>	13
<u>File – Save</u>	14
<u>Procedure</u>	14
<u>File – Save As</u>	15
<u>Procedures</u>	15
<u>File – Update Constraint Manager</u>	17
<u>File – Append</u>	18
<u>Appending Topologies</u>	18
<u>Procedure</u>	20
<u>Examples</u>	21
<u>File – Import – Measurement</u>	25
<u>File – Import – Sweep Case</u>	27
<u>File – Export – Measurement</u>	29
<u>Procedures</u>	29
<u>File – Export – Spreadsheet</u>	31
<u>Procedures</u>	31
<u>File – Capture Canvas Image</u>	32
<u>Procedure</u>	32
<u>File – Print Canvas Setup</u>	33
<u>Procedures</u>	35
<u>File – Print Canvas</u>	36
<u>Procedures</u>	37
<u>File – Print Spreadsheet</u>	38
<u>File – Script</u>	39
<u>Procedures</u>	40
<u>File – Recent Topologies</u>	42

Allegro SI SigXplorer Reference

<u>Procedure</u>	42
<u>File – Exit</u>	43

Edit Menu Commands

<u>Edit – Undo</u>	45
<u>Procedures</u>	45
<u>Edit – Move</u>	46
<u>Edit – Copy</u>	47
<u>Edit – Delete</u>	48
<u>Edit – Rotate Right</u>	49
<u>Procedure</u>	49
<u>Edit – Rotate Left</u>	50
<u>Procedure</u>	50
<u>Edit – Mirror</u>	51
<u>Procedure</u>	51
<u>Edit – Add Element</u>	53
<u>Procedures</u>	57
<u>Edit – Note</u>	61
<u>Procedures</u>	61
<u>Note Editing</u>	62
<u>Modifying, Moving, and Deleting Existing Notes</u>	62
<u>Edit – Transform for Constraint Manager</u>	64
<u>Procedure</u>	64
<u>Edit – Cleanup</u>	65
<u>Procedure</u>	65

View Menu Commands

<u>View – Zoom By Window</u>	67
<u>Procedures</u>	67
<u>View – Zoom Fit</u>	68
<u>View – Zoom Center</u>	69
<u>View – Zoom In</u>	70
<u>View – Zoom Out</u>	71
<u>View – Zoom Previous</u>	72
<u>View – Windows</u>	73

Allegro SI SigXplorer Reference

<u>View – Customize Toolbar</u>	74
<u>View – Reset UI to Cadence default</u>	75

Setup Menu Commands

<u>Setup – Constraints</u>	77
<u>Set Topology Constraints Dialog Box</u>	77
<u>Procedures</u>	94
<u>Setup – Defaults</u>	109
<u>Dialog Box</u>	109
<u>Procedures</u>	112
<u>Setup – Strobe Pins</u>	113
<u>Dialog Box</u>	113
<u>Setup – Vectors</u>	115
<u>Procedures</u>	115
<u>Setup – Optional Pins</u>	118
<u>Procedures</u>	118
<u>Setup – Manage LayerStacks</u>	120
<u>Procedures</u>	121

Analyze Menu Commands

<u>Analyze – Model Browser</u>	125
<u>Dialog Boxes</u>	125
<u>Procedure</u>	144
<u>Working with Models and Libraries</u>	144
<u>Working with Device Models</u>	148
<u>Procedures</u>	157
<u>Working with Interconnect Models</u>	159
<u>Analyze – Via Setup Preferences</u>	160
<u>Analyze – Via Model Generation</u>	166
<u>Analyze – Preferences</u>	177
<u>Dialog Boxes</u>	177
<u>Procedures</u>	193
<u>Analyze – Simulate</u>	202
<u>Dialog Boxes</u>	202
<u>Procedures</u>	204

Allegro SI SigXplorer Reference

<u>Analyze – Simulate Continue</u>	207
<u>Analyze – [S] Generation</u>	208
<u>Dialog Boxes</u>	208
<u>Procedures</u>	211
<u>Analyze – Reset Sim Data</u>	213

Help Menu Commands

<u>Help – Documentation</u>	215
<u>Help – What’s New</u>	215
<u>Help – Web Resources</u>	215
<u>Help – Licenses Used</u>	216
<u>Help – About</u>	216

Context-Sensitive Menus

<u>Editing a Custom Measurement</u>	217
<u>The Expression Name and Description Fields</u>	217
<u>The Expression Treeview Window</u>	217
<u>The Editing Pull-Down Menu</u>	218
<u>Expanding and Collapsing the Treeview</u>	218
<u>Navigating the Treeview with the Keyboard Arrow Keys</u>	219
<u>The Check Button</u>	219
<u>The Add Operand Button</u>	219
<u>The Clear Item Button</u>	219
<u>The Undo and Redo Buttons</u>	220
<u>The Cancel and OK Buttons</u>	220
<u>The Status Bar</u>	220
<u>Procedures</u>	220
<u>The Measurement Expression Editor Dialog Box</u>	221
<u>The Expression Editing Pull-Down Menu</u>	222
<u>The Editing Pull-down Menu</u>	222
<u>The Pull-Down Menu Choices</u>	224
<u>Numbers</u>	224
<u>Measurements</u>	224
<u>Math Functions</u>	225
<u>Waveform Functions</u>	226

Allegro SI SigXplorer Reference

<u>Pin Parameter Function</u>	227
<u>Simulation Functions</u>	228
<u>Editing a Custom Measurement</u>	228
<u>Creating a New Custom Measurement</u>	230
<u>Importing a Custom Measurements File</u>	231
<u>Creating a Custom Measurements File</u>	232
<u>Set Parameters</u>	236
<u>Dialog Box</u>	236
<u>Procedures</u>	238
<u>Set Buffer Parameters</u>	239
<u>Set Buffer Parameter Dialog Box</u>	239
<u>Modifying the Stimulus for an IOCell</u>	241
<u>IOCell Stimulus Edit Dialog Box</u>	241

Allegro SI SigXplorer Reference

File Menu Commands

File – New

Clears the topology canvas, the Parameters tab, and the Results tab, de-selects all measurements in the Measurements tab, and displays the *Starting New Drawing* message in the Command tab.

File – Open

Opens a file browser to display topology (.top) files. You can select the topology file to open or browse other directories if you do not see the correct topology file.

File – Example

Opens a file browser that displays examples of topologies.

Procedure

Viewing topology examples

1. Choose *File – Example*.

A file browser appears listing the examples of topology templates.

2. Select an example from the list that appears in the file browser. The types of available topology examples include:

Example	Description
<i>ibis_delay_fixture.top</i>	Measures a buffer delay with a test fixture.
<i>ibis_golden_diff.top</i>	Portrays a golden waveform of a differential pair.
<i>ibis_golden_single.top</i>	Portrays a golden waveform.
<i>ibis_vt_fixture.top</i>	Measures a VT curve with a test fixture.
<i>sparam_4port.top</i>	Illustrates an S-Parameter model in a topology.

File – Save

Saves changes to the topology (.top) file you are currently editing.

Procedure

Saving the current topology file

- Choose *File – Save*.

The current topology is saved by overwriting the existing topology file with the same name.

File – Save As

Opens the Save As file browser display topology (.top) files.

Procedures

Saving the current topology file under a new name

1. Choose *File – Save As*.

The Save as file browser appears and displays topology (.top) files.

2. From the Save in drop-down list box, select the directory where you want to save the file.
3. Enter the file name in the File name text box.
4. From the Save as type drop-down list box, select the file type you want to use.
5. Click Save.

The topology is saved in the specified directory with the specified name.

Narrowing the display of files

In the Save as type drop-down list box:

- Select Topology xxx name (*.top) to display all topology files in the directory. This is the default.
- or -
- Select All Files (".") to display all of the files in the directory.

The list of files changes to show the types of files you selected.

Using the Change Directory checkbox

Enable the Change Directory checkbox to designate which directory is set as the working directory. Change Directory is enabled by default. If you choose a new directory to save to, and Change Directory is enabled, the new directory will become your working directory for all subsequent save commands.

Allegro SI SigXplorer Reference

File Menu Commands

Note: If you wish to keep your current working directory setting, but want to save the current file to a different directory, disable the Change Directory checkbox when you execute a Save As command.

File – Update Constraint Manager

Updates the Allegro SI database with the topology template you have applied to a net or a group of nets.

File – Append

[Procedures](#) | [Examples](#)

Appends a topology to the active topology.

To save time and avoid errors, you can reuse the work contained in a topology saved from a previous session. The append command lets you *append* a topology (archived on disk) to the *active* topology (resident in memory). The result is a *combined* topology.

Important

When SigXplorer encounters an entity name in the appended topology that conflicts with the entity name in the active topology, SigXplorer retains the entity name in the active topology and renames the entity from the appended topology in the resulting combined topology. SigXplorer renames the entity (reference designator, custom measurement, constraint, vector set, custom stimulus) of the appended topology by adding an underscore and a numerical sequence to the entity name. The topology append report identifies any changes made to entity names.

For example, if both the active and the appended topologies have a part with the reference designator "IO1," the active topology will retain its reference designator. The reference designator from the appended topology will be updated to "IO1_1" in the combined topology.

Appending Topologies

SigXplorer examines the topology to be appended and compares it with the active topology. Then SigXplorer performs the merge operation based on the following rules:

■ Reference Designations

If a part in the appended topology has the same reference designator as a part in the active topology, SigXplorer retains the name of the part in the active topology and renames the part from the appended topology in the resulting combined topology. As SigXplorer updates the reference designator, it also updates dependent entities such as custom measurements, constraints, strobe pins, and parameters.

Parameters can contain expressions that refer to other reference designators. If the reference designator is changed in a topology append operation, the updated reference designation will be reflected in the expression as well.

Note: SigXplorer looks for signal models for the parts contained in the appended topology. If they are not found, SigXplorer performs the append operation and identifies parts without signal models in the topology append report.

■ Constraints

SigXplorer merges constraints of the appended and active topologies. If a user-defined constraint, or a relative propagation delay, in the appended topology has the same name as a constraint in the active topology, SigXplorer retains the constraint name in the active topology and renames the constraint from the appended topology in the resulting combined topology. When SigXplorer encounters a naming conflict for constraints defined in the Wiring, Signal Integrity, Diff Pair, and Max Parallel tabs of the *Set – Constraints* command, it retains the constraint in the active topology and discards the constraint in the appended topology. The topology append report identifies any changes made to a constraint name.

SigXplorer retains global constraints (All/All, All Drivers/Receivers, Driver/Receiver, Longest/Shortest, Longest) of the active and appended topologies as long as there is not a naming conflict; otherwise, SigXplorer retains the constraint in the active topology and discards the constraint from the appended topology.

■ Custom Measurements

SigXplorer merges custom measurements of the appended and active topologies. If a custom measurement in the appended topology has the same name as a custom measurement in the active topology, SigXplorer retains the name of the measurement in the active topology and renames the measurement from the appended topology in the resulting combined topology. The topology append report identifies any changes made to a custom measurement name.

SigXplorer supports the definition of custom constraints for custom measurements. SigXplorer merges custom constraints of the appended and active topologies. If a default constraint name for a custom constraint in an appended topology conflicts with an existing user-defined constraint or a custom constraint of an active topology, SigXplorer renames the constraint name for the appended topology in the resulting combined topology. The topology append report identifies any changes made to a custom constraint name.

■ Custom Stimulus

SigXplorer retains custom stimuli of the appended and active topologies. If a stimulus name in the appended topology has the same name as the stimulus in the active topology, SigXplorer retains the name of the stimulus in the active topology and renames the stimulus from the appended topology in the resulting combined topology. The topology append report identifies any changes made to a saved stimulus name.

■ Vector Sets

SigXplorer retains saved vector sets of the appended and active topologies. If a vector set name in the appended topology has the same name as the vector set in the active topology, SigXplorer retains the name of the vector set in the active topology and renames the vector set of the appended topology in the resulting combined topology. The topology append report identifies any changes made to a saved vector set name.

■ Strobe Pins

SigXplorer retains strobe pin groups of the appended and active topologies in the resulting combined topology.

■ Design Links

If they share the same signal model, SigXplorer merges design links of the appended and active topologies in the resulting combined topology. If each topology uses a different design link signal model, SigXplorer aborts the append operation and identifies the reason in the topology append report.

■ Default values and preferences

SigXplorer retains the default values (choose *Set – Defaults*) and the analysis preferences (choose *Analyze – Preferences*) of the active topology in the resulting combined topology.

Procedure

Appending a topology

1. With the active topology open in SigXplorer, choose *File — Append*.

The Append dialog box appears.

2. Navigate to the directory which contains the topology file that you want to append.
3. Click the topology file to select it.
4. Click *Open*.

SigXplorer appends the selected topology to the active topology, resulting in an combined topology. SigXplorer asserts the appended circuit below, and to the right, of the active circuit.

Important

You must connect disjoint topologies in the combined topology before you can simulate.

SigXplorer also displays the topology append report. This report lists all changes that were made to the appended topology when it was merged to create the combined topology. If the append operation was aborted, the reasons for the failure are reported.

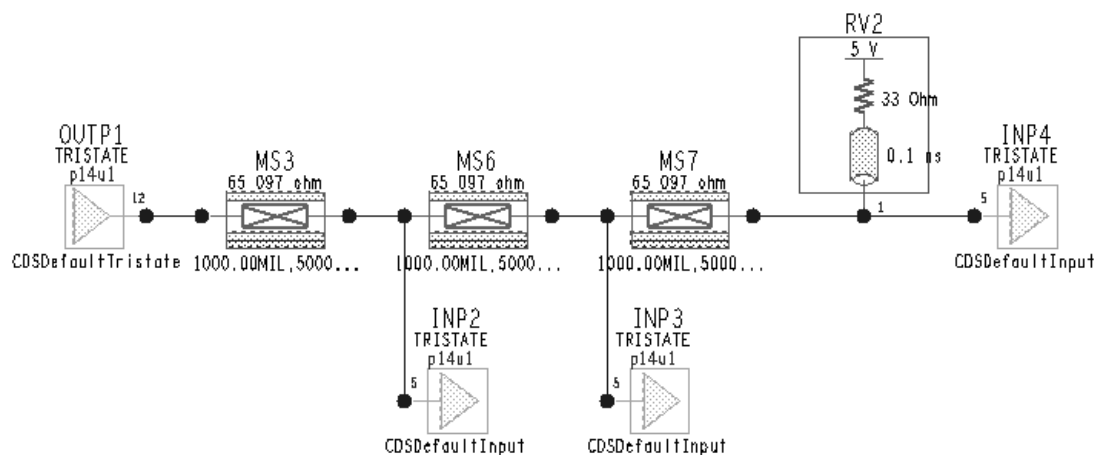
Examples

The following figures depict a topology append operation in Chip I/O Planner. It involves using SigXplorer Expert to append a topology for a laminate chip package assembly onto a board topology that represents the downstream loads.

Viewing the Active Topology

The circuit diagram shown below represents the load (on the board) as seen by a laminate chip package assembly (see [Figure 1-2](#) on page 22).

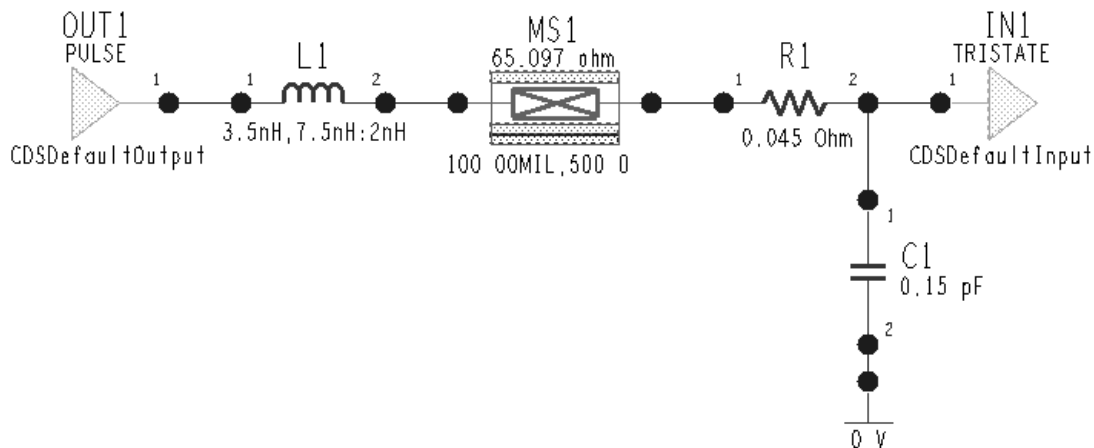
Figure 1-1 Active board topology



Understanding the Append Topology

The circuit diagram shown below represents the model for a laminated chip package assembly. The inductor models the wirebond, the transmission line models the package parasitics, and the series resistor and capacitor model the solder ball on the package.

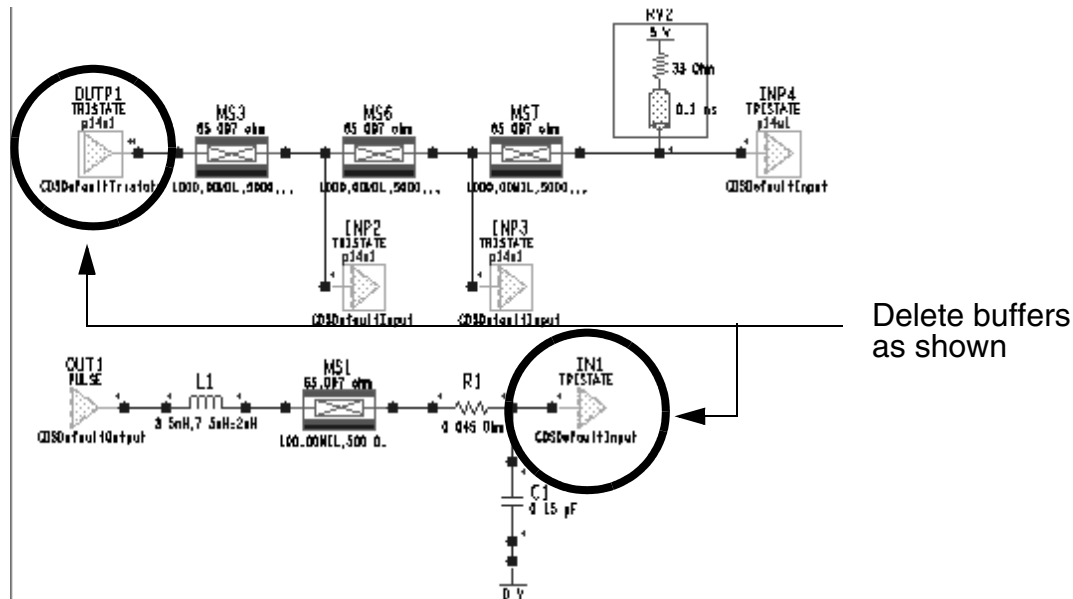
Figure 1-2 Laminate chip package assembly topology archived on disk



Combining the Topologies

The circuit diagram shown below depicts the combined topology after the append operation. Note that the package topology appears below the board topology, which you are currently editing. The active and append topologies are not yet connected.

Figure 1-3 Combined Topology (without connections)

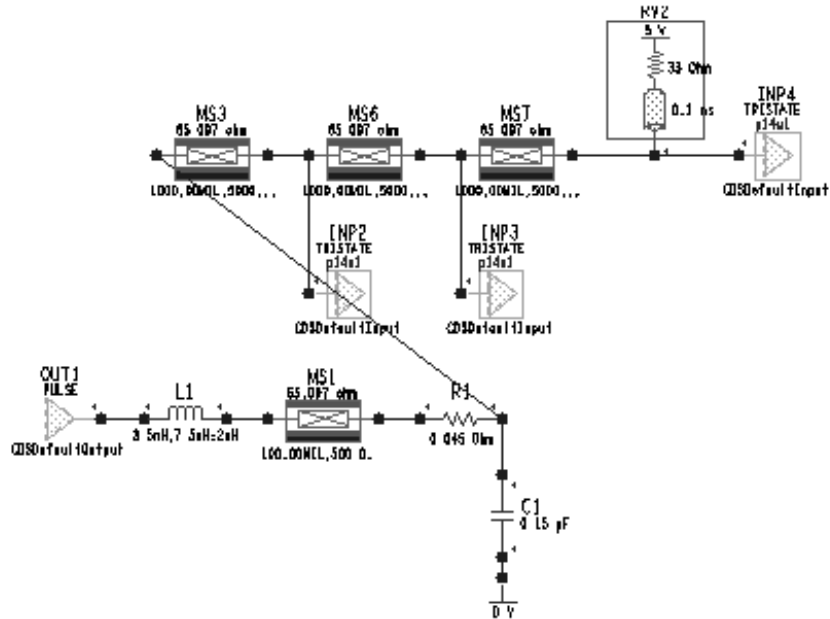


The output gate in the board topology can now be replaced with the physical package parasitics. Therefore, you should delete the output buffer of the board topology (top) and the input buffer of the package topology (bottom).

Connecting the Combined Topologies

You cannot simulate with disconnected parts; therefore, connect the two topologies as shown.

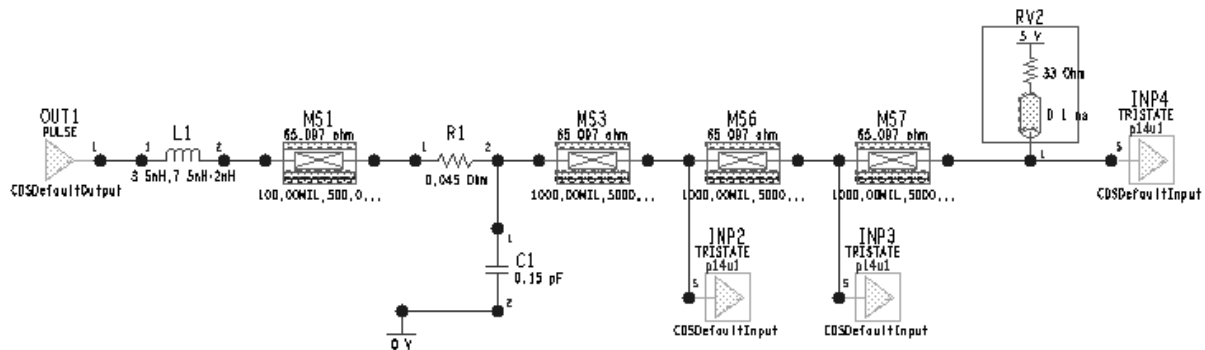
Figure 1-4 Initial connections



Cleaning up the Combined Circuit

Choose *Edit – Cleanup* to assert a smart-aligned topology. The result is shown in the diagram below.

Figure 1-5 Combined circuit, connected, and cleaned up



File – Import – Measurement

Imports a Custom Measurements (.dat) file associated with another topology.

Procedures

Importing Custom Measurements

1. Choose *File – Import – Measurement* to display the file browser window.
The Import Measurements file browser appears and displays Custom Measurements (.dat) files.
2. From the Look in drop-down list box, select the directory where you want to search for the file.
3. Enter the file name in the File name text box.
4. From the Files of type drop-down list box, select the file type you want to use.
5. Click Open.

The specified Custom Measurements file is displayed.

Narrowing the display of files

In the Files of type drop-down list box:

- Select Custom Measurements File (*.dat) to display all measurement files in the directory. This is the default.
- or -
- Select All Files (".") to display all of the files in the directory.

The list of files changes to show the types of files you selected.

Using the Change Directory checkbox

Enable the Change Directory checkbox to designate which directory is set as the working directory. Change Directory is disabled by default. If you choose a new directory to save to, and Change Directory is enabled, the new directory will become your working directory for all subsequent import commands.

Allegro SI SigXplorer Reference

File Menu Commands

Note: If you wish to keep your current working directory setting, but want to import a file from a different directory, be sure to disable the Change Directory checkbox when you execute the Import command.

File – Import – Sweep Case

[Dialog Box](#) | [Procedures](#)

Imports sweep case data that was previously saved.

Dialog Box

Option	Description
Sweep Case Selection	Displays the Case number, Topology and optional Description string for all SigXplorer sweep cases previously saved.
Delete Case	Deletes the selected case data and removes it from the case selection list.
Import Case	Imports the selected case data into SigXplorer.
Close	Ignores input and closes the dialog box.

Procedures

Importing a sweep case

1. Choose *File – Import – Sweep Case*.

The Import Sweep Case dialog box appears.

2. In the Sweep Case Selection frame, choose the sweep case you want to import from the list of cases.
3. Click Import Case.

The specified sweep case data is imported into SigXplorer.

Deleting a sweep case

1. Choose *File – Import – Sweep Case*.

The Import Sweep Case dialog box appears.

Allegro SI SigXplorer Reference

File Menu Commands

- 2.** In the Sweep Case Selection frame, choose the sweep case you want to remove from the list of cases.
- 3.** Click Delete Case.

The specified sweep case is removed from the list.

File – Export – Measurement

Creates a Custom Measurements (.dat) file containing all the custom measurements defined for this topology.

Procedures

Saving the current topology file under a new name

1. Choose *File – Export – Measurement*.

The Export Measurements file browser appears and displays Custom Measurements File (.dat).

2. From the Save in drop-down list box, select the directory where you want to save the file.
3. Enter the file name in the File name text box.
4. From the Save as type drop-down list box, select the file type you want to use.
5. Click *Save*.

The custom measurements are saved in the specified directory with the specified name.

Narrowing the display of files

In the Save as type drop-down list box:

- Select Custom Measurements File (*.dat) to display all measurement files in the directory. This is the default.

- or -

- Select All Files (".") to display all of the files in the directory.

The list of files changes to show the types of files you selected.

Using the Change Directory checkbox

Enable the Change Directory checkbox to designate which directory is set as the working directory. Change Directory is disabled by default. If you choose a new directory to save to, and Change Directory is enabled, the new directory will become your working directory for all subsequent export commands.

Allegro SI SigXplorer Reference

File Menu Commands

Note: If you wish to keep your current working directory setting, but want to save the current file to a different directory, be sure to disable the Change Directory checkbox when you execute the Export command.

File – Export – Spreadsheet

Opens the Save As file browser to display spreadsheet tabbed text (.txt) files.

Procedures

Saving the current spreadsheet under a new name

1. Choose *File – Export – Spreadsheet*.

The Save As browser appears and displays Spreadsheet Tabbed Text (.txt) files.

2. From the Save in drop-down list box, select the directory where you want to save the file.
3. Enter the file name in the File name text box.
4. From the Save as type drop-down list box, select the file type you want to use.
5. Click *Save*.

The spreadsheet is saved in the specified directory with the specified name.

Narrowing the display of files

In the Save as type drop-down list box:

- Select Spreadsheet Tabbed Text (*.txt) to display all measurement files in the directory. This is the default.
- or -
- Select All Files (".") to display all of the files in the directory.

The list of files changes to show the types of files you selected.

File – Capture Canvas Image

Use this command to capture the viewable area of the canvas to the clipboard. You can then paste the image into another application, such as a word processor or a drawing program.

Note: This command works only on Windows.

Procedure

To capture a canvas image

1. Use the zoom and move commands to set the desired view of the topology on the canvas.
2. Choose *File – Capture Canvas Image*.

SigXplorer writes the image to the clipboard as a resizable bitmap.

3. Paste the image in the application of your choice.

File – Print Canvas Setup

[Dialog Box](#) | [Procedures](#)

Displays the Plot Setup dialog box. Use this platform-specific dialog box to specify plot controls.

Dialog Box (UNIX)

Option	Description
Plot scaling	
Fit to page	Scales the plot to fit the full image on the page.
Scaling factor	Scales the plot by the specified scaling factor.
Default line weight	Specifies the line weight to be used in plotting. Converts any zero width line to a width proportional to the setting. Aids in displaying very thin lines on high-resolution output.
Plot orientation	
Auto center	Orients the plot on the center of the page. This control is automatically invoked when you choose the Fit to page option.
Mirror	Plots a mirror image (mirrored about the Y axis).
Plot method	
Color	Creates color plots. Color is determined by the method specific to the platform you are using. On UNIX, color is read from a user-supplied stipples file (sq_plot_param.stipples). If not found, plotter color defaults are used.
Black and white	Creates black and white plots.
Plot contents	
Screen contents	Plots the image bounded by the screen display.
Sheet contents	Plots the full image bounded by the sheet.

Allegro SI SigXplorer Reference

File Menu Commands

Option	Description
IPF setup	
Vectorize text	Specifies that text output to the IPF file is broken down into line vectors of the specified width. The default setting can be specified with the environment variables PLOT_VECTORIZE_TEXT and PLOT_VECTEXT_WIDTH. Note: UNIX automatically vectorizes text for the plot command, even if this setting is off.
OK	Creates the plot and closes the dialog box.
Cancel	Ignores input and closes the dialog box.
Help	Launches the SigXplorer Help system and displays the relevant Help topic.

Dialog Box (Windows NT)

Option	Description
Plot scaling	
Fit to page	Scales the plot to fit the full image on the page.
Scaling factor	Scales the plot by the specified scaling factor.
Default line weight	Specifies the line weight to be used in plotting. Converts any zero width line to a width proportional to the setting. Aids in displaying very thin lines on high-resolution output.
Plot orientation	
Auto center	Orients the plot on the center of the page. This control is automatically invoked when you choose the Fit to page option.
Mirror	Plots a mirror image (mirrored about the Y axis).

Allegro SI SigXplorer Reference

File Menu Commands

Option	Description
Plot method	
Color	Creates color plots. Color is determined by the method specific to the platform you are using. On Windows, color selection is determined by the setting in the Color/Visibility dialog box. If color is specified, but the plot is directed to a non-color plotter, output will plot in grey-scale.
Black and white	Creates black and white plots.
Plot contents	
Screen contents	Plots the image bounded by the screen display.
Sheet contents	Plots the full image bounded by the sheet.
IPF setup	
Vectorize text	Specifies that text output to the IPF file is broken down into line vectors of the specified width. The default setting can be specified with the environment variables PLOT_VECTORIZE_TEXT and PLOT_VECTEXT_WIDTH.
OK	Creates the plot and closes the dialog box.
Cancel	Ignores input and closes the dialog box.
Help	Launches the SigXplorer Help system and displays the relevant Help topic.

Procedures

Setting the plot configuration

1. Choose *File – Print Canvas Setup*.

The Plot Setup dialog box appears.

2. Select the various plot configurations you want to use, such as scaling factor, orientation, plot method, and plot contents.
3. Click *OK* to save the plot settings.

File – Print Canvas

Dialog Box | Procedures

Displays the Print dialog box for printing the current topology. The printed output is determined by the configuration settings you define using the *File – Print Canvas Setup* command.

Dialog Box (UNIX)

Option	Description
Printer	Identifies the current default printer.
Print quality	Defines the print resolution. You can choose either 300dpi or 600dpi resolution from the drop-down list. The default setting is 600dpi.
Print to file	Saves the print job to a PostScript file instead of printing directly to the printer. If you enable this checkbox, you must specify a file name to save the file to.
OK	Creates the plot and closes the dialog box.
Cancel	Ignores input and closes the dialog box.
Setup	Displays the standard Print Setup dialog box for specifying print settings.

Dialog Box (Windows NT)

Option	Description
Printer	Identifies the current default printer.
Print quality	Defines the print resolution. You can choose either 300dpi or 600dpi resolution from the drop-down list. The default setting is 600dpi.
Print to file	Saves the print job to a PostScript file instead of printing directly to the printer. If you enable this checkbox, you must specify a file name to save the file to.
OK	Creates the plot and closes the dialog box.
Cancel	Ignores input and closes the dialog box.

Allegro SI SigXplorer Reference

File Menu Commands

Option	Description
Setup	Displays the standard Windows Print Setup dialog box for specifying print settings.

Procedures

Printing the current topology

On UNIX

1. Choose *File – Print Canvas Setup*.
2. Configure the settings in the dialog box and click OK.
3. Choose *File – Print Canvas*.
4. If you want to direct output to a file, select Print to file and enter a filename. The file is written to the current working directory. Enter a full pathname to write the file to another location.
5. If you want to direct output to a printer, select Printer name and enter the printer name by pressing the down-arrow button and selecting the appropriate printer.
6. If necessary, click Pen Numbers to make color to pen assignments.
7. Click *OK*.

On Windows

1. Choose *File – Print Canvas Setup*.
2. Configure the settings in the dialog box and click OK.
3. Choose *File – Print Canvas*.
4. In the Print quality drop-down box, select the resolution. The default setting is 600dpi.
5. If you want to direct output to a file, enable the Print to file check box and enter a filename. The file is written to the current working directory. Enter a full pathname to write the file to another location.
6. If necessary, click Setup to set additional Windows Print Manager printing options.
7. Click *OK*.

File – Print Spreadsheet

Displays the standard Print dialog box for printing the contents of the currently displayed spreadsheet tab.

File – Script

Dialog Box | Procedures

Displays the Scripting dialog box, allowing you to record a series of actions in SigXplorer. You can record a script or macro, or replay them. A script is used to perform global tasks such as setting up dialog box options and adding objects to multiple topologies at the same location. A macro is a script that lets you automate a series of point selections and replay them, starting at another coordinate.

When you replay a macro, SigXplorer prompts you for a starting point (origin). The macro places the point selections you recorded relative to this starting point. This placement is useful for performing operations that need to be repeated in a design.

When you save a script, SigXplorer creates a text file (with a `.scr` extension) containing the commands you executed.

Note: The current settings in your design are recorded in the script or macro. If you want the script to display with different settings, you must change them as part of the script.

Dialog Box

Option	Description
File	Specifies the name of the file you want to record your actions in. The <code>.scr</code> extension is added to this filename.
Macro Record Mode	Specifies whether or not you want to record as a macro. To be replayed, a macro requires a starting point.
Browse	Displays Script browser for selecting a macro or script to be replayed.
Library	Displays the Select Script to Replay dialog box, where you can select from a library of predefined scripts.
Record	Starts recording your actions.
Stop	Stops recording your actions or stops replaying a script.
Replay	Starts replaying a macro or script.
Help	Launches the SigXplorer Help system and displays the relevant Help topic.
Cancel	Ignores input and closes the dialog box.

Procedures

Recording a script

1. Choose *File – Script*.

The Scripting dialog box appears.

2. In the File box, enter the name you want to give to the script.

SigXplorer automatically appends the `.scr` file extension.

3. Click Record.

The Scripting dialog box disappears.

4. Perform the tasks that you want the script to run.

5. Choose *File – Script*, then click *Stop*.

SigXplorer saves the script file. To verify that the script is saved properly, click Library and look for the file name in the list.

Recording a macro

1. Choose *File – Script*.

The Scripting dialog box appears.

2. In the File text box, enter the name you want to give to the macro.

SigXplorer automatically appends the `.scr` file extension.

3. Enable Macro Record Mode.

4. Click *Record*.

The Scripting dialog box disappears.

5. Perform the tasks that you want the macro to run.

6. Choose *File – Script*, then click *Stop*.

SigXplorer saves the macro file. To verify that the macro is saved properly, click *Library* and look for the file name in the list.

Replaying a script

1. Choose *File – Script*.

The Scripting dialog box appears.

2. In the File box, enter the name of the script you want to replay.

If necessary, click Browse or Library to locate the correct file.

3. Click *Replay*.

SigXplorer replays the script.

Replaying a macro

1. Choose *File – Script*.

The Scripting dialog box appears.

2. In the File box, enter the file name of the macro you want to replay.

If necessary, click Browse or Library to locate the correct file.

3. Click *Replay*.

SigXplorer replays the macro.

File – Recent Topologies

Use this command to open one of the last ten saved topologies.

Procedure

To open a recently saved topology

Choose *File – Recent Topologies*, and select a topology from the pull-right menu.

Note: Moving a saved topology to a different location, using the operating system's file manager, results in an broken connection between the topology and its entry in the *Recent Topologies* menu; the menu entry will be removed the next time you click it.

File – Exit

Exits SigXplorer. A pop-up asks if you want to save your work before exiting.

Allegro SI SigXplorer Reference

File Menu Commands

Edit Menu Commands

Edit – Undo

Reverses edits made to the topology canvas. Edits are undone, one at a time, in reverse order.

When there are no edits left to undo, the Undo toolbar button and Undo on the Edit menu are both grayed out and cannot be selected.

Procedures

Undoing edit operations

Perform one of the following:

- Select an element on the canvas and choose *Edit – Undo*.

-or-

- Enter CTRL+Z

Each time you use Undo, another edit operation is undone. The Undo button is disabled (grayed out) when there are no more changes to undo for this editing session.

Toolbar Icon



Edit – Move

Use this command to relocate an element on the canvas.

Procedure

To move an element on the canvas

Do one of the following:

- The most-direct method is to click on an element, or a group of elements, and drag the element to a different location on the canvas.
-or-
- Select an element, or a group of elements, on the canvas and choose *Edit – Move*.
-or-
- Select an element, or a group of elements, on the canvas, right-click and choose *Move* from the pop-up menu.
-or-
- Select an element, or a group of elements, on the canvas and enter `SHIFT+F6`.
-or-
- Select an element, or a group of elements, on the canvas and click the move icon.
The element attaches to the mouse pointer, ready to be anchored at its new location where you click on the canvas.

Toolbar Icon



Edit – Copy

Use this command to duplicate an existing element on the canvas, at a new location.

Procedure

To duplicate an element on the canvas

Do one of the following:

- Select an element, or a group of elements, on the canvas and choose *Edit – Copy*.
-or-
- Select an element, or a group of elements, on the canvas, right-click and choose *Copy* from the pop-up menu.
-or-
- Select an element, or a group of elements, on the canvas and enter `SHIFT+F5`.
-or-
- Select an element, or a group of elements, on the canvas, and then press `CTRL` and drag out a duplicate.

The duplicate element attaches to the mouse pointer, ready to be anchored at its new location where you click on the canvas.

Toolbar Icon



Edit – Delete

Use this command to remove an existing element from the canvas.

Procedure

To remove an element from the canvas

Perform one of the following:

- Select an element, or a group of elements, on the canvas and choose *Edit – Delete*.
-or-
- Select an element, or a group of elements, on the canvas, right-click and choose *Delete* from the pop-up menu.
-or-
- Select an element, or a group of elements, on the canvas and enter `SHIFT+D`.
-or-
- Select an element, or a group of elements, on the canvas and click the delete icon.

The element attaches to the mouse pointer, ready to be anchored at its new location where you click on the canvas.

Toolbar Icon



Edit – Rotate Right

Use this command to rotate an existing element on the canvas clockwise.

Note: An element can be rotated only 90° clockwise (or counter-clockwise). Once rotated, the only option is to rotate in the opposite direction.

Procedure

To rotate an element on the canvas clockwise

Perform one of the following:

- Select an element, or a group of elements, on the canvas and choose *Edit – Rotate Right*.
-or-
- Select an element, or a group of elements, on the canvas, right-click and choose *Rotate Right* from the pop-up menu.
-or-
- Select an element, or a group of elements, on the canvas and press `Alt+e+r` followed by `Enter`.
-or-
- Select an element, or a group of elements, on the canvas and click the rotate right icon.

Toolbar Icon



Edit – Rotate Left

Use this command to rotate an existing element on the canvas counter-clockwise.

Note: An element can be rotated only 90° counter-clockwise (or clockwise). Once rotated, the only option is to rotate in the opposite direction.

Procedure

To rotate an element on the canvas counter-clockwise

Perform one of the following:

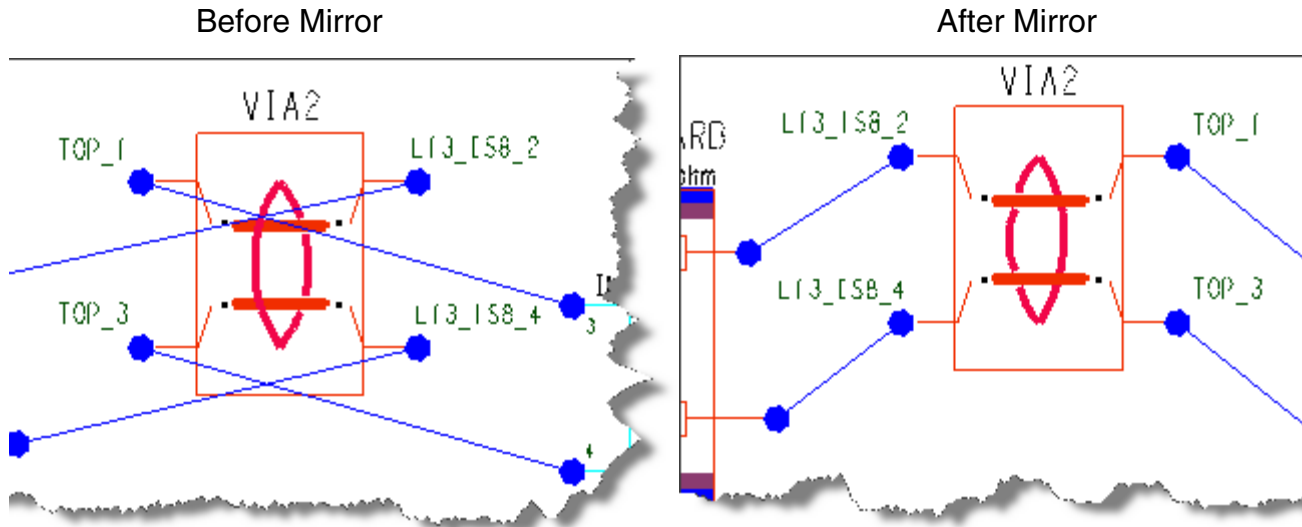
- Select an element, or a group of elements, on the canvas and choose *Edit – Rotate Left*.
-or-
- Select an element, or a group of elements, on the canvas, right-click and choose *Rotate Left* from the pop-up menu.
-or-
- Select an element, or a group of elements, on the canvas and press `Alt+e+l` followed by `Enter`.
-or-
- Select an element, or a group of elements, on the canvas and click the rotate left icon.

Toolbar Icon



Edit – Mirror

Use this command to mirror the selected object, or group of objects, on the canvas.



Procedure

To mirror an element on the canvas

Perform one of the following:

- Select an element, or a group of elements, on the canvas and choose *Edit – Mirror*.
-or-
- Select an element, or a group of elements, on the canvas, right-click and choose *Mirror* from the pop-up menu.
-or-
- Select an element, or a group of elements, on the canvas and press **Alt+e+m** followed by **Enter**.
-or-
- Select an element, or a group of elements, on the canvas and click the mirror icon.

Toolbar Icon

Allegro SI SigXplorer Reference

Edit Menu Commands



Edit – Add Element

[Dialog Box](#) | [Procedures](#)

Displays the *Add Element Browser*. Use this dialog box to select and add new topology elements to the topology canvas.

Dialog Box

Option	Description
Model Type Filter field	Displays the model types available for selection.
Model Name Pattern field	Displays the model name to match in the library search.
Format	Available only with Via Model Type selected, displays a selection of via format models.
Type	Available only with Via Model Type selected, displays a selection of single and coupled via types.
Model Name listing	Displays the models names associated with the Model Name Pattern selection.
Model Type listing	Displays the model types associated with the Model Type Filter selection.
Library	Displays the path to the library.

The Add Element Browser Filters

Filter fields at the top of the Model Browser control which part models display in the list box.

You can specify models by:

- Model type
- Characters in the model name

Displaying Models by Model Type

Use the Model Type Filter field to display a pull down menu listing the available types of topology element part models.

SigXplorer provides the following elements:

Option	Description
EspiceDevice	Displays EspiceDevice models.
GenericElement	Displays the following generic part models: <ul style="list-style-type: none">■ Capacitor■ Connector■ Current Probe■ Diode■ Inductor■ Resistor■ Source
AnyIOCell	Displays all IBIS IOCell models regardless of type.
DifferentialPart	Displays differential pair IOCell models: <ul style="list-style-type: none">■ DiffIO■ DiffInput■ DiffOutput■ DiffDummyProbe
IbisDevice	Displays IbisDevice models.
IbisInput	Displays IbisInput IOCell models.
IbisIO	Displays IbisIO IOCell models.
IbisIO_OpenPullUp	Displays IbisIO_OpenPullUp IOCell models.
IbisIO_OpenPullDown	Displays IbisIO_OpenPullDown IOCell models.
IbisOutput	Displays IbisOutput IOCell models.
IbisOutput_OpenPullUp	Displays IbisOutput_OpenPullUp IOCell models.

Allegro SI SigXplorer Reference

Edit Menu Commands

IbisOutput_OpenPullDown	Displays IbisOutput_ OpenPullDown IOCell models.
Interconnect	Displays all interconnect models of the following types: <ul style="list-style-type: none">■ Tline■ Trace■ Coupled Trace
Termination	Displays all termination models of the following types: <ul style="list-style-type: none">■ DualClamp■ HiClamp■ LowClamp■ RC■ Series■ Shunt■ Thevenin
Via	Displays all via models of the following formats and types: Formats: <ul style="list-style-type: none">■ Closed-Form■ Narrow-Band■ Wide-Band■ S-Parameter Types: <ul style="list-style-type: none">■ Single■ EdgeCoupled■ BroadsideCoupled■ CoupledWithGND■ CoupledWithPWR

Displaying Models by Name Pattern

Use the Model Name Pattern field to list topology element part models whose names match a designated character pattern. To filter by model name pattern, enter a wildcard (*) with part of the model name character string. An asterisk (*) alone shows all models matching the specified library and model type filters.

All models whose names include the character string appear in the list box. The rules for regular expressions are used by the UNIX grep command *apply*, or use a plain character string to match model names. For example, the string 74ACT matches all model names that contain 74ACT.

Model Name Pattern

Lists models whose names match a designated character pattern. To filter by model name pattern, enter a wildcard (*) with part of the model name character string. An asterisk (*) alone shows all models matching the specified library and model type filters.

Option	Description
Model Name/Model Type	Displays the model name and model types available for selection.
Library	Displays the path to the library.

Use the *Add Element Browser* to display a list of the topology element part models available to add as you create and modify the circuit topology.

Adjust the filter fields to display different groups of models, for example, IOCell, buffer models or models for generic parts. Use the ModelType Filter field to display selected groups of models. Use the Model Name Pattern field to display groups of models by name.

Procedures

Displaying topology element models

1. Perform one of the following:

- ☐ Choose *Edit – Add Element*.
- or-
- ☐ Choose *Add Element* from the pop-up menu.
- or-
- ☐ Press `Alt+e+a` followed by `Enter`.
- or-
- ☐ Click the add element icon.



The Add Element Browser dialog box appears.

2. Select a filter type from the Model Type Filter drop down list.

Models of the selected type are displayed in the Model Name list box.

3. Specify model names to display in the Model Name pattern text box.

- ☐ To display all models of the selected type, leave the filter set to * (asterisk).
- ☐ To display a selected set of model names, enter a character string, including wildcard characters.

4. If you are adding a via part model, select a format and type from the Format and Type drop-down lists.

The selected topology element part models display in the list box.

5. Click *OK* to close the Model Browser.

Selecting and place topology element models

1. Choose *Edit – Add Element*.
2. Display the appropriate element part models as described in the previous procedure.

The Add Element Browser dialog box appears.

3. Click a topology element in the list box.

The outline of the element symbol attaches to the cursor and moves with the cursor over the topology canvas.

If the topology element you are adding represents a logical change that will disable the updating of the template to a design database Xnet, a confirmation dialog box appears.

- ☐ Click *Yes* if you want to see the confirmation dialog box every time you select an element that disables updating of the topology to the design database Xnet.
- ☐ Click *No* to suppress the same confirmer from popping up again.

Note: If you selected an IbisDevice model, the Select IBIS Device Pin dialog box opens, prompting you to select a pin in the IBIS Device model.

4. To place the selected topology element, click the cursor at the desired location.

A symbol for the topology element is created and highlighted in the topology canvas. The element outline remains attached to the cursor for further placement. Each time you click, a new instance of the element is created. As each instance of the element is placed, the new symbol is highlighted, and the previous instance is dehighlighted. When you have placed the last instance of the element, click a second time in the same location. Placement of that part is terminated and the outline is no longer attached to the cursor.

5. To place a different symbol, adjust the filter as desired to display part models for a different topology element. In the list box, select a new topology element.

The previous element outline disappears, and the new outline is attached to the cursor for placement.

6. Repeat Steps 3 and 4 until you have placed all the topology symbols you need.
7. Click *OK* to close the Add Element Browser.

Adding a Clocked IOCell MacroModel

1. Choose *Edit – Add Element*.

The *Add Element Browser* appears.

2. From the pull-down menu, choose IbisIO.
3. Select the desired IOCell model and drag it to the Topology Canvas for placement. For example, CDSDefaultIO_CLK from the Standard Cadence Library.
4. Click *OK*.

Editing a Clocked IOCell MacroModel

For the driver in a differential pair, you can edit

- RefDes (or part name)
- Setup and hold times and sample period
- Stimulus parameters for clock, data, and enable signals

Modifying stimulus parameters

1. In the canvas, click the stimulus associated with the IbisIO part symbol. For example, the stimulus might be Pulse or Tristate.

The IOCell Stimulus Editor opens for the IbisIO with the current stimulus data displayed in the data fields.

2. In the IOCell Stimulus Editor, make the appropriate edits to the clock, data, and enable signals.
 3. Click *Apply* to apply the settings and retain the IOCell Stimulus Editor
- or -

Click *OK* to apply the settings and dismiss the IOCell Stimulus Editor.

Modifying the RefDes associated with the IbisIO

1. In the canvas, click the RefDes, or part name, associated with IbisIO symbol.

For example, the Parameters tab opens with the data for the selected IbisIO expanded and the RefDes in the Name column highlighted for editing.

2. Enter the new RefDes and click *Enter*.

The new RefDes replaces the old in both the Name column of the spreadsheet and with the part symbol in the canvas.

Modifying setup and hold times and the sample period

1. In the canvas, click the stimulus associated with the IbisIO part symbol. For example, the stimulus might be Pulse or Tristate. The IOCell Stimulus Editor opens for the IbisIO with the current stimulus data displayed in the data fields.
2. In the Measurement Info area of the IOCell Stimulus Editor, edit the Setup and Hold times and the Measurement Cycle. See About Modifying the Stimulus for an IOCell for more information.
3. Click *Apply* to apply the settings and retain the IOCell Stimulus Editor.

- or -

Click *OK* to apply the settings and dismiss the IOCell Stimulus Editor.

Simulating a Clocked IOCell MacroModel

- Choose *Analyze – Simulate* to start the simulation.

During the simulation, messages display in the Command tab. When the simulation is complete, the Results tab displays the simulation result data. The SigWave window opens to display the differential waveforms.

Edit – Note

[Dialog Box](#) | [Procedures](#)

Displays the Note – Editing dialog box for creating notes and attaching them to the topology. You can also change, move, and delete notes.

Dialog Box

Option	Description
Modify	Modifies a selected note string.
Move	Moves a selected note string.
Delete	Deletes a selected note string.
Create Note Button	Creates a new note string.

Procedures

Adding a note to a topology

1. Choose *Edit – Note*.
-or-
2. Click-right on the canvas and choose *Note* from the pop-up menu.
-or-
3. Click the Note icon



The *Note – Editing* dialog box appears.

4. Click *Modify*, if necessary. (The default setting is *Checked*.)
5. Click *Create Note* to open the default text editor window.

6. In the text editor window, type your note.

7. Exit the text editor.

The text editor will prompt you to save the note. When you answer Yes, the note string is saved to your cursor and not to a file.

8. Move the cursor to the topology canvas and click to place the note at the desired location.

A copy of the note appears at the location on the topology canvas. The original note remains attached to the cursor, allowing you to copy it to different locations.

9. Click *Apply* to apply the note and continue, or *OK* to exit.

Note Editing

Choose *Edit – Note* and use the Note – Editing dialog box to create and modify text notes in the Topology Canvas. Notes are similar to paper stick on notes that you can attach and easily move, or throw away when you no longer need them.

Modifying, Moving, and Deleting Existing Notes

Modifying an existing note

1. Choose *Edit – Note*.

The Note – Editing dialog box appears.

2. Click *Modify*.

3. In the Topology Canvas, click to select the note to change.

4. A text editor opens (Specify the text editor using the EDITOR environment variable).

5. In the text editor window, edit your note and exit the text editor.

Moving a note

1. Choose *Edit – Note*.

The Note – Editing dialog box appears.

2. Click *Move*.

3. In the Topology Canvas, click to select the note to move.

The note attaches itself to the cursor.

4. Move the note to the new location in the Topology Canvas and click.

The note is moved to the new location.

Note: When you use *Edit – Cleanup*, all notes are re-oriented in columns at the right side of the topology.

Deleting a note

1. Use *Edit – Note*.

The Note – Editing dialog box appears.

2. Click *Delete*.

3. In the Topology Canvas, select the note to delete.

The note is deleted.

Edit – Transform for Constraint Manager

Conditions a topology for use in Constraint Manager.

SigXplorer conditions a circuit for use in Constraint Manager by simplifying complex topologies and replacing trace elements with ideal transmission line elements and replacing vias with T-points.

SigXplorer simplifies a complex topology by converting:

1. sweep parameters to a single parameter. SigXplorer uses the average value of the sweepable range.
2. multi-line trace elements to single-line trace element. SigXplorer retains the original cross-section geometry based on the line location within the original multi-trace cross-section.
3. single-line trace elements to single-line ideal transmission lines.
4. vias to connections. SigXplorer adds T-points to connections to two or more transmission lines.
5. all single-line transmission line elements along a contiguous path to a single transmission line element, as long as the impedance and velocity are the same. SigXplorer accumulates propagation delay and length values.

If an element (trace, transmission line, or via) has parameters used by an expression of another element (IOCell, Discrete, etc.,) the transformation will fail.

Procedure

Conditioning a complex topology for use in Constraint Manager

- Choose *Edit – Transform for Constraint Manager*.

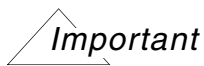
The SigXplorer canvas displays a simplified topology conditioned for use in Constraint Manager.

Edit – Cleanup

Use this command to rearrange the elements on the canvas.

Use *Edit – Cleanup* to align the parts on the topology canvas. When you use the Cleanup command, the single-segment Xnet topology elements are autopaced within the topology canvas in a logical left-to-right, top-to-bottom flow. All notes are reoriented in columns at the left side of the topology canvas.

You can use the Cleanup command at any time during a topology editing session to rearrange topology elements. Generally, the best time to use Cleanup is after editing the topology.



Cleanup works only with single Xnet topologies or with differential pair topologies.

Procedure

Perform one of the following:

- Choose *Edit – Cleanup*.
-or-
- Press **Alt+e+l** followed by **Enter**.
-or-
- Click the add element icon.

Toolbar Icon



Allegro SI SigXplorer Reference

Edit Menu Commands

View Menu Commands

View – Zoom By Window

Magnifies the display so that the specified, bounded region fills the topology canvas.

Procedures

Zooming in by defining a new window

1. Click and hold the left mouse button to stretch a bounding rectangle that defines the area to be magnified.
2. Release the left mouse button when you have defined the zoom area.

The new, magnified window area you defined is displayed in the topology canvas.

View – Zoom Fit

Changes the display so that the topology fills the canvas.

View – Zoom Center

Changes the display so that the topology is left justified and centered vertically within the canvas.

View – Zoom In

Magnifies the topology to make it larger and display less of it in the canvas.

View – Zoom Out

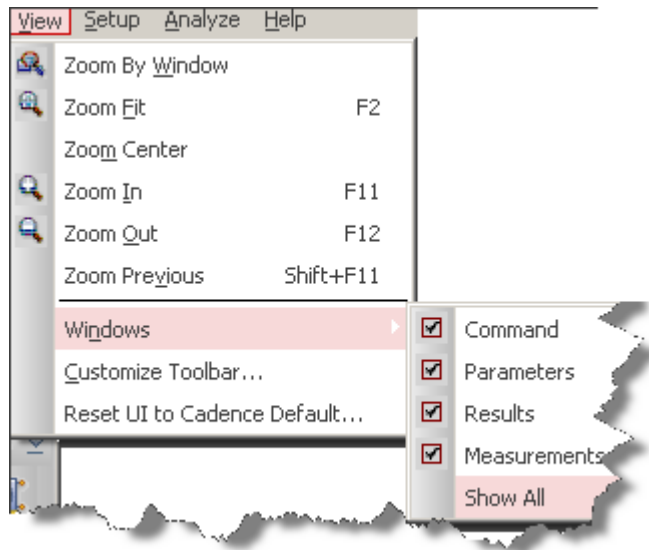
Shrinks the topology to display more of it in the canvas.

View – Zoom Previous

Displays the previous zoomed *view* of the topology on the canvas.

View – Windows

Use this command to show (or hide) the windows that are associated with the *Command*, *Parameters*, *Results*, or *Measurements* tabs.



View – Customize Toolbar

View – Reset UI to Cadence default

Allegro SI SigXplorer Reference
View Menu Commands

Setup Menu Commands

Setup – Constraints

[Dialog Box](#) | [Procedures](#)

Displays the *Set Topology Constraints* dialog box for modifying topology constraint values. These modified values are written back to the Constraint Manager when you execute the *File – Update* command.

Important

Since all old constraints are defined and meaningful on one xnet, while a differential topology can contain two separate xnets, SigXplorer version 15.0 will not allow the single-xnet constraint defined between pins on different xnets.

Set Topology Constraints Dialog Box

The Set Topology Constraints dialog box consists of 10 tabs:

[Switch-Settle tab](#)

[Prop Delay tab](#)

[Impedance tab](#)

[Rel Prop Delay tab](#)

[Diff Pair tab](#)

[Max Parallel tab](#)

[Wiring tab](#)

[User-Defined tab](#)

[Signal Integrity tab](#)

[Usage tab](#)

Allegro SI SigXplorer Reference

Setup Menu Commands

Switch-Settle tab

Use this tab to create and modify switch and settle delay rules between driver and receiver pin pairs. You can also create switch and settle delay rules that apply to all driver and receiver pin pairs in the topology.

Option	Description
<i>Existing Rules</i>	Lists Driver, Receiver, Min Switch Rise and Fall, Max Settle Rise and Fall values that are applied to the existing rules for the topology.
<i>Pins</i>	
<i>Name</i>	Pin names in the topology. Also includes the choice: All DRVRS/RCVRS.
<i>Usage</i>	Pin type assigned to pin.
<i>Rule Editing</i>	
<i>Driver</i>	Driver for new or selected rule, selected from pin list.
<i>Receiver</i>	Receiver for new or selected rule, selected from pin list.
<i>Min First Switch Delays</i>	Minimum allowable first switch delays for the paths. The constraint may be defined with two values which represent the budget for the Rising and Falling edges.
<i>Max Final Settle Delays</i>	Maximum allowable final settle delays for the paths. The constraint may be defined with two values which represent the budget for both the Rising and Falling edges.
<i>Add</i>	Adds the rule defined in Rule Editing to the Existing Rules list.
<i>Modify</i>	Modifies the rules selected in the Existing Rules list according to the changes shown in Rules Editing.
<i>Delete</i>	Deletes the rules selected in the Existing Rules list.
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Prop Delay tab

Use this tab to create and modify delay rules for pin, tee or pin-tee pairs. You can also create delay rules for all drivers and receivers, all driver and receiver pairs, and the longest and shortest Tlines.

Option	Description
<i>Existing Rules</i>	Lists the From (start) and To (end) pins or T-points, Rule-Type, Min-Delay and Max-Delay that are applied to the existing rules for the topology.
<i>Pins/Tees</i>	
<i>Name</i>	Names (reference designators) for pins and T-points in the topology. Also includes the choices: All DRVRS/RCVRS, DRIVER/RECEIVER, and LONGEST/SHORTEST.
<i>Usage</i>	Pin type assigned to pin. TEE indicates a T-point.
<i>Rule Editing</i>	
<i>From</i>	Start pin or T-point for the new or selected delay rule, selected from the Pins/Tees list.
<i>To</i>	End pin or T-point for the new or selected delay rule, selected from the Pins/Tees list.
<i>Rule Type</i>	Type of delay rule. Select Delay, Length, or %Manhattan from the drop-down list.
<i>Min Delay</i>	Minimum allowable propagation delay/length for the pin pairs.
<i>Max Delay</i>	Maximum allowable propagation delay/length for the pin pairs.
<i>Add</i>	Adds the rule defined in Rule Editing to the Existing Rules list.
<i>Modify</i>	Modifies the rules selected in the Existing Rules list according to the changes shown in Rules Editing.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Delete</i>	Deletes the rules selected in the Existing Rules list.
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Impedance tab

Use this tab to create and modify impedance rules for pin, tee or pin-tee pairs. You can also create delay rules that apply to all pin, tee or pin-tee pairs in the topology.

Impedance rules specify a baseline impedance value and an allowable delta value above and below the baseline.

Option	Description
<i>Existing Rules</i>	Lists From (start) and To (end) pins or T-points, Target, Type, and Tolerance that are applied to the existing rules for the topology.
<i>Pins/Tees</i>	
<i>Name</i>	Names (reference designators) for pins and T-points in the topology. Also includes the choice: All/All.
<i>Usage</i>	TEE indicates a T-point. T-point is indicated with "T.<pin number>".
<i>Rule Editing</i>	
<i>From</i>	Start pin or T-point for the new or selected impedance rule, selected from the Pins/Tees list.
<i>To</i>	End pin or T-point for the new or selected impedance rule, selected from the Pins/Tees list.
<i>Target</i>	Impedance target value.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Type</i>	Type of impedance rule. Select Ohms or %Ohms from the drop-down list.
<i>Tolerance</i>	Impedance tolerance. Enter a delta value above and below the baseline impedance. Express the tolerance as an absolute value or a percentage. To capture a percentage, simply include the % after the value.
<i>Add</i>	Adds the rule defined in Rule Editing to the Existing Rules list.
<i>Modify</i>	Modifies the rules selected in the Existing Rules list according to the changes shown in Rules Editing.
<i>Delete</i>	Deletes the rules selected in the Existing Rules list.
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Rel Prop Delay tab

Use this tab to assign matched interconnect delay constraint rules to pin, tee or pin-tee pairs. A matched delay constraint is two or more pin, tee or pin-tee pairs whose interconnect delay must be within a specified tolerance. You can assign matched delay rules to a single pair or to groups of pairs.

Option	Description
<i>Existing Rules</i>	Lists the Name, From (start) and To (end) pins or T-points, Scope, Delta, and Tolerance that are applied to the existing rules for the topology.
<i>Pins/Tees</i>	
<i>Name</i>	Names (reference designators) for pins and T-points in the topology. Also includes the choices: All DRVRS/RCVRS, DRIVER/RECEIVER, and LONGEST.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Usage</i>	TEE indicates a T-point. T-point is indicated with "T.<pin number>".
<i>Rule Editing</i>	
<i>Rule Name</i>	Name for a new or selected rule.
<i>From</i>	Start pin or T-point for the new or selected delay rule, selected from the Pins/Tees list.
<i>To</i>	End pin or T-point for the new or selected delay rule, selected from the Pins/Tees list.
<i>Scope</i>	<p>Controls how the members of the Group are validated. Select one of the following from the drop-down list:</p> <ul style="list-style-type: none"><input type="checkbox"/> <i>Local</i> - Creates a single match group. Checking is done only between the two pin pairs of each net, and limited to within the net. <i>Example:</i> Multiple nets with a single driver, two receivers, and a branch point where the length from the driver to each receiver must match, but no matching is needed to other nets.<input type="checkbox"/> <i>Global</i> - Creates a single match group, derived from net properties or an electrical constraint set (ECSet). The same match group can exist in multiple ECSets or properties, resulting in all objects ending up in the same match group. For hierarchical designs, use of the Global Scope in lower blocks creates a single merged match group at the top level. <i>Example:</i> Multiple nets containing pin pairs that must match to each other across each net.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Scope (con.t')</i>	<ul style="list-style-type: none">❑ <i>Bus</i> - Creates match groups based on bus names (such as MG1_BUS1, MG1_BUS2, and so on). You can apply a single ECSet to all the nets at either the net or bus level. This group type reduces the number of ECSets required to constrain a design, as opposed to requiring a separate ECSet for each bus. A limitation of this scope type is that no other signals from outside the bus can be added to these match groups. <i>Example:</i> Multiple nets organized in several buses. Pin pairs must match, but only to nets within the same bus. Typically, all nets share the same topology.❑ <i>Class</i> - Generates unique match groups for each class. Similar to Bus scope, Class scope also optimizes the number of topologies required to constraint a design. However, no other signal from outside the net class can be added to a match group with Class scope. Class scope has more flexibility than Bus scope because a class can include more signals than a bus, which is typically limited to vectored nets or nets that share a common topology. Unlike the Bus scope, the Class scope adds all the selected members, including bus members, to the match group created by the ECSet (with Class scope). <i>Example:</i> When you need the functionality of a Bus scope and also need additional non-bus members in a match group, use the Class scope.
<i>Delta Type</i>	Type for the specified delta. Select <i>Delay</i> , <i>Length</i> , or <i>None</i> from the drop-down list.
<i>Delta</i>	Allowable propagation delay/length delta for pin, tee or pin-tee pairs of the group. The delta is used to offset the pair(s) from a target pair.
<i>Tol Type</i>	Type for the specified tolerance. Select <i>Delay</i> , <i>Length</i> , or <i>Percent</i> from the drop-down list.
<i>Tolerance</i>	Relaxes the relative/match requirements.
<i>New</i>	Creates an empty rule set that you can name and define.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Add</i>	Creates a new rule based on the parameters defined in Rule Editing.
<i>Modify</i>	Modifies the rules selected in the Existing Rules list according to the changes shown in Rules Editing.
<i>Delete</i>	Deletes the rules selected in the Existing Rules list.
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Diff Pair tab

Use this tab to create and modify differential pair rules.

Option	Description
<i>Primary Gap</i>	Enter a value for the ideal edge-to-edge spacing between the pair that should be maintained for the entire length of the pair. Values you set for Neck Gap override Primary Gap values in areas that need smaller gaps to get through dense components.
<i>Line Width</i>	Enter a value for the minimum width of each member of the diff pair.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Neck Gap</i>	<p>Enter a value for the edge-to-edge spacing between a pair as it goes through tight areas full of component pins and vias.</p> <p>Neck Gap overrides any value in the Primary Gap when the differential pair's spacing collapses to or below the value of the <i>Neck Width</i>.</p> <ul style="list-style-type: none">■ Ensure that the neck gap does not go below any <i>Minimum line spacing</i> value you have set.■ You do not need to define a neck gap if you set <i>(-) Tolerance</i> with a value that accounts for the needed neck gap.
<i>Neck Width</i>	<p>Enter a value for the width of each line in a diff pair as it goes through confined areas among densely placed components.</p>
<i>Coupled Tolerance (+)</i>	<p><i>Enter a (+) Tolerance</i> value to define a band around the primary gap in which the lines of a pair can go beyond the primary gap value.</p> <p>Note: The lines are considered coupled when they are within the band specified by the <i>(+) Tolerance</i> and outside the band specified by the <i>(-) Tolerance</i>.</p>
<i>Coupled Tolerance (-)</i>	<p><i>Enter a (-) Tolerance</i> value to define a band around the primary gap in which the lines of a pair can go closer than the the primary gap value.</p> <p>Note: The lines are considered coupled when they are within the band specified by the <i>(+) Tolerance</i> and outside the band specified by the <i>(-) Tolerance</i>.</p>
<i>Minimum line spacing</i>	<p>Enter a value to constrain the distance between any two segments from each Xnet member of the diff pair. The value you enter must be less than or equal to the <i>separation</i>, minus the negative tolerance. The value must also be greater than or equal to the <i>neck gap</i> value.</p>
<i>Gather control</i>	<p>Indicates whether the line segments that diverge and converge, as the pair of nets go from driver to receiver, should be included (<i>include</i>) or excluded (<i>ignore</i>) from the uncoupled length.</p>
<i>Max uncoupled length</i>	<p>Enter the maximum allowable uncoupled length.</p>

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Phase control</i>	Choose <i>Static</i> from the pull-down menu to enable static phase control. This option checks the differential pair for mismatched length tolerance only on the overall lengths.
<i>Phase Tolerance</i>	Enter a tolerance value (in <i>time</i> or <i>length</i>) to specify a separation to which phase matching is maintained.
<i>Type</i>	Select a value by which to measure the parameters you have set in the other fields, either <i>Delay</i> or <i>Measure</i> .
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Max Parallel tab

Use this tab to assign maximum parallel routing constraint rules to signals.

Option	Description
<i>Existing Rules</i>	
<i>Coupled Length</i>	Allowable length for selected signals to run parallel.
<i>Gap</i>	Allowable gap between selected signals running parallel.
<i>Rule Editing</i>	
<i>Length</i>	Allowable length for selected signals to run parallel.
<i>Gap</i>	Allowable gap between selected signals running parallel.
<i>Add</i>	Adds the rule defined in Rule Editing to the Existing Rules list.
<i>Modify</i>	Modifies the rules selected in the Existing Rules list according to the changes shown in Rules Editing.
<i>Delete</i>	Deletes the rules selected in the Existing Rules list.
<i>OK</i>	Exits the dialog box and saves any changes you have made.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Wiring tab

Use this tab to create and modify topology scheduling parameters as well as physical and EMI (electromagnetic interference) constraint rules. These rules apply to the topology as a whole. They are not associated with specific topology elements.

Option	Description
<i>Topology</i>	
<i>Mapping Mode</i>	<p>Select one of the pre-defined mapping modes for the ECSet from the drop-down list. The Mapping Mode is used when the ECSet schedule is applied to Xnets/Nets.</p> <ul style="list-style-type: none"><input type="checkbox"/> Pinuse: Maps the pins of an ECSet to the XNet using the PINUSE setting.<input type="checkbox"/> Refdes: Maps the pins of an ECSet to the XNet using the RefDes setting.<input type="checkbox"/> Pinuse and Refdes: Employs both mapping techniques described above.<input type="checkbox"/> (Clear): No specified mapping mode. <p>For information on how the mapping modes resolve the mapping of pins for creation of the topology, refer to <u>Mapping Modes</u> in Allegro Constraint Manager Reference guide.</p>

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Schedule</i>	<p>Select one of the pre-defined schedules for the ECSet from the drop-down list.</p> <ul style="list-style-type: none">❑ Minimum Spanning Tree: Connects all of the pins together with minimum connection length. Any pin can connect to any number of other pins. This schedule starts at the primary driver, selects the closest pin to this driver, and connects it through a TLine. The search continues by selecting the next unscheduled pin that is closest to any of the scheduled pins and connecting it with a TLine to the closest scheduled pin. This continues until all pins are scheduled.❑ Daisychain: Connects the pins of the topology with minimum connection length, allowing each pin to connect to a maximum of two other pins. This schedule starts with the primary driver, selects the closest pin to this driver, and connects it with a TLine. The closest pin to the last pin scheduled is then selected and connected with a TLine. This continues until all of the pins are scheduled.❑ Source-load Daisy-chain: Connects similar to a daisy chain schedule except that all driver pins are scheduled first, followed by all receiver pins.❑ Star: Connects the driver pins in a daisy-chain pattern, then all of the receiver pins are connected to the last driver pin.❑ Far-end Cluster: Connects similar to a star schedule except that the last driver pin connects to a T-point, to which all of the receivers are connected.❑ Template: Connects according to a user-defined template schedule. You are required to interactively add and connect each T-line to form the custom net schedule.❑ (Clear): No specified schedule

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Verify Schedule</i>	Verify Schedule is used to enable design rule checks (DRC) if a schedule has been set. Select one of the following from the drop-down list: <ul style="list-style-type: none"><input type="checkbox"/> <i>Yes</i>: Enables DRC<input type="checkbox"/> <i>No</i>: Disables DRC<input type="checkbox"/> (<i>Clear</i>): No specified DRC.
<i>Physical</i>	
<i>Stub Length</i>	Stub length for daisy chain routing.
<i>Max Via Count</i>	Maximum number of vias allowed in a net.
<i>Total Trace Length</i>	Minimum and maximum trace lengths allowed in an Xnet.
<i>Layer Sets</i>	Define and/or edit the layer set data for the current topology.
<i>EMI</i>	
<i>Max Exposed Length</i>	Maximum length of interconnect allowed in a net that is not shielded by plane layers above and below.
<i>Current Exposed Length</i>	Displays the current exposed length value.
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Allegro SI SigXplorer Reference

Setup Menu Commands

User-Defined tab

Use this tab to add your own custom constraints to the topology. You can use this feature to store other supplementary constraints within a topology for later use.

Option

Description

Existing Rules

Lists the Name, Type, and Value applied to existing user-defined constraints for the topology.

Rule Editing

Name

Name for a new or selected rule.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Type</i>	<p>Type for a new or selected rule. String is the default setting. Select one of the following types from the drop-down list:</p> <p><i>Altitude</i></p> <p><i>Capacitance</i></p> <p><i>Design Units</i></p> <p><i>Distance</i></p> <p><i>Electrical Conductivity</i></p> <p><i>Failure Rate</i></p> <p><i>Impedance</i></p> <p><i>Inductance</i></p> <p><i>Integer</i></p> <p><i>Layer Thickness</i></p> <p><i>Noise Voltage</i></p> <p><i>Propagation Delay</i></p> <p><i>Real</i></p> <p><i>Resistance</i></p> <p><i>String</i></p> <p><i>Temperature</i></p> <p><i>Thermal Conductance</i></p> <p><i>Thermal Conductivity</i></p> <p><i>Velocity</i></p> <p><i>Voltage</i></p>
<i>Range</i>	Minimum and maximum values for applying the rule.
<i>Units</i>	The units of measure determined by the rule type.
	Note: <i>Range</i> and <i>Units</i> will appear in the dialog box for all rule types except <i>String</i> .

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Value (optional)</i>	Optional value for a new or selected rule.
<i>Add</i>	Adds the rule defined in Rule Editing to the Existing Rules list.
<i>Modify</i>	Modifies the rules selected in the Existing Rules list according to the changes shown in Rules Editing.
<i>Delete</i>	Deletes the rules selected in the Existing Rules list.
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Signal Integrity tab

Use this tab to create and modify crosstalk, noise, and physical constraint rules. These rules apply to the topology as a whole. They are not associated with specific topology elements.

Option	Description
<i>Reflection</i>	
<i>Overshoot</i>	Maximum overshoot value allowed, in mV, for both the rising edge (High State) and falling edge (Low State).
<i>Min Noise Margin</i>	Minimum allowable delta between the switching threshold and the receiver waveform, after the waveform has crossed the threshold but before the onset of a transition that crosses both thresholds. The constraint is expressed as two values for rising (High State) and falling (Low State) transitions.
<i>Edge Distortions</i>	

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Edge Sensitivity</i>	<p>Indicates whether or not a net is sensitive to non-monotonicity in the receiver waveform. If the constraint is not set, the net is insensitive. If the constraint is set, select one of the following values from the drop-down list:</p> <ul style="list-style-type: none"><input type="checkbox"/> Rising: Only the rising edge is sensitive.<input type="checkbox"/> Falling: Only the falling edge is sensitive.<input type="checkbox"/> Both: Both edges are sensitive.<input type="checkbox"/> Neither: Neither edge is sensitive. <p>(Clear): No constraint.</p>
<i>First Incident Switch</i>	<p>Indicates whether the receiver of a driver/receiver pin pair is required to switch on the first incident wave. If the constraint is set, select one of the following values from the drop-down list:</p> <ul style="list-style-type: none"><input type="checkbox"/> Rising: Only the rising edge needs to switch on the first incident wave.<input type="checkbox"/> Falling: Only the falling edge needs to switch on the first incident wave.<input type="checkbox"/> Both: Both edges need to switch on the first incident wave.<input type="checkbox"/> Neither: Neither edge needs to switch on the first incident wave.<input type="checkbox"/> (Clear): No constraint.
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Usage tab

Use this tab to view application specific information on constraint usage. The Usage tab lists the various DRC, Electrical DRC, and Electrical constraints that are in effect for the current topology analysis.

Procedures

Working with switch-settle constraints

The following procedures explain how to apply and modify switch-settle constraints.

Adding a switch-settle constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Switch-Settle tab.

3. In the Pins frame, select an output pin, or enter a pin name in the *Driver* text box under Rule Editing.

The pin appears in the *Driver* text box.

4. In the Pins frame, select an input pin, or enter a pin name in the *Receiver* text box under Rule Editing.

The pin appears in the *Receiver* text box.

5. For *Min First Switch Delay*, enter minimum switch delay values in the *Rise* and *Fall* fields.

6. For *Max Final Settle Delay*, enter maximum settle delay values in the *Rise* and *Fall* fields.

7. Click *Add*.

The new rule appears in the Existing Rules list.

8. Click *Apply*.

The new values will be applied.

Changing a switch-settle constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Switch-Settle tab.
3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

4. Clear the values in the *Driver* and *Receiver* text boxes, then select the pins that you want to edit from the Pins list. (You can also enter the names of existing pins in the Driver and Receiver text boxes.)
5. Enter new values for *Min First Switch Delays* and *Max Final Settle Delays* in the corresponding *Rise* and *Fall* text boxes.
6. Click *Modify*.

The modified rule appears in the Existing Rules list.

7. Click *Apply*.

The new values will be applied.

Deleting a switch-settle constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Switch-Settle tab.
3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

4. Click *Delete*.

The rule is deleted.

Working with propagation delay constraints

The following procedures explain how to apply and modify propagation delay constraints.

Adding a propagation delay constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Prop Delay tab.

3. In the Pins/Tees frame, select the start pin or T-point, or type the name of an existing pin or TLine in the *From* text box.

The *From* text box under Rules Editing shows the selected pin or T-point.

4. In the Pins/Tees frame, select the end pin or T-point, or type the name of an existing pin or TLine in the *To* text box.

The *To* text box under Rules Editing shows the selected pin or T-point.

5. Select *Delay*, *Length*, or *%Manhattan* from the Rule Type drop-down list.

The value fields change to reflect the selected Rule Type.

6. Depending on the rule type selected, enter the appropriate value in the *Min Delay* and *Max Delay* text boxes.

- ☐ For *Delay*, enter Min Delay and Max Delay values.
- ☐ For *Length*, enter Min Length and Max Length values.
- ☐ For *%Manhattan*, enter % Min and % Max values.

7. Click *Add*.

The new rule appears in the Existing Rules list.

8. Click *Apply*.

The new values will be applied.

Changing a propagation delay constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Prop Delay tab.

3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

Allegro SI SigXplorer Reference

Setup Menu Commands

4. Clear the *From* and *To* values, then select the pins or T-points that you want to edit from the Pins/Tees list. (Or, enter the names of existing pins or T-points in the *From* and *To* text boxes.)
5. Select *Delay*, *Length*, or *%Manhattan* from the Rule Type drop-down list.
The value fields change to reflect the selected Rule Type.
6. Depending on the rule type selected, enter the appropriate value in the Min Delay and Max Delay text boxes.
 - ☐ For *Delay*, enter Min Delay and Max Delay values.
 - ☐ For *Length*, enter Min Length and Max Length values.
 - ☐ For *%Manhattan*, enter % Min and % Max values.
7. Click *Modify*.
The modified rule appears in the Existing Rules list.
8. Click *Apply*.
The new values will be applied.

Deleting a propagation delay constraint

1. Choose *Setup – Constraints*.
The Set Topology Constraints dialog box appears.
2. Select the Prop Delay tab.
3. Select a rule in the Existing Rules list.
The corresponding rule information displays in the Rule Editing frame.
4. Click *Delete*.
The rule is deleted.

Working with impedance constraints

The following procedures explain how to apply and modify impedance constraints.

Adding an impedance constraint

1. Choose *Setup – Constraints*.

Allegro SI SigXplorer Reference

Setup Menu Commands

The Set Topology Constraints dialog box appears.

2. Select the Impedance tab.
3. In the Pins/Tees frame, select the start pin or T-point, or enter the name of an existing pin or TLine in the *From* text box under Rule Editing.

The selected pin or T-point appears in the *From* text box.

4. In the Pins/Tees frame, select the end pin or T-point, or enter the name of an existing pin or TLine in the *To* text box under Rule Editing.

The selected pin or T-point appears in the *To* text box.

5. Enter an impedance value in the Target text box.
6. Select an option (*Ohms* or *%Ohms*) from the Type drop-down list.
7. Enter a value in the Tolerance text box.

- ☐ For *Ohms*, enter a numeric delta value.
- ☐ For *%Ohms*, enter a percentage delta value.

8. Click *Add*

The new rule appears in the Existing Rules list.

9. Click *Apply*.

The new values will be applied.

Changing an impedance constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Impedance tab.
3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

4. Clear the *From* and *To* values, then select the pins or T-points that you want to edit from the Pins/Tees list. (Or, enter the names of existing pins or T-points in the *From* and *To* text boxes.)
5. Enter a new impedance value in the Target text box.

6. Select an option (Ohms or %Ohms) from the Type drop-down list.

7. Enter a new value in the Tolerance text box.

- ☐ For Ohms, enter a numeric delta value.
- ☐ For %Ohms, enter a percentage delta value.

8. Click *Modify*.

The modified rule appears in the Existing Rules list.

9. Click *Apply*.

The new values will be applied.

Deleting an impedance constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Impedance tab.

3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

4. Click *Delete*.

The rule is deleted.

Working with relative propagation delay constraints

The following procedures explain how to apply and modify relative propagation delay constraints.

Adding a relative propagation delay constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Rel Prop Delay tab.

3. In the Rule Editing frame, click *New* to assign a name to the rule, or enter a name in the Rule Name text box.

Allegro SI SigXplorer Reference

Setup Menu Commands

The rule name appears in the Rule Name text box.

4. In the Pins/Tees frame, select the start pin or T-point, or enter the name of an existing pin or TLine in the *From* text box under Rule Editing.

The selected pin or T-point appears in the From text box.

5. In the Pins/Tees frame, select the end pin or T-point, or enter the name of an existing pin or TLine in the *To* text box under Rule Editing.

The selected pin or T-point appears in the *To* text box.

6. Select the desired option from the Scope and Delta Type drop-down lists.

7. Enter the desired value in the Delta text box.

8. Select the desired option from the Tol Type drop-down list.

9. Enter the desired value in the Tolerance text box.

Note: Be sure to enter a tolerance value appropriate to the tolerance type you select:

- ☐ For Delay, enter a numeric delta delay value.
- ☐ For Length, enter a numeric delta length value.
- ☐ For Percent, enter a percentage delay or length value.

10. Click *Add*.

The new rule appears in the Existing Rules list.

11. Click *Apply*.

The new values will be applied.

Changing a relative propagation delay constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Rel Prop Delay tab.

3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

4. If you want to change the name of the rule, click *New*, or enter a new name in the Rule Name text box.

Allegro SI SigXplorer Reference

Setup Menu Commands

The new rule name appears in the Rule Name text box.

5. Clear the *From* and *To* values, then select the pins or T-points that you want to edit from the Pins/Tees list. (Or, enter the names of existing pins or T-points in the *From* and *To* text boxes.)
6. Change the values for *Scope*, *Delta Type*, *Delta*, *Tol Type*, and *Tolerance*, as needed.

Note: Be sure to enter a tolerance value appropriate to the tolerance type you select:

- For *Delay*, enter a numeric delta delay value.
- For *Length*, enter a numeric delta length value.
- For *Percent*, enter a percentage delay or length value.

7. Click *Modify*.

The modified rule appears in the Existing Rules list.

8. Click *Apply*.

The new values will be applied.

Deleting a relative propagation delay constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Rel Prop Delay tab.

3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

4. Click *Delete*.

The rule is deleted.

Working with diff pair constraints

The following procedures explain how to apply and modify diff pair constraints.

Adding a diff pair constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Diff Pair tab.
3. Enter a value in the appropriate text box for the particular constraint you wish to add.
4. Click *Apply*.

The constraint will be added.

Changing a diff pair constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Diff Pair tab.
3. Enter a new value in the appropriate text box for the particular constraint you wish to change.
4. Click *Apply*.

The new value will be applied.

Deleting a diff pair constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Diff Pair tab.
3. Clear the value in the appropriate text box for the particular constraint you wish to delete.
4. Click *Apply*.

The constraint will be deleted.

Working with max parallel constraints

The following procedures explain how to apply and modify max parallel constraints. You can define a maximum of four length/gap pairs. Each pair defines a maximum parallel coupled length between the given net and any other net (assuming the two nets are separated by an air gap that is less than or equal to the given gap distance value).

Adding a max parallel constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Max Parallel tab.
3. Enter a coupled length distance value in the Length text box under the Rule Editing frame.
4. Enter a gap value in the Gap text box under the Rule Editing frame.
5. Click *Add*

The new rule appears in the Existing Rules list.

6. Repeat steps 3 - 5 to add additional rules, up to a maximum of four rules.
7. Click *Apply*.

The max parallel constraints will be applied.

Changing a max parallel constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Max Parallel tab.
3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

4. Enter new values in the Length and Gap text boxes.
5. Click *Modify*.

The modified rule appears in the Existing Rules list.

6. Click *Apply*.

The new values will be applied.

Deleting a max parallel constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Max Parallel tab.

3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

4. Click *Delete*.

The rule is deleted.

Working with wiring constraints

The following procedures explain how to apply and modify wiring constraints. You can apply one of several generic topology schedules to a topology once the required parts have been placed on the canvas. Selecting one of these schedules will cause all of the necessary TLines to be automatically added and connected to the IOCell pins to form the schedule type selected.

Applying a generic schedule to a topology

1. Place all required topology parts on the canvas.

Note: Rules for generic topology schedules apply.

2. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

3. Select the Wiring tab.

4. Select the desired generic schedule type from the Schedule drop-down list.

5. Click *Apply*.

The generic template will be applied and the topology will be scheduled.

Adding a wiring constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Wiring tab.

3. Select the desired Topology options from the drop-down lists for Mapping Mode, Schedule, and Verify Schedule.

Allegro SI SigXplorer Reference

Setup Menu Commands

4. Enter the desired values for the Physical parameters (Stub Length, Max Via Count, Total Trace Length).
5. Enter the desired value for the EMI Max Exposed Length.
6. Click *Apply*.

The wiring constraints will be applied.

Changing a wiring constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Wiring tab.
3. Enter new values for Topology, Physical, or EMI in the appropriate text boxes, or select new options from the appropriate drop-down lists.
4. Click *Apply*.

The new values will be applied.

Working with user-defined constraints

The following procedures explain how to apply and modify user-defined constraints.

Adding a user-defined constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the User-Defined tab.
3. Enter a unique name for the new rule in the Name text box.
4. Select a rule type from the Type drop-down list.
5. Enter minimum and maximum values in the Range text boxes.

Note: Range and Units will appear in the dialog box for all rule types except String. The value for Units is predetermined by the rule type and cannot be modified.

6. In the Value (optional) text box, enter an optional value if needed.
7. Click *Add*.

The new rule appears in the Existing Rules list.

8. Click *Apply*.

The new rule will be applied.

Changing a user-defined constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the User-Defined tab.
3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

4. Enter a new name in the Name text box.
5. Select a new type from the Type drop-down list.
6. Enter new minimum and maximum values in the Range text boxes.

Note: Range and Units will appear in the dialog box for all rule types except String. The value for Units is predetermined by the rule type and cannot be modified.

7. Enter a new value in the Value (optional) text box, if needed.
8. Click *Modify*.

The modified rule appears in the Existing Rules list.

9. Click *Apply*.

The new values will be applied.

Deleting a user-defined constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the User-Defined tab.
3. Select a rule in the Existing Rules list.

The corresponding rule information displays in the Rule Editing frame.

4. Click *Delete*.

The rule is deleted.

Working with signal integrity constraints

The following procedures explain how to apply and modify signal integrity constraints.

Adding a signal integrity constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Signal Integrity tab.
3. In the Reflection frame, enter the desired values for Overshoot and Min. Noise Margin.
4. In the Edge Distortion frame, select the desired options from the drop-down lists for Edge Sensitivity and First Incident Switch.
5. In the Xtalk/SSN frame, enter the desired values for Max Xtalk, Max Peak Xtalk, and Max SSN, and define the Active and Sensitive Xtalk Window parameters.
6. Click *Apply*.

The signal integrity constraints will be applied.

Changing a signal integrity constraint

1. Choose *Setup – Constraints*.

The Set Topology Constraints dialog box appears.

2. Select the Signal Integrity tab.
3. Enter new values for Reflection, Distortion, or Xtalk/SSN in the appropriate text boxes, or select new options from the appropriate drop-down lists.
4. Click *Apply*.

The new signal integrity constraints will be applied.

Applying a generic schedule to a topology

1. Place all required topology parts on the canvas.

Note: Rules for generic topology schedules apply.

2. Choose *Setup – Constraints*.

The Set Constraints dialog box appears.

3. Select the Wiring tab.

4. In the *Schedule* field, select the desired generic schedule type from the drop-down menu.

5. Click *Apply*.

The generic template will be applied and the topology automatically scheduled.

Automatically rescheduling the topology

Note: Rules for generic topology schedules apply. All existing TLines will be deleted.

1. Choose *Setup – Constraints*.

2. Select the Wiring tab.

3. In the Schedule field, select the desired generic schedule option from the drop-down menu.

4. Click *Apply*.

The generic template will be applied and the topology automatically rescheduled.

Defining or editing a layer set constraint

1. Choose *Setup – Constraints*.

2. Select the Wiring tab.

3. In the Layer Sets field, enter (or edit) a constraint for the net or Xnet topology extracted into a top file. An example of the syntax for the constraint is

`LS1:LS2:LS3 ...Ln`

4. Click *Apply*.

The generic template will be applied and the topology automatically rescheduled.

Setup – Defaults

[Dialog Box](#) | [Procedures](#)

Displays the Default Parameter Values dialog box. Here you can set the default parameter attribute values for topology element part models.

These parameter values are automatically associated with the part symbols added from the Model Browser. Use the *Edit – Add Part* command to add new parts.

Note: When you select a part from the Part Type drop-down list, the dialog box changes to display only those parameters associated with the selected part. Each part has a different set of parameters. Any current default values are also displayed.

Dialog Box

The Set Default Values dialog box consists of two tabbed dialogs.

Parameters tab

Use this tab to set default parameter attribute values for topology element part models.

Note: The parameters that are displayed in the Parameters tab adjust to list the appropriate part parameters for the particular topology element you select. If the parameter does not have a default value, the text box is blank.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Part Type</i>	Displays a drop-down list of the topology elements whose default parameter values you can edit. The list includes: <i>Cable</i> <i>Capacitor</i> <i>DiffIO</i> <i>DiffIOPkg</i> <i>DiffInput</i> <i>DiffInputPkg</i> <i>DiffOutput</i> <i>DiffOutputPkg</i> <i>Diode</i> <i>DualClampTerm</i> <i>ESpiceDevice</i> <i>HiClampTerm</i> <i>Inductor</i> <i>LowClampTerm</i> <i>RCTerm</i> <i>Resistor</i> <i>SeriesTerm</i> <i>ShuntTerm</i> <i>Source</i> <i>TheveninTerm</i> <i>Trace</i> <i>Tline</i>
<i>OK</i>	Exits the dialog box and saves any changes you have made.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Parameters tab

Units tab

Use this tab to specify the preferred units for design parameters. The units you specify here apply to all design parameters within a topology.

Units tab

Option	Description
<i>Propagation Delay</i>	Select the preferred units from the drop-down list box. The default is ns.
<i>Noise Voltage</i>	Select the preferred units from the drop-down list box. The default is mV.
<i>Inductance</i>	Select the preferred units from the drop-down list box. The default is nH.
<i>Capacitance</i>	Select the preferred units from the drop-down list box. The default is pF.
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Procedures

Setting the default values for topology element parameters

Default values for topology element parameters are used for a new symbol when it is created.

Note: This procedure modifies the default parameter values that are associated with all new topology element symbols when you add them to the topology canvas. You can also modify the parameter values for a specific topology element, after you place it, by editing the part values in the spreadsheet.

1. Choose *Set – Defaults*.

The Default Values dialog box appears.

2. Under the *Parameters* tab, choose *Cable* from the Part Type drop-down list.

The Cable part model has one associated parameter, Length, which has a default value of 39370.08 MIL.

Note that the parameters that are displayed in the Parameters tab adjust to list the appropriate part parameters for the particular topology element you select. If the parameter does not have a default value, the text box is blank.

3. Modify the values in the corresponding text boxes for any parameters you wish to change.

4. Click the *Units* tab.

As needed, modify the unit values for *Propagation Delay*, *Noise Voltage*, *Inductance*, or *Capacitance*.

5. Click *Apply* to apply the changes and continue editing, or click *OK* to exit.

Setup – Strobe Pins

Dialog Box

Displays the Set Strobe Pin Groups dialog box where you can mark and group strobe and data pins.

Dialog Box

Option	Description
<i>Existing Strobe Groups</i>	Lists the Strobe Pin name, the associated Active Edges and the Data Pins for each strobe pin in the topology.
<i>Available Pins</i>	Displays all pins in the topology that are not currently assigned as strobe or data pins.
<i>Pin Name Filter</i>	Limits the Pin Names displayed in the list box. Initially the field contains an asterisk (*) character to display all available pins.
<i>Pin</i>	Lists the names of available pins in the topology.
<i>Signal</i>	Identifies the Signal associated with the pin.
<i>Strobe Selection</i>	Displays the selected Strobe Pin and its associated Active Edges and Data Pins for modification.
<i>Strobe Pin</i>	Displays the selected strobe pin.
<i>Active Edge</i>	Displays the active edges associated with the strobe pin where data can be triggered for the data pins. Select Rising, Falling, or Both from the drop-down list.
<i>Data Pins</i>	Lists the data pins associated with the strobe pin.
<i>Add</i>	Adds the strobe pin group currently displayed in the Strobe Selection area to the Existing Strobe Groups area and clears the Strobe Selection area.
<i>Modify</i>	Add the modifications made to the strobe pin group currently displayed in the Strobe Selection area. Displays the modified strobe pin group in the Existing Strobe Groups area and clears the Strobe Selection area.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Delete</i>	Deletes the selected strobe pin group from the Existing Strobe Groups and Strobe Selection areas and returns pins to the Available Pins area.
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Setup – Vectors

[Dialog Box](#) | [Procedures](#)

Displays the Vector Set Operations dialog where you can save a snapshot of all IOCell stimuli and parameters in the topology as a named vector set. From this dialog box, you can also restore and delete vector sets, or view them in SigWave.

Dialog Box

Option	Description
<i>Name</i>	Displays a drop-down list of available test vector sets. Use this field to enter the name of a new test vector set.
<i>Save</i>	Saves a new test vector set.
<i>Restore</i>	Restores the test vector set selected in the Name drop-down list box.
<i>Delete</i>	Removes the test vector set selected in the Name drop-down list box.
<i>View</i>	Invokes SigWave to display the test vector set selected in the Name drop-down list box.
<i>OK</i>	Exits the dialog box and saves any changes you have made.
<i>Apply</i>	Applies any changes you have made without exiting the dialog box.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

Procedures

Creating a test vector set

1. Choose *Setup – Vectors*.

The Vector Set Operations dialog box appears.

2. In the Name field, enter the name for the test vector set. To replace a name that is already displayed in the name field, select the name before typing.

3. Click Save.

The test vector set is saved. Its name is added to the pull-down menu.

4. Click *OK* or *Apply*.

Restoring a saved test vector set

1. Choose *Setup – Vectors*.

The Vector Set Operations dialog box appears.

2. In the Name field pull-down menu, click to select the name of a test vector set.

3. Click Restore.

The topology is changed to reflect the parameters and IOCell stimuli in the restored test vector set.

4. Click *OK* or *Apply*.

Deleting a test vector set

1. Choose *Setup – Vectors*.

The Vector Set Operations dialog box appears.

2. In the Name field pull-down menu, click to select the name of a test vector set.

3. Click Delete.

The name of the selected test vector set is removed from the pull-down menu.

4. Click *OK* or *Apply*.

Viewing a test vector set

1. Choose *Setup – Vectors*.

The Vector Set Operations dialog box appears.

2. In the Name field pull-down menu, click to select the name of a test vector set.

3. Click View.

Allegro SI SigXplorer Reference

Setup Menu Commands

SigWave is invoked in timing diagram mode with the selected test vector set displayed.

4. Click *OK* or *Apply*.

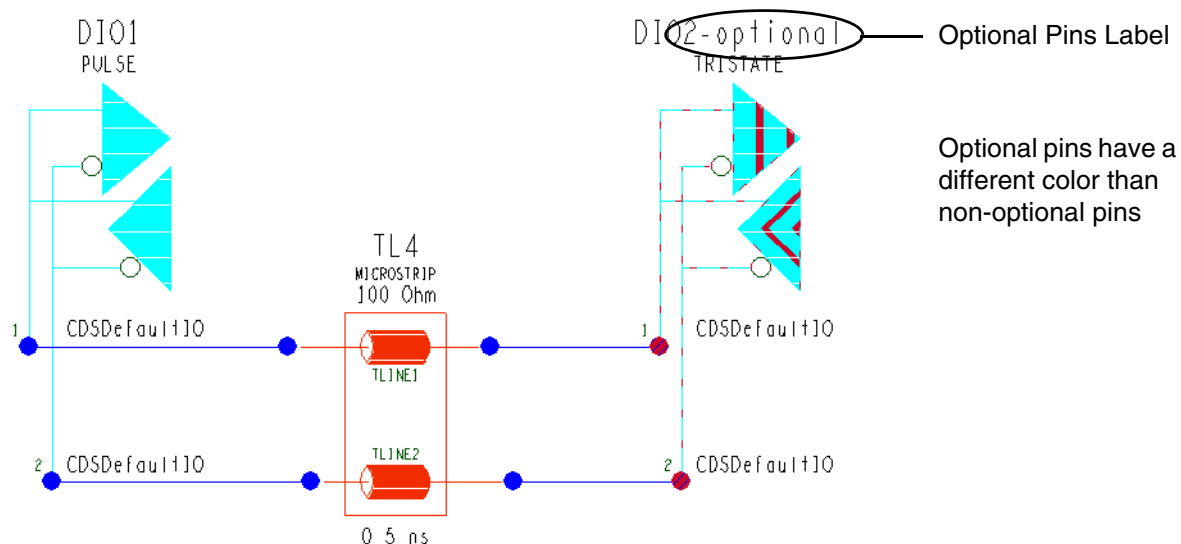
Setup – Optional Pins

Use this command to specify *optional* pins in a topology.

In modern bus design, it is common to have buses that have a similar purpose, yet have a slight difference in the number of pins in each net in the bus. Topology mapping mandates that the number of pins in a topology exactly match the number of pins in a net or Xnet. To relax this restriction, you can designate one or more pins in the topology as *optional*. In this way, the topology can successfully map to a net or an Xnet that may not have the same number of pins.

Important

You cannot make vias, T-lines, T-points, traces, and termination networks optional.



Note the optional pins on a differential buffer. SigXplorer adds the label, *optional*, to pins that you designate as optional. If you make the inverting pin of a differential buffer optional, SigXplorer makes the non-inverting pin optional. The converse is also true.

Procedures

Making a pin optional

1. Choose *Set – Optional Pins*.

Allegro SI SigXplorer Reference

Setup Menu Commands

2. Click on a pin in the canvas.

Note: You cannot make pins of vias, T-lines, T-points, traces, and termination networks optional.

3. Optionally, select additional pins.
4. Right-click and choose *Done* from the pop-up menu.

Removing an optional pin

1. Choose *Setup – Optional Pins*.
2. Click on a designated *optional* pin in the canvas.
3. Optionally, select additional pins in which to remove the *optional* designation.
4. Right-click and choose *Done* from the pop-up menu.

Setup – Manage LayerStacks

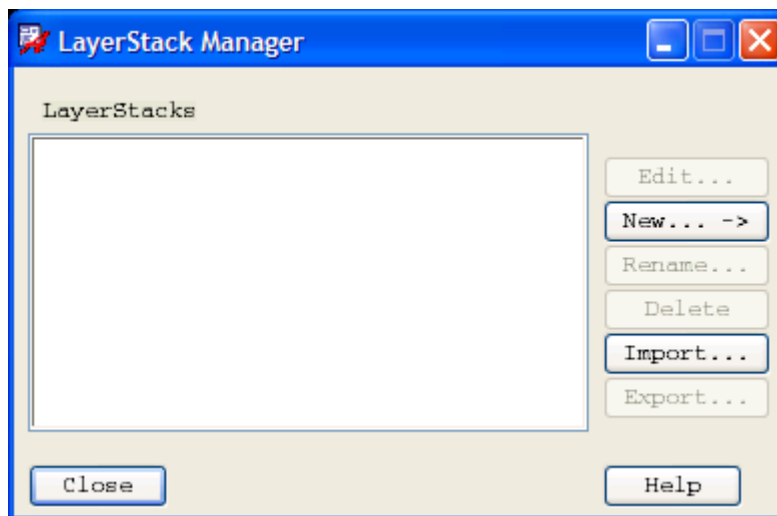
[Dialog Box](#) | [Procedures](#)

This is the dialog is used for managing LayerStacks in the current topology. In 16.3, SigXplorer is stack-up aware:

- SigXplorer uses the exact same stack-up data as an Allegro board file.
- Traces in the topology can easily be limited to a particular stack-up

Topologies extracted from a board file also extract the board's stack-up. Multiple stack-ups can be managed for topologies that span multiple designs.

There are options in this dialog to either add from a set of provided default stack-ups or to import them from .brd or tech files.



Dialog Box

Option	Description
<i>LayerStacks</i>	Lists the available layer stacks.
<i>Edit</i>	Launches the Layout Cross Section dialog box where you can edit the selected layer stack.
<i>New</i>	Creates a new layer stack. Provides options to create stack of 2, 4, 6, and 8 layers.

Allegro SI SigXplorer Reference

Setup Menu Commands

Option	Description
<i>Rename</i>	Renames the layer stack name of the selected layer stack with a new name.
<i>Delete</i>	Deletes the selected layer stack.
<i>Import</i>	Imports a layer stack from a specified board (.brd) or technology file (.tcf).
<i>Export</i>	Exports the selected layer stack to a specified technology file.
<i>Close</i>	Closes the dialog box.
<i>Help</i>	Launches the SigXplorer Help system and displays the relevant Help topic.

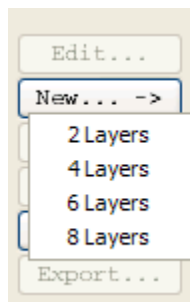
Procedures

Creating a new layer stack

1. Choose *Setup – Manage LayerStacks*

The LayerStack Manager dialog box appears.

2. Click *New*.



3. Select the number of layers you want in the new layer stack.
4. Specify a name for the new layer stack and click *OK*.

The new layer stack name is added to the LayerStacks list.

Editing a layer stack

1. Choose *Setup – Manage LayerStacks*

The LayerStack Manager dialog box appears.

2. Select a layer stack in the LayerStacks list.
3. Click *Edit*.

The Layout Cross Section dialog box appears.

4. Make the required changes and click *OK*.

Renaming a layer stack

1. Choose *Setup – Manage LayerStacks*

The LayerStack Manager dialog box appears.

2. Select a layer stack in the LayerStacks list.
3. Click *Rename*.
4. Specify a name for the new layer stack and click *OK*.

Deleting a layer stack

1. Choose *Setup – Manage LayerStacks*

The LayerStack Manager dialog box appears.

2. Select a layer stack in the LayerStacks list.
3. Click *Delete*.
4. Click *Yes* in the confirmation box.

The layer stack is deleted.

Importing a layer stack

1. Choose *Setup – Manage LayerStacks*

The LayerStack Manager dialog box appears.

2. Click *Import*.
3. Specify the name of the board (.brd) or technology file (.tcf) from which you want to import the layer stack.
4. Click *Open*.

5. Specify a new name for the layer stack to be imported. and click *OK*.

The imported layer stack is added to the LayerStacks list.

Exporting a layer stack

1. Choose *Setup – Manage LayerStacks*

The LayerStack Manager dialog box appears.

2. Select the layer stack to be exported.
3. Click *Export*.
4. Specify a name for the new technology file (.tcf).
5. Click *Save*.

The selected layer stack is exported as a technology file.

Allegro SI SigXplorer Reference

Setup Menu Commands

Analyze Menu Commands

Analyze – Model Browser

Dialog Boxes | Procedures

Displays the SI Model Browser. Use the SI Model Browser for specifying the device and interconnect libraries used by the simulator during signal analysis. These libraries contain the device and interconnect models used by the simulator to build circuit simulations.

Other associated dialog boxes launched via the SI Model Browser enable you to create and edit device and interconnect models contained in these libraries.

Dialog Boxes

SI Model Browser

DML Library
Management

[Set Model Search Path](#)

IBIS Device Pin Data

Buffer Delays

Analog Output Model Editor

IOCell Editor

V/I Curve Editor

V/T Curve Editor

[Set V/I Curve Point](#)

Via Model Generator

IBIS Device Model Editor

SI Model Browser

Using SI Model Browser (and its associated dialog boxes) you can perform the following basic model development tasks:

- List the models in a library.
- Create a device model with default values or clone an existing device model and add the newly created model to the working library.
- Delete a model from the working library.
- Translate a model.

Allegro SI SigXplorer Reference

Analyze Menu Commands

The SI Model Browser's tabbed interface accommodates the model type that you want to translate, be it IBIS, Spectre, Spice, IML, or HSPICE. You need to select the appropriate tab, click the model, and click the *Translate* button to translate it. From these tabs, you can also edit a model directly in its native format. Once translated, these models also appear under the DML tab.

Each tab contains a field for filtering the listed models, as well as a button to set the model's library search path and to set its associated file extensions (Set Model Search Path dialog box).

You can filter fields at the top of the SI Model Browser control which models are displayed in the Model Browser list box. You can specify which models are listed in the model search list by library, by model type, or by characters in the model name.

Displaying a List of Models

Model List Options

Option	Display shows . . .	Function
<i>Library Filter</i>	Currently selected device or model library.	Changes the current device or library (click arrow).
<i>Model Type Filter</i>	Current model filter setting.	Changes the model filter to display only models of a particular type (click arrow).
<i>Model Name Pattern</i>	Current model name pattern setting.	Changes the model name pattern string to display only models whose name is included in the specified character string (edit type-in box).

Note: Use * for a wildcard selection.

Creating Models and Adding them to a Working Library

You can add a device or interconnect model to the working device or interconnect model library in either of two ways:

- By copying (or cloning) an existing model.
- By creating a new model with default values.

You must first create a device model and add it to the working library before you can edit it to characterize a particular device.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Create / Add Model Buttons

Button	Function
<i>Add-></i>	Displays the <i>Add Model</i> pop-up menu and enables you to choose a device and interconnect model type to add to your working device or interconnect library. Note: Menu options vary according to the library type selected. The following menu option is common when either a device or interconnect library is selected. <i>CloneSelection</i> Copies or clones the model that you select in the SI Model Browser list, prompts you to name the copy, and adds the renamed copy to the working library.
<i>Delete</i>	Deletes the selected model.
<i>Edit</i>	Displays a text editor or a model editor, depending on the type of model you select in the SI Model Browser search list.
<i>Select</i>	Selects a model.
<i>Set Search Path</i>	Launches the <u>Set Model Search Path</u> dialog box.

DML Library Management

You use the DML Library Management dialog box to create and manage your libraries of device and interconnect models, and launch Model Editor. You can also use it to specify which device and interconnect libraries you want SigXplorer to access, as well as the order of library access (in the Set Model Search Path dialog box).

Libraries are searched starting at the top of the list. If a model is included in two or more libraries, you can use the search order (in the Set Model Search Path dialog box) to determine which library the simulator searches first. The simulator uses the first model found.

You can also set a particular library as the working library. A working library is the only library to which the simulator can add models. If you want to add to a library that is not the working library, you must make it the working library before you start the process of adding the model.

Allegro SI SigXplorer Reference

Analyze Menu Commands

You can have at most two working libraries: one working device model library and one working interconnect model library.

Option	Function
<i>Working Library</i>	Sets the working (or active) library for device models. (Before you edit or add new models, make the target library the <i>working</i> library.)
<i>Ignore Library</i>	Ignores the library during search.
<i>Select for Merge/Index</i>	Select the libraries for merging or indexing.
<i>Create New Lib</i>	Displays a file browser where you can specify the device/interconnect library (.dml or .iml) to be created and added to the device library search list.
<i>Check Lib</i>	Runs the <code>dmlcheck</code> utility on the selected model and displays the result in a log file.
<i>Merge Libs</i>	Merges all .dml files present in the library list into one .dml file.
<i>Make Lib Index</i>	Creates an index file for all of the .dml files present in the library list. (The working library is excluded.)
<i>Set Search Path</i>	Opens the Set Model Search Path dialog box.

Set Model Search Path

Use the *Set Model Search Path* dialog box to specify the directories in which to search for signal models, and their search order.

Option	Function
<i>Add Directory</i>	Adds a directory containing device library or device index to the device library search list.
<i>Move To Top</i>	Raises the selected device library to the topmost position in the device library search list.
<i>Move Up</i>	Raises the selected device library one position up in the device library search list.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Option	Function
<i>Move Down</i>	Lowers the selected device library one position down in the device library search list.
<i>Move To Bottom</i>	Lowers the selected device library to the bottom position in the device library search list.
<i>Remove Library</i>	Removes selected device libraries from the device library search list.
<i>Reset To Default</i>	Resets the library list to default as specified by the <code>SI_MODEL_PATH</code> directive in the <code>cds.cpm</code> file.

Analog Output Model Editor Dialog Box

Option	Function
<i>Model</i>	Displays the name of the Analog Output model.
<i>Series Resistance</i>	Displays the resistance value for a series resistor.
<i>Rise</i>	Displays the path to an Analog Workbench file or displays a file browser.
<i>Fall</i>	Displays the path to an Analog Workbench file or displays a file browser.
<i>Pulse</i>	Displays the path to an Analog Workbench file or displays a file browser.
<i>Inv Pulse</i>	Displays the path to an Analog Workbench file or displays a file browser.

IBIS Device Model Editor Dialog Box

The IBIS Device Model Editor dialog box contains three tabs that you can use to perform the following tasks.

Edit information for the pins associated with the IBIS device model.

Allegro SI SigXplorer Reference

Analyze Menu Commands

- Group power and ground pins and assign them to power and ground buses.
- Group signal pins and assign IOCell models and IOCell supply buses.

Edit Pins Tab

Model Info Area

Option	Function
<i>Model Name</i>	Name of the IBIS device model.
<i>Manufacturer</i>	Name of the model manufacturer (not used by SigNoise).
<i>Package Model</i>	Name of a package model associated with the IBIS device model.

Estimated Pin Parasitics Area

Option	Function
<i>Resistance</i>	Minimum, typical, and maximum values for resistance.
<i>Capacitance</i>	Minimum, typical, and maximum values for capacitance.
<i>Inductance</i>	Minimum, typical, and maximum values for inductance.

IBIS Pin Data Area.

Option	Function
<i>Pin</i>	The pin number.
<i>Signal</i>	The signal associated with the pin.
<i>IOCell</i>	The associated IOCell model.
<i>Resistance</i>	The resistance, if you are using individual pin parasitics.
<i>Capacitance</i>	The capacitance, if you are using individual pin parasitics.
<i>Inductance</i>	The inductance, if you are using individual pin parasitics.
<i>DiffPair Mate</i>	The inverse pin, if the pin is part of a differential pair.
<i>Wire</i>	The wire number, which determines which wire of the PackageModel is used for this pin.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Edit Pins Buttons

Button	Function
<i>Add Pin Data</i>	Prompts for the name of a new pin to add, and displays the IBIS Device Pin Data dialog box to add or modify data including buffer delays for a new pin.
<i>Measure Delays</i>	<p>Measures buffer delays by simulating each pin with the proper test load. On pins with a Model Selector assigned, buffer delays are simulated for each selectable IOCELL. If you use a Package Model, you must perform simulations for each driver pin. Otherwise, pins with identical parasitics and IOCELL assignments will share simulation data. A progress meter displays the status of the process for buffer delay simulations, especially for complex parts. You can click the Stop button to cancel the simulation.</p> <p>Options:</p> <p><i>Unmeasured Drivers</i> - creates data for drivers not previously processed.</p> <p><i>All Drivers</i> - creates data for all drivers, refreshes previously processed data.</p> <p><i>Clear All Delays</i> - deletes all buffer and differential pair delays from the model.</p>
<i>Set WireNumbers</i>	<p>Sets the wire number for each pin based on a sort criteria.</p> <p>Options:</p> <p><i>Order by Pin Name</i> - Pins are sorted by pin name. Wire numbers are assigned numerically starting at one. Alphabetic and numeric portions of names are separately considered so that, for example, A2 appears before A10 and B6.</p> <p><i>Order by IOCell Name</i> - Pins are sorted first by IOCell model name. Second, pins with the same IOCell model assigned are sorted by pin name. Wire numbers are then assigned numerically starting at one.</p> <p>Note: After the wire numbers have been set, the pin list is displayed in wire number order.</p>
<i>DML Check</i>	Runs the <code>dmlcheck</code> utility on the model being edited and displays the result in a text window.
<i>OK</i>	Runs <code>dmlcheck</code> if changes to the model are made. Otherwise, choosing OK closes the window.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Assign Power/Ground Pins Tab

All Pins Area

Option	Function
<i>Pin #</i>	The pin number.
<i>IOCell</i>	Any IOCell model currently assigned to the pin.
<i>Pwr Bus</i>	Any power bus currently assigned to the pin. This column is blank for a pin that is not currently assigned to a power bus.
<i>Gnd Bus</i>	Any ground bus currently assigned to the pin. This column is blank for a pin that is not currently assigned to a ground bus.
<i>Pinuse</i>	The pin use code. UNSPEC indicates that no pin use is assigned.
<i>Net Name</i>	The name of any net connected to the pin. This column is blank when you are editing a model selected from a library. This column displays a net name when you are editing a model associated with a device that exists in the active design.
<i>Nets Shown for Component field</i>	The RefDes for the device associated with the model being edited. This occurs when you invoke the IBIS Device Model Editor for a specific instance of a device selected in the Model Assignment dialog box.
<i>Sort By (column buttons)</i>	Selects one of the columns on which to sort the data.
<i>Filters (pulldown menus)</i>	Filters the information displayed in the column. Initially the field contains an asterisk (*) so that all data is displayed. The <i>Power Bus</i> menu lists existing power buses. The <i>Ground Bus</i> menu lists existing ground buses. The <i>Pinuse</i> menu lists existing pin use codes.

All Pins Area Buttons

Button	Function
<i>Select All</i>	Selects all pins currently displayed in the <i>All Pins</i> list box and re-displays them in the <i>Selected Pins</i> list box.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Button	Function
<i>Deselect All</i>	Deselects all pins currently selected and clears the <i>All Pins</i> list box.
<i>Deselect One</i>	Deselects one pin in the <i>All Pins</i> list box.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Select Pins Area

Option	Function
<i>Assign/De-assign buttons</i>	Assigns or de-assigns the group of pins listed in the <i>Selected Pins</i> list box to the IOCell model, power bus, or ground bus named in the associated field.
<i>Pin #</i>	The pin number for pins selected from the <i>All Pins</i> list box.
<i>IOCell</i>	The IOCell model assigned for pins selected from the <i>All Pins</i> list box.
<i>Pwr Bus</i>	The power bus name for pins selected from the <i>All Pins</i> list box.
<i>Gnd Bus</i>	The ground bus name for pins selected from the <i>All Pins</i> list box.
<i>IOCell Browse (field and button)</i>	Enter the IOCell model name to be assigned to the group of pin or click Browse to display the Model Browser and select an IOCell model there.
<i>Pwr Bus (field and menu)</i>	Select an existing power bus name from the menu.
<i>Gnd Bus (field and menu)</i>	Select an existing ground bus name from the pull down menu.

Select Pins Area Buttons

Button	Function
<i>DML Check</i>	Runs the <code>dmlcheck</code> utility on the model being edited and displays the result in a text window.
<i>OK</i>	Closes the window without running <code>dmlcheck</code> .

Assign Signal Pins Tab

All Pins Area

Option	Function
<i>Pin #</i>	The pin number.
<i>IOCell</i>	Any IOCell model currently assigned to the pin.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Option	Function
<i>Pwr Bus</i>	Any power bus currently assigned to the pin. This column is blank for a pin that is not currently assigned to a power bus.
<i>Gnd Bus</i>	Any ground bus currently assigned to the pin. This column is blank for a pin that is not currently assigned to a ground bus.
<i>Pinuse</i>	The pin use code. UNSPEC indicates that no pin use is assigned.
<i>Net Name</i>	The name of any net connected to the pin. This column is blank when you are editing a model selected from a library. This column displays a net name when you are editing a model associated with a device that exists in the active design.
<i>Nets Shown for Component field</i>	The RefDes for the device associated with the model being edited. This occurs when you invoke the IBIS Device Model Editor for a specific instance of a device selected in the Model Assignment dialog box.
<i>Sort By (column buttons)</i>	Selects one of the columns on which to sort the data.
<i>Filters (pulldown menus)</i>	<p>Filters the information displayed in the column. Initially the field contains an asterisk (*) so that all data is displayed.</p> <p>The <i>Power Bus</i> menu lists existing power buses.</p> <p>The <i>Ground Bus</i> menu lists existing ground buses.</p> <p>The <i>Pinuse</i> menu lists existing pin use codes.</p>

Allegro SI SigXplorer Reference

Analyze Menu Commands

All Pins Area Buttons

Button	Function
<i>Select All</i>	Selects all pins currently displayed in the <i>All Pins</i> list box and re-displays them in the <i>Selected Pins</i> list box.
<i>Deselect All</i>	Deselects all pins currently selected and clears the <i>All Pins</i> list box.
<i>Deselect One</i>	Deselects one pin in the <i>All Pins</i> list box.

Select Pins Area

Option	Function
<i>Assign/De-assign buttons</i>	Assigns or de-assigns the group of pins listed in the <i>Selected Pins</i> list box to the IOCell model, power bus, or ground bus named in the associated field.
<i>Pin #</i>	The pin number for pins selected from the <i>All Pins</i> list box.
<i>IOCell</i>	The IOCell model assigned for pins selected from the <i>All Pins</i> list box.
<i>Pwr Bus</i>	The power bus name for pins selected from the <i>All Pins</i> list box.
<i>Gnd Bus</i>	The ground bus name for pins selected from the <i>All Pins</i> list box.
<i>IOCell Browse (field and button)</i>	Enter the IOCell model name to be assigned to the group of pin or click Browse to display the Model Browser and select an IOCell model there.
<i>Pwr Bus (field and menu)</i>	Select an existing power bus name from the menu.
<i>Gnd Bus (field and menu)</i>	Select an existing ground bus name from the pull down menu.

Select Pins Area Buttons

Button	Function
<i>DML Check</i>	Runs the <code>dmlcheck</code> utility on the model being edited and displays the result in a text window.
<i>OK</i>	Closes the window without running <code>dmlcheck</code> .

IBIS Device Pin Data Dialog Box

From the IBIS Device Model Editor, you can display the IBIS Device Pin Data dialog box to:

- Add or edit data (including individual pin parasitics) for the pins in the IBIS device model.
- Add or edit buffer delay information for the pins in the IBIS device model.

IBIS Pin Map Area

Option	Function
<i>Pin</i>	The pin whose data is displayed.
<i>Signal</i>	The signal associated with the pin. Pins with an NC signal are not connected. You can ignore these pins in the IBIS Device Model Editor.
<i>Resistance</i> <i>Capacitance</i> <i>Inductance</i>	The Individual Pin Parasitic values for the pin (if you are not using a package model).
<i>Wire Number</i>	The wire number for the pin. (This can be the same as the pin number if numeric.) Wire numbers specify the wire numbers for the package model and are used only for IBIS device models that have a package model.
<i>IOCell</i>	The IOCell model associated with the pin. Pins with an NC model are not connected. You can ignore these pins in the IBIS Device Model Editor.

If you want to view the voltage versus current (V/I) curves for an IOCell model before you assign it to a pin as part of the IBIS device model, open the model in the IOCell Editor and use the View VI button.

Note: Whenever you have made changes to the IOCell models for a device, regenerate the buffer delay values for the device using All Drivers mode.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Option	Function
<i>Power Bus</i>	Name of the power bus.
<i>Power Clamp Bus</i>	Name of the power clamp bus.
<i>Ground Bus</i>	Name of the ground bus
<i>Ground Clamp Bus</i>	Name of the ground clamp bus.

Diff Pair Data Area

Option	Function
<i>Type</i>	Identifies the pin listed in the <i>Pin</i> field as the Inverting or Non-inverting pin of the differential pair. When the <i>Type</i> field displays <i>None</i> , the pin identified in the <i>Pin</i> field is not part of a differential pair.
<i>Mate Pin</i>	The name of the differential pair mate pin to the pin identified in the <i>Pin</i> field.
<i>Launch Delay</i>	Minimum, typical and maximum launch delay values for the pin, if it is an Output or IO pin.
<i>Input High</i>	Minimum, typical and maximum differential logic threshold values.
<i>Input Low</i>	
<i>Output High</i>	
<i>Output Low</i>	

IBIS Device Pin Data Buttons

Button	Function
<i>Buffer Delays</i>	Displays the Buffer Delays dialog box that enables you to change the buffer delay information for a pin. This dialog box contains the data that SigNoise uses to calculate buffer delay values for rising and falling drivers (output buffers).

Buffer Delays Dialog Box

Option	Function
<i>IOCell Test Fixture:</i> <i>Resistor</i> <i>Capacitor</i> <i>Term Voltage</i> <i>Ref Voltage</i>	Displays values entered in the Delay Measurement tab of the IOCell Editor for the associated IOCell model.
<i>Rise Delay</i>	Fast, Typical, and Slow values for rise delay measured from IOCell delay measurement information. Use these fields to edit output buffer delay values directly.
<i>Fall Delay</i>	Fast, Typical, and Slow values for fall delay measured from IOCell delay measurement information. Use these fields to edit output buffer delay values directly.
Button	Function
<i>Edit IOCell</i>	<p>Starts IOCell Editor for the associated IOCell model where you can edit test fixture values (resistor, capacitor, term voltage, and ref voltage) and other IOCell model information.</p> <p>Test fixture values specify the loading conditions under which SigNoise measures buffer delays. Test fixture values are associated with the IOCell model for a pin rather than with the pin itself.</p>
<i>Measure Delays</i>	<p>Refreshes the delay values if the IOCell model information has changed: slow, typical, and fast buffer delay values for rising and falling drivers. SigNoise performs the buffer delay measurements in All Drivers mode. (You can also measure buffer delays from the IBIS Device Model Editor.)</p> <p>Whenever any IOCell model data changes, it is important to recalculate buffer delay values in All Drivers mode using either the <i>Measure Delays</i> button or the <i>Measure Delays–All Drivers</i> button in the IBIS Device Model Editor.</p>

Allegro SI SigXplorer Reference

Analyze Menu Commands

IOCell Editor Dialog Box

General Tab

Input Section Tab

Output Section Tab

Delay Measurement Tab

Common Buttons

Button	Function
<i>View VI</i>	Displays the minimum, typical, or maximum VI curve.

General Tab

Option	Function
<i>Name</i>	Displays the name of the model.
<i>Type</i>	Displays the type of model.
<i>Technology</i>	Displays a pop-up menu of technologies. Choices are CMOS, TTL, and ECL.
<i>Die Capacitance</i>	Displays minimum, typical, and maximum values for die capacitance.
<i>Reference Temperature</i>	Displays minimum, typical, and maximum reference temperatures.

Button	Function
<i>Power Clamp</i>	Starts the V/I Curve Editor for PowerClamp
<i>Ground Clamp</i>	Starts the V/I Curve Editor for GroundClamp

Allegro SI SigXplorer Reference

Analyze Menu Commands

Input Section Tab

Option	Function
<i>Logic Thresholds</i>	Displays minimum, typical, and maximum values for high and low input thresholds.
<i>View VI</i>	Displays the minimum, typical, or maximum VI curve.

Output Section Tab

Option	Function
<i>Ramp (20%/80%)</i>	Displays minimum, typical, and maximum dV and dT values for rising and falling slew rates.

Button	Function
<i>PullUp</i>	Invokes the VI curve editor to examine PullUp VI curves.
<i>PullDown</i>	Invokes the VI curve editor to examine PullUp VI curves.
<i>Rise Wave</i>	Invokes the VT curve editor to examine Rise Wave VT curves
<i>Fall Wave</i>	Invokes the VT curve editor to examine Fall Wave VT curves

Delay Measurement Tab

Option	Function
<i>Test Fixture -- Resistor, Capacitor, and Termination Voltage</i>	Displays test fixture values for resistance, capacitance, and termination voltage.
<i>V Measure</i>	Displays the reference voltage.

V/I Curve Editor Dialog Box

Option	Function
<i>Reference Voltage</i>	Displays the minimum, typical, and maximum reference voltages.
<i>V/I Convention</i>	Specifies either IBIS or Databook format.
<i>Voltage</i>	The voltage of the curve point.
<i>Min I</i>	The minimum tolerance of the curve point in mA.
<i>Typ I</i>	The typical tolerance of the curve point in mA.
<i>Max I</i>	The maximum tolerance of the curve point in mA.

Button	Function
<i>Add</i>	Adds, modifies, or deletes a curve point. (Displays the Set V/I Curve Point dialog box.)
<i>View</i>	Displays the minimum, typical, and maximum curves in the SigWave window.

V/T Curve Editor Dialog Box

Option	Function
<i>Test Package (R, L, C)</i>	Resistance, delay, and capacitance for the test package.
<i>V/T Curve Test Fixture (R, L, C) and (Vmin, Vtyp, Vmax)</i>	Resistance, delay, and capacitance for the test fixture.

Button	Function
<i>View</i>	Display the curve in the SigWave window.

Button	Function
<i>Import</i>	Imports the <code>.wave</code> file.

Set V/I Curve Point Dialog Box

Option	Function
<i>Voltage</i>	The voltage of the curve point.
<i>Min I</i>	The minimum tolerance of the curve point in mA.
<i>Typ I</i>	The typical tolerance of the curve point in mA.
<i>Max I</i>	The maximum tolerance of the curve point in mA.
<i>Delete</i>	Deletes the selected line from the VI Curve Editor.

Procedure

[Working with Libraries](#)

[Working with Device Models](#)

[Working with Interconnect
Models](#)

Working with Models and Libraries

Specifying a working device model library/interconnect model library

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click the *Library Management* button.

The DML Library Management dialog opens.

3. In the *DML Libraries* list, click the *Working Library* check box next to the library, which you want to designate as the *working* device model library.

Note: For IML Libraries, select the library in the *IML Libraries* list (bottom pane of the window) instead of DML Libraries.

4. Click the *Set Search Path* button.

The Set Model Search Path dialog appears. The library file name you designated as the working library appears in the *Directories To Be Searched for Model Files* list. You can change the search order of libraries in this dialog box.

5. Click *OK*.
6. Click *OK*.

Adding a device library or index

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click *Set Search Path*.
3. In the Set Model Search Path dialog box, click *Add Directory* and browse to the location where the desired library or index files are present.
4. Click *OK*.
5. Use the *Move To Top*, *Move Up*, *Move Down*, or *Move To Bottom* buttons to set the search priority.
6. Click *OK*.

The directory containing libraries or index files is added to the *Directories To Be Searched for Model Files* search list. If the library is not in the working directory, the full path to the library is displayed in the list box.

Adding a standard Cadence Library

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click *Set Search Path*.
3. In the Set Model Search Path dialog box, click *Add Directory* and browse to the location where one of the following libraries is present:
 - ☐ `cds_models.ndx` (A small sample model library.)
 - ☐ `cds_partlib.ndx` (The standard Cadence parts library.)
4. Click *OK*.

Deleting a library from the search list

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. In either the *Device Library Files* list or the *Interconnect Library Files* list, select the library you want to delete.
3. Click *Remove Library*.

The selected library is deleted from the search list.

Creating a device model index

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click *Library Management*.
3. Click the *Select for Merge/Index* check boxes next to the `.dml` files for which you want to create an index.
4. Click *Make Lib Index*.

The Save As dialog box appears.

5. Enter a name for the new index file and click *Save*.
6. Click *Yes* in the message box stating that the selected files be included in the index.
7. Click *OK*.

Note: Index files (`.ndx`) are read-only. For this reason, you cannot include a `.dml` file that is designated as the working library, since the simulator automatically saves edits to the working library file.

Creating a device model index from the operating system command line

- Use the `mkdeviceindex` utility from the operating system command line to create a library index for one or more device model library files.

Reordering the libraries in the search list

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click *Set Search Path*.
3. In the Set Model Search Path dialog box, select a library and use the *Move To Top*, *Move Up*, *Move Down*, or *Move To Bottom* buttons to reorder the libraries in the search list.
4. Click *OK*.

Merging device model libraries

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click *Library Management*.
3. Click the *Select for Merge/Index* check boxes next to the .dml files which you want to merge.
4. Click *Merge Libs*.

The Save As dialog box appears.

Note: All of the .dml files shown in the *Device Library Files* search list will be merged together. Files with extensions other than .dml are ignored.

5. Enter a name for the new merged file and click *Save*.
6. Click *Yes* in the message box stating that the selected files be merged.
7. Click *OK*.

The new merged file replaces all of the .dml files previously listed in the search list.

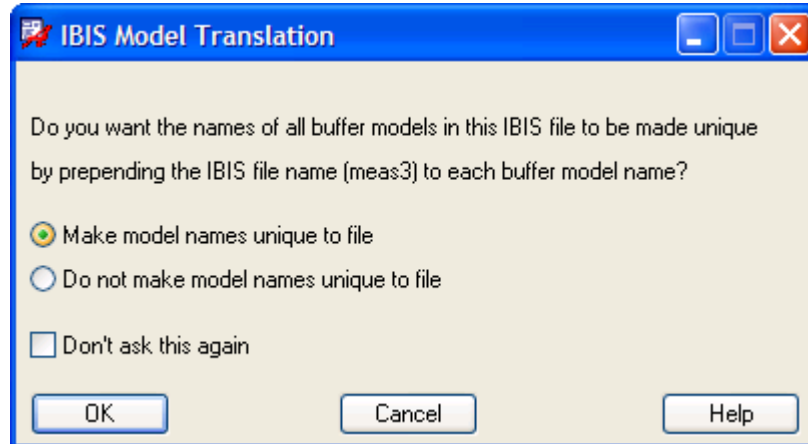
Translating other device model library formats to DML

The SI Model Browser's tabbed interface accommodates the model type that you want to translate to a .dml format, be it IBIS, Spectre, Spice, IML, or HSPICE. You need to select the appropriate tab, click the model, and click the *Translate* button to translate it. From these tabs, you can also edit a model directly in its native format. Once translated, these models also appear under the DML tab.

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select the model to be translated.
3. Click *Translate*.



4. Choose whether you want to make model names unique to the file.
5. Click *OK*.

The selected file is translated into the specified .*dm1* file. The new .*dm1* file is added to the search list.

Any warnings or error messages that are generated during the translation process are displayed in a corresponding text window.

Note: For information on translating IBIS and Spice models, refer to *Translating IBIS and Spice Models* section of the *Working with Signal Models and Libraries* chapter of Allegro SI SigXplorer User Guide.

Working with Device Models

Note: See the Cadence Sample Device Model Library for commented device model examples and sample formats. The Cadence sample device model library is located in your installation hierarchy in the following directory:

```
/install_dir/share/pcb/signal/cds_iocells.ndx
```

Analog Output Models

An Analog Output model characterizes a driver pin on an analog device. In Analog Output models, you specify Cadence Analog Workbench (AWB) wave files for rising and falling edges, pulses, and inverted pulses to describe the behavior of the driver pin.

Editing an Analog Output Model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select *AnalogOutput* in the Model Type Filter list.
3. Select an *AnalogOutput* model and click *Edit*.

The Analog Output Model Editor appears with the current data for the selected model.

Note: SigWave also launches, so you can view the waveform files that you are loading.

4. In the Analog Output Model Editor, specify a resistance value for a series resistor in the *Series Resistance* text box.
5. Specify the paths to one or more AWB wave files in the *Rise*, *Fall*, *Pulse*, or *Inv Pulse* text boxes.

- or -

Click on the *Rise*, *Fall*, *Pulse*, and *Inv Pulse* buttons with the text boxes empty to display a File browser that enables you to select AWB wave files to load.

6. When the paths to the wave files are displayed, click on the *Rise*, *Fall*, *Pulse*, and *Inv Pulse* buttons to load the specified AWB files.

The SigWave window shows you the waveforms for the AWB wave files in the model.

7. Click *OK*.

The Analog Output model is updated with the specified changes.

Cable Models

A Cable model is similar to a PackageModel. Both contain RLGC matrices. However, you insert a Cable model into a DesignLink model and you insert a PackageModel into an IBIS Device model.

Note: Before you create a new model, be sure that the library you want to add it to is designated as the *working* library.

Creating a Cable model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click the *Add-> button and then select Cable*.

A dialog box appears.

3. Enter the name of the model in the *New Cable model name* text box, then click *OK*.

The Cable model is created and added to the SI Model Browser list box.

Editing a Cable model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select *Cable* in the Model Type Filter list.

3. Select a Cable model and click *Edit*.

Your default text editor appears displaying the model syntax.

4. Edit the syntax to modify the model, then choose *File – Save* in the text editor to save the changes.

DesignLink Models

Note: Before you create a new model, be sure that the library you want to add it to is designated as the *working* library.

Creating a DesignLink model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click the *Add-> button and then select DesignLink*.

A dialog box appears.

3. Enter the name of the model in the *Model Name* text box, then click *OK*.

The DesignLink model is created and added to the Model Browser list box.

Editing a DesignLink model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select *DesignLink* in the Model Type Filter list.
3. Select a *DesignLink* model to edit in the SI Model Browser list box, then click *Edit*.

The System Configuration Editor appears with the current data for the selected model.

4. Modify the DesignLink parameters as desired, then click *OK*.

The model is updated with the specified changes.

ESpice Device Models

ESpice device models are models of discrete devices, which are written in a `.subckt` SPICE declaration.

Note: Before you create a new model, be sure that the library you want to add it to is designated as the *working* library.

Creating an ESpice device model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click the *Add-> button and then select ESpiceDevice*.

The Create ESpice Device Model dialog box appears.

3. Enter a name in the *Model Name* text box.
4. Click to display a menu of discrete device types in the *Circuit Type* field.

A menu appears.

5. Select one of the circuit type options; *Resistor*, *Capacitor*, or *Inductor*.
6. Specify an appropriate value in the *Value* text box. For example, specify a resistance value for a resistor.
7. Enter pin names in the *Single Pins* text box. Single pins have only one connection inside the package. The other type of pin (a common pin) has more than one connection inside the package.

Important

Be careful not to include a space in the pin name. Otherwise, a model with a double pin count is created.

8. Enter a pin name in the *Common Pin* text box. Common pins are typically the pins in a package that connect to power or ground.

For example, a `SIP8` resistor pack can have seven resistors in its IC that can be designed to be pullups or pulldowns. In the resistor pack's model there is one common pin through which all seven resistors in the IC connect to power or ground and seven single pins that connect the interconnect in the design to the resistors in the IC.

Note: If the model has no common pin, leave the field blank.

9. In the *PinCount* text box, enter the number of physical pins in the package.

10. Click *OK*.

The ESpice device model is created and added to the Model Browser list box.

Editing an ESpice device model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select *ESpiceDevice* in the Model Type Filter list.

3. Select an `ESpiceDevice` model and click *Edit*.

Your default text editor appears displaying the model syntax.

4. Edit the syntax to modify the model, then choose *File – Save* in the text editor to save the changes.

IBIS Device Models

IBIS Device models are assigned to ICs and connectors with the `SIGNAL_MODEL` property. An `IbisDevice` model for a connector has package parasitics but no `IOCell` models.

Note: Before you create a new model, be sure that the library you want to add it to is designated as the *working* library.

Creating an IBIS Device model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click the *Add-> button and then select IbisDevice*.

The Create IBIS Device Model dialog box appears.

3. Enter a name in the *Model Name* text box.
4. Enter the number of pins in the model in the *Pin Count* text box.
5. Enter package pin parasitic values in the *Pin Parasitics* R, L, and C text boxes.

The values that you enter here apply to all pins in the model. If you need different parasitic values for some pins, you can change them by editing the model in the IBIS Device Model Editor.

6. Enter IOCell models in the *IOCell Model* text boxes.

The simulator fills these fields with the default IOCell models you specified in the Signal Analysis Parameters dialog box. If you want the model to use IOCells other than your default IOCell models, enter these IOCell models here.

7. Enter in the *Pins* text boxes (to the right of the IOCell Model fields) the names (pin numbers) of the pins that use these models and enter the names of the power and ground pins.
8. Click *OK*.

The model is created and added to the Model Browser list box.

Editing an IBIS Device model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select an `IbisDevice` model to edit in the Model Browser list box, then click *Edit*.

The IBIS Device Model Editor appears with the current data for the selected model.

3. Use the three tabs of the IBIS Device Model Editor to edit the model as desired.

Use the *Edit Pins* tab to modify information about the pins associated with the IBIS Device model.

Use the *Assign Power/Ground Pins* tab to group power and ground pins and assign them to power and ground buses or to auto-assign buses to individual pins.

Use the *Assign Signal Pins* tab to group signal pins and assign them to IOCell models and buses.

4. When your edits are complete, click *OK*.

The device model is updated with the specified changes.

Adding a pin to an IBIS Device Model

1. Choose Analyze – Model Browser.

The SI Model Browser dialog box appears.

2. Select the `IbisDevice` model to edit in the Model Browser, then click *Edit*.

The IBIS Device Model Editor appears with the current data for the selected model.

3. In the *IBIS Device Pin Data* area, click *Add Pin Data*.

A prompt appears.

4. Enter a name for the new pin, then click *OK*.

The new pin is added to the *IBIS Pin Data* list box.

Editing the pin data for an existing pin on an IBIS Device model

1. Choose Analyze – Model Browser.

The SI Model Browser dialog box appears.

2. Select the `IbisDevice` model to edit in the Model Browser, then click *Edit*.

The IBIS Device Model Editor appears with the current data for the selected model.

3. In the *IBIS Pin Data* list box, click to select the individual pin you want to edit.

The IBIS Device Pin Data dialog box appears with the current data for the specified pin.

Note: The IBIS Device Model Editor remains open in the background. If you click a different pin, the IBIS Device Pin Data dialog box changes to display data for that pin.

4. Modify the pin data as desired, then click *OK*.

The pin is updated with the specified changes.

IOCell Models

Note: Before you create a new model, be sure that the library you want to add it to is designated as the *working* library.

Creating an IOCell model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click *Add->*, then select one of the following model types from the pop-up menu.

- ☐ IbisIO
- ☐ IbisIO_OpenPullUp
- ☐ IbisIO_OpenPullDown
- ☐ IbisOutput
- ☐ IbisOutput_OpenPullUp
- ☐ IbisOutput_OpenPullDown
- ☐ IbisInput
- ☐ IbisTerminator

A dialog box appears.

3. Enter a name for the model, then click *OK*.

A new IOCell model of the type you selected is created using default values and its name is added to the Model Browser list box.

Editing an IOCell model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select one of the following device model types to edit in the Model Browser list box, then click *Edit*.)

- ☐ IbisIO
- ☐ IbisIO_OpenPullUp
- ☐ IbisIO_OpenPullDown
- ☐ IbisOutput
- ☐ IbisOutput_OpenPullUp
- ☐ IbisOutput_OpenPullDown
- ☐ IbisInput
- ☐ IbisTerminator

The IOCell Editor appears with the current data for the selected model.

3. Use the four tabs of the IOCell Editor to modify the model data.

Use the *General* tab to describe the model. (You can invoke the VI Curve editor for PowerClamp and GroundClamp VI curves from this tab.)

Use the *Input Section* tab to describe the high and low logic thresholds for an input buffer.

Use the *OutputSection* tab to describe the rise and fall times for an output buffer. (You can invoke the VI Curve editor for PullUp and PullDown VI curves from this tab. You can invoke the VT Curve editor for RisingWave and FallingWave VT curves from this tab.)

Use the *Delay Measurement* tab to describe the test fixture and the measurement threshold (Vmeasure) used for buffer delay measurement.

4. When your edits are complete, click *OK*.

The IO Cell model is updated with the specified changes.

Package Models

A Package model is similar to a Cable model. Both contain RLGC matrices. However, you insert a Cable model into a DesignLink and you insert a Package model into an IBIS Device model. Cadence recommends that you create new Package models by cloning an existing Package model from the sample library and editing that copy to characterize the device you are modeling.

Note: Before you create a new model, be sure that the library you want to add it to is designated as the *working* library.

Procedures

Creating a Package model by copying and editing an existing model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. In the Model Browser list box, highlight the `PackageModel` you want to copy, then click *Edit*.

Your default text editor opens with the contents of the Package model.

3. Edit the syntax to modify the model, then choose *File – Save As* in the text editor to save the file as a new Package model.

Editing a Package Model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select an `PackageModel` to edit in the Model Browser list box, then click *Edit*.

Your default text editor opens displaying the model syntax.

3. Edit the syntax to modify the model, then choose *File – Save* in the text editor to save the changes.

Adding a Package Model to an IBIS Device Model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select an `IbisDevice` model to edit in the Model Browser list box, then click *Edit*.

The IBIS Device Model Editor appears with the current data for the selected model.

Allegro SI SigXplorer Reference

Analyze Menu Commands

3. In the Model Browser (still open in the background), use the browser to find and click the `PackageModel` you want to assign to the IBIS device model. The model appears in the *Package Model* field of the IBIS Device Model Editor dialog box.

4. Click *OK* in the IBIS Device Model Editor.

The `PackageModel` is added to the `IbisDevice` model.

Working with Interconnect Models

Note: See the *Allegro PCB SI User Guide* for information regarding the Interconnect Description Language (IDL) and the formats used for interconnect models.

Trace, MultiTrace, Pin or Shape Models

Note: Before you create a new model, be sure that the library you want to add it to is designated as the *working* library.

Editing a Trace, MultiTrace, Pin or Shape model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select a model to edit from the Model Browser list box, then click *Edit*.

Your default text editor opens displaying the model syntax.

3. Edit the syntax to modify the model, then choose *File – Save* in the text editor to save the changes.

Analyze – Via Setup Preferences

Dialog Boxes

Via Model Setup

The settings in this dialog box determine how to extract and model vias for simulation.

<i>Model Solver</i>	The selected model solver. When you select FSvia, the <u>output formats</u> for single vias are enabled.
---------------------	--

<i>Model Option</i>	Activates specific dialog box options according to the model generation type selected, <i>Closed Form</i> , <i>Detailed Closed Form</i> , or <i>Analytical Solution</i> , when you select FSVia.
---------------------	--

*Output Format for
Single Vias*

- ***S Parameter Circuit*** Specifies that S Parameter syntax be used in the via model output format. This is the only format that supports coupled via models.

Format Details:

- The most accurate via format. Accurately captures the via behavior over the entire frequency range.
- Expect slower simulation performance over circuit-based formats as more processing is required.
- Start Frequency for multi-gigahertz applications is recommended at 10MHz.
- If DC convergence issues occur, you can drop to 1MHz (but no lower than 0.1MHz).
- End Frequency should be about $2/t_{\text{rise}}$ ($1/t_{\text{rise}}$ minimum). Go up to $5/t_{\text{rise}}$ for greater accuracy, similar to when you use a fine waveform resolution like 5ps or 10ps.
- No. of Freq Points should be 128 points for most via models (this is the default value)

Note: If you are to include S Parameter via models in larger S Parameter circuits, their accuracy must be similar to that of the desired final circuit. Otherwise, unpredictable results may occur.

Allegro SI SigXplorer Reference

Analyze Menu Commands

- *Wide Band Equivalent Circuit* Specifies the use of wideband equivalent circuit syntax in the via model output format.
Format Details:
 - *Start Frequency* for MGH applications recommended at 10MHz.
 - *End Frequency* should be about $2/t_{\text{rise}}$ ($1/t_{\text{rise}}$ minimum). Set to $5/t_{\text{rise}}$ for greater accuracy (similar to when you use a fine waveform resolution like 5ps or 10ps).
 - Leaving *Approx Order* set to 10 is recommended. You can increase it to 12 if *End Frequency* goes beyond 20GHz for improved accuracy.
 - There is some loss of accuracy compared to the S Parameter format. However, simulation time is significantly faster.
 - There is some risk of instability with this type of model. Convergence issues are possible if the frequency range is stretched too far.

Allegro SI SigXplorer Reference

Analyze Menu Commands

- ***Narrow Band Equivalent Circuit*** Specifies the use of narrow band equivalent circuit syntax in the via model output format.

Format Details:
 - The narrowband model is derived from the *Target Frequency*.
 - Use a target frequency that is near the middle of the energy content.
 - Good rule of thumb is $1/(1000 \times \text{risetime})$. For a driver with 100ps rise times a target frequency of 10MHz is recommended.
 - If *Target Frequency* is too high, then low frequency (DC losses) are dramatically overestimated.
 - If *Target Frequency* is too low, then high frequency effects (skin effect and dielectric loss) are underestimated. However, these are small effects in a via.
 - This is the least accurate of the via model formats. However, it is very stable and simulates very quickly.

Frequency Dependent Parameters

- ***Target Frequency*** This field is active only when you have selected *Narrow Band Equivalent Circuit* as your via model type. The default value is 10MHz.
- ***Start Frequency*** With *Wideband Equivalent Circuit* selected, specifies the start frequency for the equivalent circuit (RLC values).

With *S Parameter* selected, specifies the frequency of the start point with respect to the *No. of Frequency Points*.

The remaining points are at equal intervals between the start frequency and end frequency.

Allegro SI SigXplorer Reference

Analyze Menu Commands

- *# of Frequency Points*
Approximate Order

When the output format is set to *S Parameters*, specifies the number of frequency points for which to generate S parameters.

When the output format is set to *Wideband Equivalent Circuit*, specifies the order of the equivalent circuit generated.

The *Approximation Order* value must be within the range of 1 to 15 inclusive. The higher the order - the more accurate the solution at the cost of processing time.
- *Reference Impedance*

Specifies the reference impedance used for generating the model.
- *End Frequency*

With *Wideband Equivalent Circuit* selected, specifies the end frequency for the equivalent circuit (RLC values).

With *S Parameter selected*, specifies the frequency of the end point with respect to the *No. of Frequency Points*.

The remaining points are at equal intervals between the start frequency and end frequency.
- *Frequency Sweep Type*

Displays the scale used for selecting the frequency points between the start and end frequencies.
- *Step Size*

View-only field that displays the frequency step time based on start and end frequencies and number of frequency points. (The recommended frequency step size is 10MHz.) Specifically, the equation used is

$$\frac{(\text{end_frequency} - \text{start_frequency})}{(\text{\#_of_frequency_points})}$$

If the number of frequency points is 1, the step size should be 0.

Triangle Mesh Level

Allegro SI SigXplorer Reference

Analyze Menu Commands

Num. of Prism Layers

Port Width Factor

*Absorbing Boundary
Conditions*

Top ABC Distance Ratio

*Bottom ABC Distance
Ratio*

Num. of ABC Layers

Sweep Type

Exact VectorFit

.snp

Analyze – Via Model Generation

Displays the Via Model Generator dialog box.

Via Model Generator Dialog Box

For multi-gigahertz designs, it is critical to model via structures accurately over a very high frequency range. You can modify existing via models in a topology or create them from scratch using the Via Model Generator dialog box. For further details on Via Modeling, see [*Allegro SI SigXplorer User Guide*](#).

The Via Model Generator is a tabbed dialog box that accommodates inputs for single and coupled via modeling, as described below.

Description Tab Controls

The controls in this tab let you define the single or coupled vias.

Option	Function
<i>Via Selection</i>	
<i>Via Type</i>	Lets you select the type of via you want to generate: <i>Single</i> <i>Coupled Signal</i> <i>Coupled Signal and Ground</i> <i>Coupled Signal and Power</i> The default selection is <i>Single Via</i> . When you select a coupled via, the appropriate controls associated with coupled vias become active.
<i>Separation (S)</i>	Lets you set the distance between the center points of the selected coupled via.
<i># of Gnd. Vias</i>	Lets you add multiple ground vias to your coupled via selection. The number you can add is dependent on the settings you establish for antipad diameter and separation distance.
<i>Layer Span Information</i>	
<i>Stackup</i>	Displays a text window containing stackup information for the source file (.brd, .tech, or .mcm) named in the adjacent text box. Once generated, the via model is based on this stackup.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Option	Function
<i>Browse</i>	Displays an Open File browser that enables you to select the source file containing the desired stackup information.
<i>Via</i>	Specifies which of the coupled vias you are defining. For example, if your selected via type is <i>Coupled Signal Vias</i> , the via selections are <i>Signal Via 1</i> and <i>Signal Via 2</i> .
<i>Copy From</i>	Active only with Coupled Signal Vias, this control lets you make a duplicate copy of the via name displayed in the <i>Via</i> control.
<i>Begin Layer</i>	Specifies the layer in the stackup where the via begins. Unless this is a blind or buried via, the <i>TOP</i> (default) layer should be specified.
<i>End Layer</i>	Specifies the layer in the stackup where the via ends. Unless this is a blind or buried via, the <i>BOTTOM</i> (default) layer should be specified.
<i>Drill Diameter</i>	Specifies the drill diameter for the via model. This is typically 2 mils larger than the finished hole size after plating. This value must be a positive (non-zero) number.
<i>Antipad Diameter</i>	Specifies the antipad diameter for the via model. This is typically the pad size plus 10 mils. This value must be a positive (non-zero) number.
<i>Pad Diameter</i>	Specifies the pad diameter for the via model. This is typically the drill size plus 10 mils. This value must be a positive (non-zero) number.
<i>Layer Type Filter</i>	Specifies the type of layer on which information is displayed. Options are Conductors, Dielectrics, and Planes.

Option

Function

Expanded View

Opens the LayerSpan Expanded View dialog box that provides a larger window from which to view layer span information. Both the expanded and regular view spreadsheets include the following:

All the layers that compose the stackup, color-coded for conductor, dielectric, plane, and surface layers. Layers are editable, depending on your selection of beginning and ending layers. The following conditions apply for the via type you want to edit:

Signal Via: You can edit *Pad*, *Connection*, *Trace Width*, and *Trace Angle* on all conductor layers.

Pads reside on all conductor layers by default.

Only your beginning and ending layers have connections.

Trace Width is the same as the layer thickness.

Trace Angle is 0 degrees.

Ground/Power Via: Checks in the *Connection* column indicate the layer is connected to Ground or Plane (dependent on via type).

The following conditions apply for all via types:

The *Pad Diameter* column lets you enter custom pad diameter values for each conductor. It's default value is the same as the value in the *Pad Diameter* field above the spreadsheet.

Changes that you make to the value of *Pad Diameter* in the spreadsheet generates a query window that asks if you want to update all the conductors to the new value.

In this release, you cannot specify multiple connections to one signal layer in the Via Model Generator. However, you can do this in the SigXplorer canvas (after a via model is generated) by connecting multiple signals to a port corresponding to a layer specified in the spreadsheet.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Option	Function
<i>Via Model Name</i>	Specifies the name of the via model. Note: The default naming schema for the via model is: <i>VIA_<LayerSpan>_<SignalConnections></i> . This allows you to identify the scenario it represents.
<i>Ok</i>	When clicked from the <i>Description</i> tab, this button prompts you to choose whether you want to generate a via model with the current settings in the <i>Modeling Options</i> tab. If you choose to generate, the dialog box closes when the simulation is complete.
<i>Apply</i>	When clicked from the <i>Description</i> tab, this button prompts you to choose whether you want to generate a via model with the current settings in the <i>Modeling Options</i> tab. If you choose to generate, the dialog box remains open after the simulation is complete.
<i>Cancel</i>	Closes the dialog box without saving any changes.

Modeling Options Tab Controls

The controls in this tab are used for generating the model after setting the conditions in the *Descriptions* tab.

<i>Model Generation Options</i>	Activates specific dialog box options according to the model generation type selected (<i>Closed Form</i> or <i>Analytical Solution</i>).
<i>Start Frequency</i>	With <i>Wideband Equivalent Circuit</i> selected, specifies the start frequency for the equivalent circuit (RLC values). With <i>S Parameter selected</i> , specifies the frequency of the start point with respect to the <i>No. of Frequency Points</i> . Note: The remaining points are at equal intervals between the start frequency and end frequency.

Allegro SI SigXplorer Reference

Analyze Menu Commands

<i>S Parameter</i>	<p>Specifies that S Parameter syntax be used in the via model output format. See S Parameter Format Details on page 172 for recommendations on setting <i>Frequency Dependent Parameters</i> as well as other information.</p> <p>Note: This is the only format that supports coupled via models.</p>
<i>Plot</i>	<p>Displays a graph of the S parameters in SigWave prior to model generation. This allows you to verify the parameters you have set. After the model is generated, the points of these parameters are added as a model to your working interconnect library.</p>
<i>Do not Regenerate</i>	<p>Allows you to look at an existing waveform or regenerate one. Active in <i>S Parameter</i> mode only.</p>
<i>Wideband Equivalent Circuit</i>	<p>Specifies the use of wideband equivalent circuit syntax in the via model output format. See Wideband Equivalent Circuit Details on page 172 for recommendations on setting <i>Frequency Dependent Parameters</i> as well as other information.</p> <p>Note: This format does not support coupled via models.</p>
<i>Narrow Band Equivalent Circuit</i>	<p>Specifies the use of narrow band equivalent circuit syntax in the via model output format. See Narrowband Equivalent Circuit Details on page 173 for recommendations on setting <i>Target Frequency</i> as well as other information.</p> <p>Note: This format does not support coupled via models.</p>
<i>Target Frequency</i>	<p>Specifies the target frequency for a narrow band via model.</p>
<i>Start Frequency</i>	<p>With <i>Wideband Equivalent Circuit</i> selected, specifies the start frequency for the equivalent circuit (RLC values).</p> <p>With <i>S Parameter</i> selected, specifies the frequency of the start point with respect to the <i>No. of Frequency Points</i>.</p> <p>Note: The remaining points are at equal intervals between the start frequency and end frequency.</p>

Allegro SI SigXplorer Reference

Analyze Menu Commands

<i>End Frequency</i>	<p>With <i>Wideband Equivalent Circuit</i> selected, specifies the end frequency for the equivalent circuit (RLC values).</p> <p>With <i>S Parameter selected</i>, specifies the frequency of the end point with respect to the <i>No. of Frequency Points</i>.</p> <p>Note: The remaining points are at equal intervals between the start frequency and end frequency.</p>
<i>No. of Frequency Points/ Approximate Order</i>	<p>When the output format is set to <i>S Parameters</i>, specifies the number of frequency points for which to generate S parameters.</p> <p>When the output format is set to <i>Wideband Equivalent Circuit</i>, specifies the order of the equivalent circuit generated.</p> <p>Note: The <i>Approximation Order</i> value must be within the range of 1 to 15 inclusive. The higher the order - the more accurate the solution at the cost of processing time.</p>
<i>Freq. Sweep Type</i>	<p>Displays the scale used for selecting the frequency points between the start and end frequencies.</p> <p>In this release, only linear scale is permitted.</p>
<i>Ref. Impedance</i>	<p>Specifies the reference impedance used for generating the model.</p>
<i>Step Size</i>	<p>View-only field that displays the frequency step time based on start and end frequencies and number of frequency points. (The recommended frequency step size is 10MHz.) Specifically, the equation used is</p> $(\text{end_frequency} - \text{start_frequency}) / (\#_of_frequency_points)$ <p>If the number of frequency points is 1, the step size should be 0.</p>
<i>Via Model Name</i>	<p>Specifies the name of the via model.</p> <p>Note: The default naming schema for the via model is: <i>VIA_<LayerSpan>_<SignalConnections></i>. This allows you to identify the scenario it represents.</p>
<i>Generate</i>	<p>Generates the via model and adds it to your working interconnect library.</p>

Via Model Formats

S Parameter Format Details

- The most accurate via format. Accurately captures the via behavior over the entire frequency range.
- Expect slower simulation performance over circuit-based formats as more processing is required.
- *Start Frequency* for multi-gigahertz applications is recommended at 10MHz.
 - If DC convergence issues occur, you can drop to 1MHz (but no lower than 0.1MHz).
- *End Frequency* should be about $2/t_{\text{rise}}$ ($1/t_{\text{rise}}$ minimum). Go up to $5/t_{\text{rise}}$ for greater accuracy, similar to when you use a fine waveform resolution like 5ps or 10ps.
- *No. of Freq Points* should be 128 points for most via models (this is the default value)

Note: If you are to include S Parameter via models in larger S Parameter circuits, their accuracy must be similar to that of the desired final circuit. Otherwise, unpredictable results may occur.

S Parameter Settings Example

Edge Rate	Start Freq.	End Freq.	Bandwidth	No. of Freq. Points
100 ps	10 MHz	20GHz	20 GHz	128

Wideband Equivalent Circuit Details

- *Start Frequency* for MGH applications recommended at 10MHz.
- *End Frequency* should be about $2/t_{\text{rise}}$ ($1/t_{\text{rise}}$ minimum). Set to $5/t_{\text{rise}}$ for greater accuracy (similar to when you use a fine waveform resolution like 5ps or 10ps).
- Leaving *Approx Order* set to 10 is recommended. You can increase it to 12 if *End Frequency* goes beyond 20GHz for improved accuracy.
- There is some loss of accuracy compared to the S Parameter format. However, simulation time is significantly faster.

- There is some risk of instability with this type of model. Convergence issues are possible if the frequency range is stretched too far.

Narrowband Equivalent Circuit Details

- The narrowband model is derived from the *Target Frequency*.
 - ❑ Use a target frequency that is near the middle of the energy content.
 - ❑ Good rule of thumb is $1/(1000 \times \text{risetime})$. For a driver with 100ps rise times a target frequency of 10MHz is recommended.
 - ❑ If *Target Frequency* is too high, then low frequency (DC losses) are dramatically overestimated.
 - ❑ If *Target Frequency* is too low, then high frequency effects (skin effect and dielectric loss) are underestimated. However, these are small effects in a via.
- This is the least accurate of the via model formats. However, it is very stable and simulates very quickly.

Via Models

For Multi-gigahertz designs, you need to characterize via structures in the interconnect over a very high frequency range. For further details on Via Modeling, see the [*Allegro SI SigXplorer User Guide*](#).

Note: Before you create a new model, be sure that the library you want to add it to is designated as the *working* library.

Creating a via model

1. Choose *Analyze – Via Model Generation*.

The Via Model Generator dialog box appears.

2. On the *Description* tab, select a via type from the *Via Type* drop-down menu.
3. If you select a coupled via type, enter a value in the *Separation (S)* field and in the *No. of Gnd. Vias* field.
4. Select the specific via to edit using the *Via* drop-down menu. The default selection for coupled vias is always the first signal via.
5. If you selected *Coupled Signal Vias* as your via type, you can use the *Copy From* button to copy one signal via to a second signal via.

Allegro SI SigXplorer Reference

Analyze Menu Commands

6. In the *StackUp* field, enter a pathname to a .brd, .tech, or .mcm file containing the desired stackup information for the via model.

- or -
7. Click *Browse* to display a File browser to search for the appropriate file containing the desired stackup information.
8. Choose the beginning and ending layers from the drop-down menus in the *Begin Layer* and *End Layer* fields. The menus are populated with a list of conductor layers from the stackup in the specified file.
9. Enter the appropriate values in the *Drill Diameter*, *Pad Diameter*, and *Anti-Pad Diameter* fields to suit the via model.

Note: These values must be set to a positive (non-zero) number.

10. Make the appropriate changes to the columns in the Layer Span Spreadsheet, as described in the [dialog box description](#) on page 127.
11. If you have already set the conditions in the *Modeling Options* tab, click *Apply* or *Ok* to begin the simulation.

- or -
12. In the Modeling Options tab, click the *Model Generation Options* drop-down menu and choose the type of via model to generate.

Closed Form (simple lumped circuit approximation).

- or -

Analytical Solution (recommended for high frequency applications greater than 1 GHz).

13. If you choose *Analytical Solution*, choose an output format for the via model. Otherwise, the format options are grayed out.

If you choose the *S-Parameter* option (the only option available for coupled via models), you can click *Plot* to display the waveform.

If you choose *Narrow Band Equivalent Circuit*, you can modify the *Target Frequency* value.

If you chose either the *S-Parameter* or *Wideband Equivalent Circuit* option, you can modify the *Frequency Dependant Parameters*.

14. Enter a name for the via model in the *Via Model Name* text box. If you leave the field blank, a name is automatically created when you generate.

Note: The default via name is `VIA_<Layer Span>_<Signal Connections>`

15. Click *Ok* or *Apply* to create the via model.

The via model is generated and added to your working interconnect library.

Editing a via model

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Click the *IML Models* tab.

3. In the *Interconnect Library Files* list box, select the interconnect library containing the via model you wish to edit by double-clicking on its entry.

The Model Browser dialog box displays a list of models in the selected interconnect library.

4. Select the via model you wish to edit from the Model Browser list box, then do one of the following:

Click *Solve* to change the via model format without changing the model geometry (for example, to change from *Closed Form* format to *Analytical Solution* format).

The Via Model Generator dialog box appears with all options disabled except *Ok/Apply*.

- or -

Click *Edit* to change one or more via model parameters.

The Via Model Generator dialog box appears with all options enabled.

5. Modify the current via model parameters in the Via Model Generator as required, then click *Ok*. See [Creating a via model](#) for further details on setting parameters.

The via model is regenerated with the specified changes.

Adding a via model as a via part

After you have generated a via model, you can add the model as a via part in the SigXplorer design canvas.

1. Select *Edit – Add Element* to open the Add Element Browser.
2. From the *Model Type Filter* control, select *Via*.

Allegro SI SigXplorer Reference

Analyze Menu Commands

The via models found in the interconnect libraries are listed

3. Optionally, you can filter the list by selecting a specific *Format* and *Type* from the appropriate drop-down menus. Additionally, you can specify model names by entering a character string (including wild cards) in the *Model Name Pattern* text box.

4. Select a via model from the list.

Your selection becomes attached to the cursor as a via symbol, which you can then place in the design canvas.

5. After you have placed the symbol (or iterations of the symbol), right-click and select *End Add* from the pop-up.

The Parameter spreadsheet will display the via output format (Closed-Form, Narrow-Band, Wide-Band, or S-Parameter) for the added via.

Analyze – Preferences

[Dialog Box](#) | [Procedures](#)

Displays the Analysis Preferences dialog box. Use this dialog box to set simulation defaults for:

- Pulse stimuli
- Simulation duration
- Waveform resolution

From the Analysis Preferences dialog box, you can also define fast/typical/slow simulations and advanced measurement parameters for glitch and eye diagram measurements.

Dialog Boxes

The Analysis Preferences dialog box consists of six tabbed dialogs as well as associated secondary dialog boxes.

Analysis Preferences Dialog Box

Pulse Stimulus tab

Use this tab to define the characteristics of the pulse stimulus.

Option	Description
<i>Measurement Cycle</i>	Defines the pulse stimulus by setting the pulse number to measure from a series of pulses. This value controls the simulation duration so that the requested number of pulses propagates before the simulation stops. The default is 1.
<i>Switching Frequency</i>	Defines the pulse stimulus by setting the frequency of the pulse stimuli for nets that have no specific pulse rate assigned. The default is 50 MHz.
<i>Duty Cycle</i>	Defines the pulse stimulus by setting the length of the high portion of the cycle as a fraction. The default, 0.5, represents equal high and low portions of the cycle.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Option	Description
<i>Offset</i>	Defines the pulse stimulus by setting the launch time offset between the primary driver and neighbor net drivers in simulations. For positive nonzero values of Offset, the neighbor drivers launch after the primary driver. The default is 0ns.

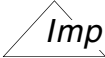
Simulation Parameters tab

Use this tab to specify how simulations are performed.

Option	Description
<i>Fixed Duration</i>	If enabled, the specified value determines the simulation duration (the length of time a simulation will run). The default is 25ns. If disabled, the simulator determines the duration dynamically for each simulation.
<i>Waveform Resolution</i>	Sets the waveform resolution as the default or as one of the specified values. Controls how many data points are generated by the simulation and how far apart they are in time.
<i>Default Cutoff Frequency</i>	Indicates the bandwidth within which interconnect parasitics are to be solved. The default is 0GHz. The specified default cutoff frequency is used by the Bem2d field solver. The Ems2d field solver also uses this value <i>unless</i> a different cutoff frequency is specified in the <u>EMS2D Preferences dialog box</u> .
<i>Buffer Delays</i>	<p>Specifies how the buffer delays are obtained for the measurement calculations. Select one of the following options from the drop-down list:</p> <ul style="list-style-type: none"><input type="checkbox"/> From Library: Specifies that the buffer delay is obtained from the model stored in the library. This is the default.<input type="checkbox"/> On-the-fly: Specifies that the buffer delay is calculated using the IBIS model's standard load circuit.<input type="checkbox"/> No Buffer Delay: Assumes 0ns buffer delays.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Option	Description
<i>Save Sweep Cases</i>	If enabled, indicates that sweep simulation waveforms and environment data save to a case directory. Note: Saving waveforms from sweeps can consume large amounts of disk space.
<i>Algorithm Model Generation</i>	This option is On by default. It enables the retrieval of algorithm-based models for use in simulation when no traditional interconnect model matching the search criteria can be found. For additional information, see Algorithm Based Modeling in the PCB SI User Guide.
<i>Simulator</i>	Allows you to choose a simulator for models. Choices are Tlsim, Hspice, and Spectre*.
<div> Important</div> <p>*Allegro PCB SI supports Spectre transistor-level models. Spectre enables simulation of Spectre transistor-level models with nets on PCB systems. The Spectre interface is supported only on Sun Solaris 8 and 9, HP UX 11.0 and 11.11i, and Linux RHEL 3.0. Spectre is not bundled with PCB PDN Analysis. Both driver and receiver models must be Spectre models wrapped in DML.</p>	
<i>Simulator Preferences</i>	Opens the Advanced Simulator Preferences dialog box for Spectre (if supported) and Hspice.
<i>Solver</i>	These options allow you to select a field solver for simulation. <i>Bem2d:</i> Specifies the Boundary Element 2.5D field solver based on static and quasi TEM conditions for single and coupled trace geometry extractions. This option does not solve for coplanar waveguides. <i>Ems2d FW:</i> Specifies the Electromagnetic Solution Full Wave field solver for coplanar waveguides. See the PCB SI User Guide section, Dynamic Analysis with the EMS2D Full Wave Field Solver , for general information on EMS2D.
<i>Solver Preferences</i>	Launches the EMS2D Preferences dialog box , from where you can set various frequency settings for the EMS2d field solver.

Advanced Simulator Preferences Dialog Box (for Spectre and Hspice)

When you select the Spectre or Hspice simulator options, you can open the *Advanced Simulator Preferences* dialog box for the selected simulator. The controls in this dialog box let you impose simulator-specific preferences in addition to generic simulator preferences.

Note: You must have the Spectre and/or Hspice simulators specified in your path, as well as any libraries used in the simulator circuits.

Option	Function
<i>Command</i>	Displays the command syntax of the selected simulator. A default command is displayed initially. You can edit this field to add/modify options.
<i>.TRAN options START =</i>	Specifies the transient sweep start time interval over which the simulation occurs. Left unset, the simulator assumes START to be zero (0).
<i>Use initial condition</i>	When checked, directs the simulator to use the initial conditions of circuit components and interconnects specified in the data statements of the model.
<i>Set Options</i>	Opens a text editor from where you can specify the <code>.options</code> statements that will be written to the beginning of the generated main simulator file. Each <code>.option</code> statement must be on a separate line

EMS2D Preferences Dialog Box

The settings in this dialog box determine how the Ems2d field solver will analyze for net extraction. (See the PCB SI User Guide section, [Dynamic Analysis with the EMS2D Full Wave Field Solver](#), for general information on EMS2D.)

Frequency Settings

Default Frequencies Directs Ems2d to use standard Bem2d settings to solve the model.

Frequency Parameters

Start Frequency Specifies the frequency of the start point with respect to the *# of Frequency Points*. The remaining points are at equal intervals between the start frequency and end frequency.

Allegro SI SigXplorer Reference

Analyze Menu Commands

<i>End Frequency</i>	Specifies the frequency of the end point with respect to the <i>No. of Frequency Points</i> . The remaining points are at equal intervals between the start frequency and end frequency. The value of this parameter overrides the value of the Default Cutoff Frequency in the analysis Preferences dialog box.
<i># of Frequency Points</i>	<p>Specifies the number of frequency points for which to generate the model.</p> <p>The maximum number of frequency points is 4096.</p>
<i>Frequency Point File</i>	<p>Lets you select a frequency point file to provide specific frequency points in GHz. Frequency point files are ASCII-text files that you create using a <code>.frequency</code> extension. The files can reside at a location of your choice. The format of the file should resemble this example:</p> <pre>0.0001 0.0002 0.001 0.002 1 2 10 20 . . .</pre>
<i>Fast Frequency Sweep (Reduced Order Model)</i>	Directs the Ems2d to scale the computation when selecting the frequency points between the start and end frequencies.
<i>Output SParameter Waveform</i>	Outputs into your signoise.run directory an S-Parameter Touchstone file for each model you generate. You can then view the wave form in SigWave.

Simulation Modes tab

Use this tab to specify how to perform a single simulation or simulation sweeps.

Option

Description

FTS Modes

Sets the simulation speed modes for the simulations. For a single simulation, select one FTS mode. For simulation sweeping by driver slew rate, select multiple FTS modes.

- ☐ Fast: Performs simulations in Fast mode.
- ☐ Typical: Performs simulations in Typical mode.
- ☐ Slow: Performs simulations in Slow mode.
- ☐ Fast/Slow: Performs simulations in Fast mode for the driver and Slow mode for the receiver.
- ☐ Slow/Fast: Performs simulations in Slow mode for the driver and Fast mode for the receiver.

Driver Excitation

Specifies the drivers to stimulate. Select one of the following from the drop-down list:

- ☐ Active_Driver: for a single simulation
- ☐ All_Drivers: for simulation sweeping where each eligible driver is active for a simulation

S-Parameters Tab

Use the S-Parameters tab to set S-Parameter transient simulations options

Option	Description
<i>Transient Simulation Method</i>	<p>The method Tlsim uses to model and simulate the S-Parameter elements in the circuit netlist. The options are:</p> <ul style="list-style-type: none">■ <i>Convolution</i>: a direct-approach frequency-domain to time-domain conversion method using N^2 complexity algorithm■ <i>Fast Convolution</i>: an approximation-approach to the direct Convolution option using $N\log N$ complexity algorithm. This faster option is useful in cases where your simulation involves high numbers of time steps.
<i>DC Extrapolation Method</i>	<p>The method Tlsim uses to globally extrapolate the low-frequency points of the S-Parameters (down to 0Hz) if they are missing.</p> <p>The options are:</p> <ul style="list-style-type: none">■ <i>Default</i>: Sequences through all options until it finds the first option that satisfies the condition.■ <i>MagPhase</i>: Extrapolates the DC values based on the magnitude and phase values.■ <i>Reallmag</i>: Extrapolates the DC values based on the real and imaginary values.■ <i>SmithChart</i>: Extrapolates the DC values based on an exact-approach method.■ <i>FirstPoint</i>: Extrapolates the DC values based on the DC value that is equal to the first non-zero point.
<i>Enforce Impulse Response Causality</i>	<p>Instructs Tlsim to use the Hilbert transform to enforce the S-Parameters impulse response causality. This option is useful when you are simulating noisy S-Parameters that are not causal; that is, do not represent a physical system.</p>

Note: You can set environment variables at the system level to direct the behavior of Tlsim when running simulations:

SetTlsimTimeStep

Examples:

```
setenv SetTlsimTimeStep 10  
setenv SetTlsimTimeStep 50
```

Description:

When set, Tlsim uses a specified time step in picoseconds for simulations.

Measurement Modes tab

Use this tab to characterize how simulation results are obtained.

Option	Description
<i>Measure Delays At</i>	<p>Specifies how the voltage threshold from delays are measured. Select one of the following from the drop-down list:</p> <ul style="list-style-type: none"><input type="checkbox"/> Input Thresholds: Specifies that delays are measured at the Input Logic Thresholds, Vil and Vih.<input type="checkbox"/> V Measure: Specifies that delays are measured at the driving IOCell's Buffer Delay Measurement Threshold, Vmeas or Vmeasure.
<i>Receiver Selection</i>	<p>Sets the receiver selection mode. Select one of the following from the drop-down list:</p> <ul style="list-style-type: none"><input type="checkbox"/> All: Reports simulation results for all receivers.<input type="checkbox"/> Select One: Reports simulation results for the selected receiver.
<i>Custom Simulation</i>	<p>Specifies the type of simulation to perform. Select one of the following from the drop-down list:</p> <ul style="list-style-type: none"><input type="checkbox"/> Reflection<input type="checkbox"/> Crosstalk<input type="checkbox"/> EMI

Option	Description
<i>Drvr Measurement Location</i>	<p>Specifies the driver location from which to compute measurement locations. Select one of the following from the drop-down list:</p> <ul style="list-style-type: none"><input type="checkbox"/> Model Defined: The default setting. The driver pin measurement location is defined by design context and the related component model.<input type="checkbox"/> Pin: The pin measurement location at the external pin node.<input type="checkbox"/> Die: The pin measurement location at the internal die node, if present.
<i>Rcvr Measurement Location</i>	<p>Specifies the receiver location from which to compute measurement locations. Select one of the following from the drop-down list:</p> <ul style="list-style-type: none"><input type="checkbox"/> Model Defined: The default setting. The receiver pin measurement location is defined by design context and the related component model.<input type="checkbox"/> Pin: The pin measurement location at the external pin node.<input type="checkbox"/> Die: The pin measurement location at the internal die node, if present.
<i>Report Source Sampling Data</i>	<p>Specifies whether or not to report source sampling data.</p>

Advanced Settings

Click this button to access the Advanced Measurement Parameters dialog box. From here, you can set measurement parameters that govern glitch tolerance and measure eye opening and peak-to-peak jitter.

Glitch Tolerance

The glitch tolerance setting is a relative percentage of the faster of the rising and falling edges of each IO cell buffer model you need to measure. When a glitch occurs between the starting and ending points of a cycle, a glitch violation is reported if the value of the glitch exceeds the

Allegro SI SigXplorer Reference

Analyze Menu Commands

tolerance percentage entered in the Glitch Tolerance field. The glitch is *not* reported as a cycle.

You can specify the glitch measurements you want to measure by selecting them in the *Reflection* category of the *Measurements* spreadsheet tab:

- *Glitch* is the tolerance check of the rising and falling waveform
- *GlitchRise* is the tolerance check on the rising waveform. If no glitch occurs in the rising waveform, the Results spreadsheet denotes a PASS in the GlitchRise column. If one does occur, it reports a FAIL.
- *GlitchFall* is the tolerance check on the falling waveform. If no glitch occurs in the falling waveform, the Results spreadsheet denotes a PASS in the GlitchFall column. If one does occur, it reports a FAIL.

Glitch tolerance values are saved in the topology file and in the `sigxp.run` case management directory. If the tolerance values in these locations differ, the tolerance in the topology file takes precedent.

Eye Diagram Measurements

To measure the eye diagrams of drivers which have a custom stimulus (that is, a stimulus other than pulse, rise, fall, etc.), eye diagram measurements report the horizontal and vertical eye opening and peak-to-peak jitter within wave forms. Following simulation, the measurements are displayed as EyeHeight, EyeJitter, and EyeWidth in the *Results* spreadsheet of the SigXplorer GUI. The Eye Diagram spreadsheet in the Set Advanced Measurements dialog box displays the current eye diagram parameter settings for the combinations of the topology's drivers and receivers.

- *Driver* and *Receiver* display the driver/receiver combinations in the topology.
- *ClockFreq* displays the value of the Custom Stimulus state set in the IOCell Stimulus Edit dialog box
- *ClockOffset* displays the value in nanoseconds of 1/2 the clock frequency value.
- *ClockStart* is editable and lets you define the point in time that the eye pattern data should start. The value default is *0ns*.

Allegro SI SigXplorer Reference

Analyze Menu Commands

EMI tab

Use this tab to set preferences and defaults for EMI single net simulation. The standard EMI preferences establish an environment appropriate for EMI simulation during design.

Option	Description
<i>EMI Regulation</i>	<p>Specifies one of six available EMI regulations against which to evaluate the design. Select one of the following from the drop-down list:</p> <ul style="list-style-type: none"><input type="checkbox"/> FCC Class A (default setting)<input type="checkbox"/> FCC Class B<input type="checkbox"/> CISPR Class A<input type="checkbox"/> CISPR Class B<input type="checkbox"/> VCCI Class 1<input type="checkbox"/> VCCI Class2 <p>The selected regulation determines which regulation curve is superimposed on the emission level in the SigWave display, and the reported pass/fail status of the emission level.</p>
<i>Design Margin</i>	<p>Sets the design margin value. The Design Margin value is subtracted from the regulation curve. It affects the graphic display of the regulation curve as well as the pass/fail status of the report. The default is 10dB.</p>
<i>Analysis Distance</i>	<p>Sets the distance between the board and the receiving antenna in the measurements setup. The Analysis Distance value takes precedence over any measurement distance specified by the regulation. The default is 3m.</p>

Fast/Typical/Slow Definitions

Click this button to access the Fast/Typical/Slow Simulations Definition tabbed dialog box. Use this to set default parameter values for fast, typical, and slow simulation speed modes. This dialog box consists of six tabbed dialogs. Each tab allows you to define corresponding parameters for Fast, Typical, and Slow simulations.

Allegro SI SigXplorer Reference

Analyze Menu Commands

General tab

Use this tab to define general simulation parameters.

Option	Description
<i>Launch Delay</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>Die Capacitance</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>Ramp Rates</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> TypSlew<input type="checkbox"/> FastSlew<input type="checkbox"/> SlowSlew

Allegro SI SigXplorer Reference

Analyze Menu Commands

Pin Parasitics tab

Use this tab to define pin parasitic parameters.

Option	Description
<i>Resistance</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>Capacitance</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>Inductance</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum

Allegro SI SigXplorer Reference

Analyze Menu Commands

Reference Voltages tab

Use this tab to define reference voltage parameters.

Option	Description
<i>Pullup</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>Pulldown</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>Power Clamp</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>Ground Clamp</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum

Allegro SI SigXplorer Reference

Analyze Menu Commands

V/I Currents tab

Use this tab to define V/I current parameters.

Option	Description
<i>Pullup</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>Pulldown</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>Power Clamp</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>Ground Clamp</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum

Allegro SI SigXplorer Reference

Analyze Menu Commands

Terminators tab

Use this tab to define terminator parameters.

Option	Description
<i>ac resistor</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>ac capacitor</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>power resistor</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum
<i>ground resistor</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow: <ul style="list-style-type: none"><input type="checkbox"/> Typical<input type="checkbox"/> Minimum<input type="checkbox"/> Maximum

Thresholds tab

Use this tab to define threshold parameters.

Option	Description
<i>Input Logic Thresholds</i>	
<i>High</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow:
<i>Low</i>	
	<input type="checkbox"/> Typical
	<input type="checkbox"/> Minimum
	<input type="checkbox"/> Maximum
<i>Buffer Delay Thresholds</i>	
<i>V measure</i>	Select one of the following from the drop-down lists for Fast, Typical, and Slow:
	<input type="checkbox"/> Typical
	<input type="checkbox"/> Minimum
	<input type="checkbox"/> Maximum

Procedures

Specifying the pulse stimulus

1. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

2. Select the *Pulse Stimulus* tab.
3. Enter the desired values for Measurement Cycle, Clock Frequency, Duty Cycle, and Offset.
4. Click *OK*.

Specifying the simulation duration

1. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

2. Select the *Simulation Parameters* tab.
3. Enable the Fixed Duration checkbox and enter the desired value in the text box.
4. Click *OK*.

Specifying the waveform resolution

1. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

2. Select the *Simulation Parameters* tab.
3. Select the desired value from the Waveform Resolution drop-down list.
4. Click *OK*.

Specifying the cutoff frequency

1. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

2. Select the *Simulation Parameters* tab.
3. Enter the desired value in the Cutoff Frequency text box.

Note: The default value of 0GHz does not denote a loss-less connection. If you take the default value, S parameters are generated and plotted with only DC interconnect loss enabled. To account for AC loss in the S parameters, set a non-zero cutoff frequency.

If you generate using 0GHz, a confirmation box displays before generation.

4. Click *OK*.

Specifying how buffer delays are obtained

1. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

2. Select the *Simulation Parameters* tab.
3. Select either From Library or On-the-fly from the Buffer Delays drop-down list.
4. Click *OK*.

Saving sweep cases

1. Choose *Analyze – Preferences* from the main menu in SigXplorer.
The Analysis Preferences dialog box appears.
2. Select the *Simulation Parameters* tab.
3. Click *Save Sweep Cases*.
4. Select other tabs as necessary to set additional preferences for simulation sweeps.
5. Click *OK*.
6. Run the simulation.

Specifying the FTS mode for a single simulation

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Select the *Simulation Modes* tab.
3. Under FTS Mode, select the single simulation speed setting you want to apply.
4. Click *OK*.

Specifying the FTS modes for simulation sweeping

Select a range of FTS Modes to sweep by driver slew rate.

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Select the *Simulation Modes* tab.
3. Under FTS Mode, select the range of simulation speed settings you want to apply.
4. Click *OK*.

Selecting a driver for a single simulation

1. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

2. Select the Simulation Modes tab.
3. Click to select the Simulation Modes tab.
4. Select *Active_Driver* from Driver Excitation drop-down list.
5. Click OK.

The simulation will be performed for the selected active driver.

Selecting drivers for simulation sweeping

1. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

2. Select the Simulation Modes tab.
3. Select *All Drivers* from the Driver Excitation drop-down list.
4. Click OK.

A sequence of simulations is performed where each eligible driver in the topology drives a simulation in turn.

Selecting receivers for simulation

1. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

2. Select the Measurement Modes tab.
3. Select either All or Select One from the Receiver Selection drop-down list.
4. Click OK.

Selecting a custom simulation

1. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

2. Select the Measurement Modes tab.
3. Select Reflection, Crosstalk, or EMI from the Custom Simulation drop-down list.
4. Click OK.

Selecting an EMI regulation

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Select the EMI tab.
3. Select the desired EMI regulation from the drop-down list.
4. Click OK.

Specifying the design margin

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Select the EMI tab.
3. Enter the desired value in the Design Margin text box.
4. Click OK.

Specifying the analysis distance

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Select the EMI tab.
3. Enter the desired value in the Analysis Distance text box.
4. Click *OK*.

Specifying Fast/Typical/Slow simulation settings

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.

2. Click Fast/Typical/Slow Definitions.
3. Select one of the following tabs: *General*, *Pin Parasitics*, *Reference Voltages*, *V/I Currents*, *Terminators*, or *Thresholds*.
4. Select the desired speed settings from the various drop-down lists.
5. Repeat steps 3 and 4, as needed, for the different tabs.
6. Click *OK*.

Specifying glitch settings

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Click *Advanced Settings*.
3. Click the *Measurement Modes* tab.
The Set Advanced Measurement Parameters dialog box appears.
4. Click the *Glitch Tolerance* tab.
5. Enter a percentage value in the *Glitch Tolerance* field.
6. Click *OK*.

Reporting and viewing eye diagram measurements

Note: Your topology must contain drivers with a custom stimulus.

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Click *Advanced Settings*.
3. Click the *Measurement Modes* tab.
The Set Advanced Measurement Parameters dialog box appears.
4. Click the *Eye Diagram* tab.
The Eye Diagram spreadsheet displays the current eye diagram parameter settings for the combinations of the topology's drivers and receivers (as described in [Eye Diagram Measurements](#) on page 186).

Allegro SI SigXplorer Reference

Analyze Menu Commands

5. In the *Reflections* section of the *Measurements* spreadsheet of the SigXplorer GUI, check *EyeHeight*, *EyeJitter*, and *EyeWidth*.
6. Run the simulation.
7. Click the *Results* tab of the SigXplorer spreadsheet to view:
 - ❑ *EyeHeight*, the value based on an imaginary vertical line at the mid-point of the eye width (see figure, below)
 - ❑ *EyeJitter*, the peak-to-peak value of the clock period minus the value of the eye width
 - ❑ *EyeWidth*, the value based on an imaginary horizontal line equal to the value of *vil* and *vih* (as measurements of input thresholds) or *Vmeas* (as measurements of output).

Setting up a Spectre/Hspice simulation

1. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

2. Set the standard preferences listed in the dialog box, as described in the [Dialog Boxes](#) sections of this topic.
3. Click the *Simulation* tab.
4. From the Simulator drop-down menu, select either Spectre or Hspice.

The *Set Simulator Preferences* button becomes active.
5. Set the control parameters for the conditions under which you want the simulator to run. These controls are described in [Advanced Simulator Preferences Dialog Box \(for Spectre and Hspice\)](#) on page 180.
6. Close the dialog box.

Running a Hspice or Spectre simulation

Upon completion of the setup for simulation, shown above, perform this procedure for running the simulation.

1. Add the path to the simulator to your user \$PATH as well as to the paths for any libraries you may use.

2. Develop DML MacroModels for the IO buffer subcircuits. The basic composition of your models include:

☐ **Name of the MacroModel**

Identical to the name of the 7-terminal subcircuit that you insert in the MacroModel section.

☐ **Body of the MacroModel**

Identical to the body of any DML buffer MacroModel. It specifies basic IO buffer information, including (but not limited to):

Rise and fall times

Logic thresholds

Model types

Test fixtures

This is illustrated in the following example of a portion of a MacroModel body:

```
(Technology CMOS)

(Model

  (ModelType IO)

  (Polarity "Non-Inverting")

  (Enable "Active-High"))

Logic Thresholds

(Output

  (High

    (typical 2.5))

  ....
```

☐ **MacroModel Subsection**

Similar to an ESPICE MacroModel subsection, the difference being that you insert a simulator-specific 7 terminal wrapper subcircuit for the IO buffer rather than an ESPICE subcircuit. You must also indicate the simulator for which the MacroModel is targeted. This is illustrated in the following example of a MacroModel subsection:

```
(MacroModel

  (NumberOfTerminals 7)

  (language "simulator_name")
```


Allegro SI SigXplorer Reference

Analyze Menu Commands

*The syntax of the subcircuit. If not specified, defaults to ESPICE syntax.

```
(SubCircuits "
```

* The Subcircuits section contains the 7-terminal subcircuit wrapper

* for the IO buffer.

```
simulator language=spice
```

```
.subckt <simulator_name>_out 1 2 3 4 5 6 7
```

*Calls the subcircuit containing the buffer's transistor-level model.

```
X_<simulator_name> 1 2 3 4 5 6 7
```

```
Any_<simulator_name>_transistor_model_subcircuit
```

```
.ends <simulator_name>_out
```

```
")
```

3. From the SI Model Browser (*Analyze – Model Browser*) load the DML libraries that contain the IO buffer MacroModels, IbisDevices, and Packages.
4. From the SI Model Browser, assign an IbisDevice to a component.
5. Edit the IbisDevice to assign the IO buffer models (or the DML MacroModel for a Spectre IO buffer) to the appropriate pins and, if necessary, to assign a package model to the IbisDevice.

Note: If no IbisDevice is assigned to a component, you must use IO buffers targeted for your simulator type as defaults.

6. Set the simulator preferences from the controls in the *Simulation* tab of the Analysis Preferences dialog box and in the Advanced Simulator Preferences dialog box.
7. Perform the simulation, then generate and view the reports and waveforms.

Setting transient simulation preferences

1. Open a .brd or .mcm database file.
2. Choose *Analyze – Preferences*.

The Analysis Preferences dialog box appears.

3. Click on the *S-Parameters* tab, select the simulation and extrapolation methods of your choice and whether you want to enable impulse response causality.

Click *OK* to save your settings and close the Analysis Preferences dialog box.

Analyze – Simulate

[Dialog Boxes](#) | [Procedures](#)

Performs a simulation or simulation sweep using the simulation parameters set with the Analysis Preferences dialog box.

Note: Simulation sweeps are available in Signal Explorer Expert only.

Dialog Boxes

Sweep Sampling Dialog Box

Option	Description
<i>Percent</i>	Specifies sweep coverage as a percentage of full coverage. The default is 100%.
<i>Count</i>	Specifies sweep coverage as an explicit number of simulations. The default is the number of simulations required for full coverage.
<i>Random Number Seed</i>	Random seed number used for partial sweep coverage. The default is 1.
<i>Continue</i>	Initiates sweep simulation.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Sweep Sampling with Case Control Dialog Box

Option	Description
<i>Percent</i>	<p>Specifies sweep coverage as a percentage of full coverage.</p> <p>The default is 100%.</p>
<i>Count</i>	<p>Specifies sweep coverage as an explicit number of simulations.</p> <p>The default is the number of simulations required for full coverage.</p>
<i>Random Number Seed</i>	<p>Random seed number used for partial sweep coverage.</p> <p>The default is 1.</p>
<i>Case</i>	<p>Displays the Case number that will contain the saved data.</p> <p>Note: When <i>SigXplorer</i> is initially invoked, it uses the default case directory (<code>case1</code>). When a sweep simulation is initiated with the <i>Save Sweep Cases</i> preference enabled, the system selects the <i>next unused</i> case as the repository for the saved sweep data.</p>
<i>Desc</i>	<p>Specifies a text description that documents the nature of this particular sweep.</p> <p>The sweep description is saved with the sweep case data.</p>
<i>Continue</i>	<p>Initiates sweep simulation and saves a sweep case.</p>

Procedures

Performing a single simulation

- Choose *Analyze – Simulate*.

The simulation begins. The Command tab becomes active and displays simulation messages. A Progress Meter appears to graphically show the progress of the simulation.

When the simulation completes, the Results tab becomes active and displays voltage and delay data. The SigWave window also launches to display waveforms for the completed simulation.

Performing a simulation sweep

1. Determine the sweep criteria that you want to use for your simulation.

You can sweep by:

- ☐ varying part parameter values (explained in the next step)
- ☐ varying driver slew rates

For details, see [Specifying the FTS modes for simulation sweeping](#) on page 195.

- ☐ sequencing active drivers

For details, see [Selecting drivers for simulation sweeping](#) on page 196.

Note: If you specify multiple criteria, SigXplorer employs a hierarchical ordering when performing the simulations. For example, if you select multiple FTS Modes as well as several part parameter values for sweeping, then all part parameter sweeps will be executed for each selected FTS Mode. Additionally, if you also select *All_Drivers*, then part parameter sweeps for each selected FTS Mode will execute as each driver activates in sequence.

2. Select and edit any part parameter values to be swept.

- a. Click on a part parameter in the topology canvas.

The part parameter is highlighted in the Parameters spreadsheet

- or -

Click on the part parameter directly in the Parameters spreadsheet.

- b. Click on the down-arrow icon that appears next to the parameter value in the spreadsheet.

The Set Parameter dialog box appears.

- c. Edit the sweep setup for the parameter using a linear range of values or a list of discrete values. You can also choose to use an expression string that references other parameters (count value is determined by the parameters used in the expression).
- d. Repeat the previous three steps for all other part parameters involved in the simulation sweep.

3. Choose *Analyze – Simulate*.

The Sweep Sampling dialog box appears displaying the total number of simulations required to fully analyze the topology.

4. Use the *Percent* or *Count* text box to specify full or partial sweep coverage.

5. If you specified partial sweep coverage, enter a number in the *Random Number Seed* text box.

Partial sweep coverage is obtained by randomly sampling the full solution space using Monte Carlo methods. To vary sample point sets, SigNoise selects sweep count points based on the random number seed that you specify.

6. Click *Continue* to invoke the sweep.

SigNoise is initialized, the simulations commence, and a sweep report is displayed in the Results spreadsheet.

Saving sweep cases

1. Choose *Analyze – Preferences* from the main menu in SigXplorer.

The Analysis Preferences dialog box appears.

2. Select the Simulation Parameters tab.

3. Click *Save Sweep Cases*.

4. Select other tabs as necessary to set additional preferences for simulation sweeps.

5. Click *OK*.

Allegro SI SigXplorer Reference

Analyze Menu Commands

6. Choose *Analyze – Simulate* to run a simulation sweep.

The Sweep Sampling with Case Control dialog box appears enabling you to save a case containing data pertinent to the current sweep.

Note: Once saved, sweep cases can be restored as needed for comparative analysis. For details on restoring sweep cases, see [File – Import – Sweep Case](#).

Analyze – Simulate Continue

Continues a paused simulation sweep using the simulation parameters set with the Analysis Preferences dialog box.

Analyze – [S] Generation

Generates S-Parameter data for use in time domain analysis.

[Dialog box](#) | [Procedures](#)

Dialog Boxes

S - Parameter Generation Dialog Box

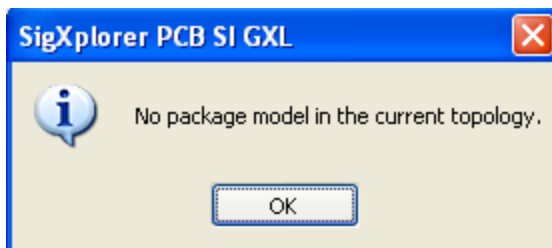
Option	Description
<i>Start Frequency</i>	Enter a value to specify the start frequency. The default is 0Hz. Note: The start frequency must be less than the end frequency.
<i>End Frequency</i>	Enter a value to specify the end frequency. The default is 20GHZ. Note: The end frequency must be greater than the start frequency.
<i>Number of Frequency Points</i>	Enter the number of points in the frequency range. The default is 2048 points.
<i>Step Size</i>	View-only field that displays the frequency step time based on start and end frequencies and number of frequency points. (The recommended frequency step size is 10MHz.)
<i>Frequency Sweep Type</i>	Select a frequency sweeping type from the pulldown menu. The default is <i>Linear</i> .
<i>Reference Impedance</i>	Enter the impedance for the S-Parameters output. The default is 50ohm.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Include Package Models in S-Parameter Model

Select this option if the current topology contains package models. If there are no package models in the topology and you select this option, the following message appears:



A package model is included in one of the following ways:

- The ports are moved from the package pins to the die side of the package and the s-params are then generated.
- The ports remain at the package pins and the die side is left floating.
- The ports remain at the package pins and the die side is grounded or terminated.

Note: If the *Include Package Models in S-Parameter Model* option is enabled, the *Substitute With the Generated S-Parameter* option is not available.

Set S-Parameter Ports

Click *Add* to automatically add ports for:

- IOCells
- Non-zero voltage source
- Diodes
- Nodes

If you click *Edit Port*, the Port - Editing dialog box appears (see [Port – Editing Dialog Box](#) on page 210).

For more information on port setting, see the [Allegro SI SigXplorer User Guide](#).

Model

Enter a model name to store S-Parameter Touchstone and ESpice model data.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Substitute With the Generated S-Parameter

When checked, SigXplorer updates the topology and places the S-Parameter black box according to the designated port settings. The default is unchecked.



Tip

When using the *Substitute with the Generated S-Parameter* switch, all the elements that are not properly terminated with a port will be included in your S-Parameter model. You typically place S-Parameter ports at active elements, such as IOCells, sources, and diodes to isolate these active elements. Placing a port at such elements terminates that node using the Reference Impedance value.

You must be careful to isolate only that part of the topology that you want to substitute; otherwise, you will include undesired elements into the S-Parameter model. If the resulting S-Parameter model contains unconnected nodes, the resulting simulations will be inaccurate.

Save the Current Topology

This option only appears when *Substitute With the Generated S-Parameter* box is checked.

Port – Editing Dialog Box

If you select *Edit Port*, this dialog box appears. Use it to manually add, modify, or delete ports.

Option

Description

Reset All Ports

Auto delete all ports from the topology

Add

Manually add any non-existing ports wherever you want on the topology.

Modify

Edit port names.

Delete

Delete all existing ports.

Procedures

Generating S-Parameters

1. Using SigXplorer, open a topology (either create one on-the-fly or extract an existing topology from Allegro PCB Editor).
2. Choose *Analyze – [S] Generation*.

The **S-Parameter Generation** dialog box appears.

3. Enter values in the fields for:
 - ☐ Start Frequency
 - ☐ End Frequency
 - ☐ Number of Frequency Points
 - ☐ Frequency Sweep Type
 - ☐ Reference Impedance

Note: See description of [S - Parameter Generation Dialog Box](#) on page 208.

4. Click the *Include Package Models into S-Parameter Model* if the topology contains package models.

Note: Opting to include package models disables the option to *Substitute with the Generated S-Parameter*. The S-Parameter model will include those package models.

5. Click *Add Ports* for each element to automatically generate ports or click *Edit Ports* to manually set them.

Note: If you click *Edit Ports*, the **Port - Editing** dialog box appears. For more information, see [Port – Editing Dialog Box](#) on page 210.

6. Enter a model name in the *Model* field to create a directory to store the S-Parameter and ESpice model data.

Note: After generation, the S-Parameter, ESpice data, and a backup topology file appear with this model name in the current working directory.

7. Click *Substitute with the Generated S-Parameter* to generate the S-Parameter data.

The topology updates, and the generated S-Parameter black box element appears on the canvas according to the port settings.

For more information on generating S-Parameter data, see the [Allegro SI SigXplorer User Guide](#).

Analyze – Reset Sim Data

Resets the simulation environment and reloads the libraries.

Allegro SI SigXplorer Reference

Analyze Menu Commands

Help Menu Commands

Help – Documentation

Opens the User Materials page for SigXplorer. From this page, you can access help documents available on SigXplorer and related tools in the Cadence Help viewer.

Help – What's New

Opens the What's New in release document for the current release in the Cadence Help viewer.

Help – Web Resources

Provides access to Web resources available for SigXplorer. This menu includes the following options:

Help – Web Resources – Online Support

Opens the in a web browser. Cadence Online Support is the online customer support web site for Cadence software users.

Help – Web Resources – Education Services

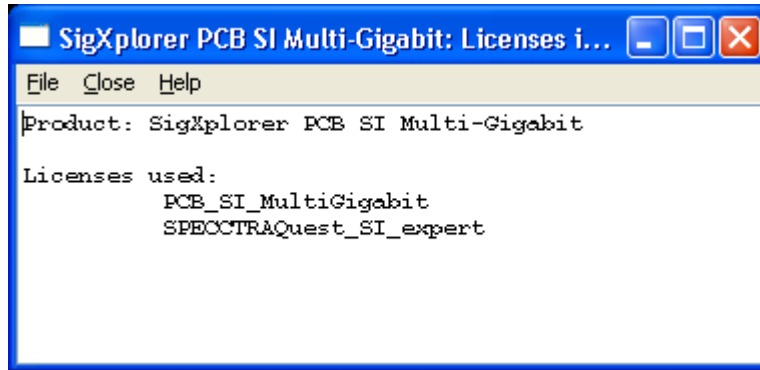
Opens the Cadence Education Services web site in a web browser. The Education Services web site provides information on training courses and related services from Cadence.

Help – Web Resources – Design Communities

Opens the Cadence PCB Design Community web page.

Help – Licenses Used

Displays information about the current license being used for the product.



Help – About

Displays the About dialog box for Allegro SigXplorer. It shows the version number of the installed product, and the information about copyright and patents.

Context-Sensitive Menus

Editing a Custom Measurement

When you open the Measurement Expression Editor to edit an existing expression, the name of the expression appears in both the Expression Name field and in the banner. The current contents of the expression appear hierarchically in the treeview window.

As the editor opens, the stored expression is parsed and checked. An error message and an empty expression are displayed if the expression fails to parse.

The Expression Name and Description Fields

Enter a name for the expression in the Expression Name field. The expression is saved under this name. When you rename an expression, the expression is saved under the new name from the time you rename it. The original expression still exists in its original form under the original name. You can delete the original expression if you no longer need it.

The Expression Name field displays up to 46 characters. You can enter up to 256 characters which can include both lowercase and uppercase alphabetic characters, numeric digits, and the underscore (_) character. You will not be able to save an expression whose name contains characters other than these. You cannot begin a measurement expression name with a number. Recognition of measurement names is case sensitive.

Enter an optional, brief description of the expression in the Description field. What you enter here should help you to remember what the expression calculates. The Description field displays up to 46 characters. You can enter up to 256 characters which can include all characters except the double-quote (") character.

The Expression Treeview Window

The expression is displayed hierarchically in the treeview window. The treeview can display the following items: placeholders, numbers, functions, parameter names, and measurement names.

Each item added to the treeview is initially a placeholder with a name ending in _TBD. For a new expression, the treeview contains one placeholder, VALUE_TBD.

The Editing Pull-Down Menu

The editing pull-down menu contains most of the editing options you will use to edit measurement expressions in the treeview. Display the editing menu by right-clicking on objects in the treeview. As you edit the expression, the content of the editing menu adjusts dynamically depending on which item you click in the treeview. As you select items in the treeview, the buttons on the dialog box are also enabled and disabled in response to what object is selected.

When you right-click a placeholder, the pull-down menu displays eligible replacements for the placeholder. Select a replacement from the pull-down menu. The treeview changes to display the modified expression.

For example, click the right mouse button on `VALUE_TBD` to display the pull-down menu of replacements. Click the right mouse button to select `CrossingTime` from the pull-down menu. The treeview changes to replace `VALUE_TBD` with the following `CrossingTime` waveform function and its five arguments:

CrossingTime

- `PIN_TBD`
- `NODE_TBD`
- `THRESH_VOLT_TBD`
- `EDGE_TBD`
- `CROSSING_NUMBER_TBD`

Arguments to functions display subordinate to the function names.

A numeric argument of a known type, for example the threshold voltage, is displayed with the appropriate units as specified in the Allegro SI Analysis Parameters dialog box in the Units tab.

Expanding and Collapsing the Treeview

Click the left mouse button on the `[+]` and `[-]` controls to expand `[+]` and collapse `[-]` items in the treeview. When you add a new item to the treeview it is initially expanded. When you open an existing expression, it is initially expanded. You can collapse any item to display more of a large expression.

Navigating the Treeview with the Keyboard Arrow Keys

Use the keyboard up and down arrow keys to move the cursor through the items and arguments in the treeview. As you move the cursor to an item or argument, it is selected. The buttons on the dialog box are enabled and disabled in response to what is selected.

Use the left and right arrow keys to navigate steps in the hierarchy and to expand and collapse the treeview. The left arrow key expands an item with a [+] icon and moves down one argument from an item with a [-] icon. The right arrow key collapses an item with a [-] icon and moves up from an argument to its item.

The Check Button

Use the Check button anytime during your editing session to statically check the validity of the expression you are constructing.

To be valid, an expression must pass the following checks.

- No placeholders (*_TBD items) can exist in the expression.
- All numeric items in the expression must be numbers, recognized parameters, measurements, or function calls.
- Non-numeric items are also checked.

Error and warning messages display in the status bar. If possible, the item in error is selected.

The Add Operand Button

Use the Add Operand button to add an operand to a math function that accepts an arbitrary number of operands. This button is enabled only when one of the following math functions is selected in the treeview: Sum, Difference, Product, Quotient, Min, and Max. These functions have two operands when you create them but you can add more.

The Clear Item Button

Use the Clear Item button to remove data from a selected item in the treeview.

- For a math function with 3 or more added optional arguments, the optional arguments are removed from the selected item.
- For all other items, the selected item is replaced with the appropriate placeholder (*_TBD).

The Undo and Redo Buttons

The Undo and Redo buttons work on edits made in the treeview only.

- Use Undo to restore the expression to its state before the last edit.
- Use Redo to reverse the effect of the most recent Undo.

You can use Undo repeatedly to reverse all changes made to the expression during the current editing session. Undo is enabled if there are edits to undo. Redo is enabled following an Undo up until you make an edit.

The Cancel and OK Buttons

When you use OK, a check is performed on the expression before it is saved. If the check fails, you are prompted whether you want to continue editing to resolve the problem or to close the dialog box without saving the expression. If you choose to continue editing, the dialog box remains open with the first check error displayed in the status bar.

When you use Cancel, the dialog box closes without altering the expression. No changes are saved.

The Status Bar

The Status Bar at the bottom of the Measurement Expression Editor dialog box displays error and warning messages produced by using the Check button.

Procedures

Editing an Existing Custom Measurement

- In the Measurement tab of the SigXplorer spreadsheet, click in the Description cell for the named custom measurement expression you want to edit. Then, click the icon at the right end of the Description cell.

or
- In the Measurement tab of the SigXplorer spreadsheet, right-click in the Name or Description cell for the named custom measurement expression you want to edit. Then, select Edit from the pull-down menu.

The editor opens displaying the selected expression in the Expression Treeview window. The Check, Add Operand, and Clear Item buttons are active.

Creating a New Custom Measurement

1. In the Measurement tab, click the right mouse button to display a pulldown menu.
2. In the pulldown menu, select Add to open the Measurement Expression Editor.

The editor opens empty other than entry for VALUE_TBD in the Expression Treeview window. The Check and Clear Item buttons are active.

The Measurement Expression Editor Dialog Box

Use this dialog box to create and modify customized measurement expressions for SigXplorer.

Options and Buttons	Description
Expression Name field	Displays the name under which the expression is saved. (The expression name also displays in the banner.)
Description	Displays text describing the expression.
Expression Treeview	Displays the expression hierarchically in treeview format.
Check	Verifies whether the measurement expression is valid.
Add Operand	Adds an operand placeholder to the selected function.
Clear Item	Replaces an operand argument in an expression with an operand placeholder (for the 1st or 2nd term) or removes an operand argument (the 3rd or greater term).
Undo	Restores the expression to its state before the most recent edit.
Redo	Reverses the effect of an Undo operation.
OK	Performs a check operation and, if the check passes, closes the editor and saves the expression. If the check fails, you can continue editing or close the form without saving.

The Expression Editing Pull-Down Menu

The Editing Pull-down Menu

When you right-click on an object in the treeview, a pull-down menu of editing options displays. The menu contains only those choices appropriate for the selected object. Based on the choice you make from the pull-down menu, the selected object in the treeview is replaced with the selected menu item. If the selected menu item contains placeholders, repeat this process to define the remaining placeholders.

Releasing the right mouse button on one of the editing options on the pull-down menu performs the action indicated in the following table.

Object selected in the treeview	Popup menu choices displayed	Action associated with making each choice
VALUE_TBD	Number	A dialog box appears for entering a number.
	DefinedMeasurement	Item changes to MEASUREMENT_TBD
	PinParameter	Item changes to the PinParameter function, with arguments.
	<waveform_functions>	Item changes to the selected function, with arguments.
	<math_functions>	Item changes to the selected function, with arguments.
	<simulation_functions>	Item changes to the selected function.
MEASUREMENT_TBD	<measurement names>	Item changes to the name of the selected standard or user-defined measurement.
PIN_TBD (argument of a waveform or PinParameter function)	Driver, Receiver, Strobe, or a pin name.	Item changes to Driver, Receiver, Strobe, or to the selected pin name.

Allegro SI SigXplorer Reference

Context-Sensitive Menus

NODE_TBD (argument of a waveform function)	PkgPin or DiePad	Item changes to the selected node name.
	MacroModel node names for the selected pin.	Item changes to the selected node name.
CROSSING_NUMBER_TBD (argument of a CrossingTime waveform function)	FIRST	Item changes to FIRST.
	LAST	Item changes to LAST.
	A number 2 through 9.	Item changes to the selected number.
START_TIME_TBD (argument of a waveform function)	Same choices as VALUE_TBD.	Same actions as VALUE_TBD.
STOP_TIME_TBD (argument of a waveform function)	Same choices as VALUE_TBD.	Same actions as VALUE_TBD.
THRESH_VOLT_TBD (argument of a waveform time function)	Same choices as VALUE_TBD.	Same actions as VALUE_TBD.
EDGE_TBD (argument of a waveform function)	Rise, Fall, RiseOrFall, PreviousRise, PreviousFall, PreviousRiseOrFall, NextRise, NextFall, or NextRiseOrFall.	Item changes to the selected edge name. Previous and Next items measure edges in the previous clock cycle and the next clock cycle.
PARAMETER_TBD (argument of a PinParameter function)	<pin_parameter_names>	Item changes to the selected pin parameter name.
A number	Same choices as VALUE_TBD.	Same actions as VALUE_TBD.
A math function	Same choices as VALUE_TBD.	Same actions as VALUE_TBD.
A waveform function	Same choices as VALUE_TBD.	Same actions as VALUE_TBD.

The Pull-Down Menu Choices

The following choices are available from the editing pull-down menu

- Numbers
- Measurements
- Math Functions
- Waveform Functions
- Pin Parameter Function
- Simulation Functions

Numbers

When you select Number from the pull-down editing menu, a dialog box displays where you enter a number.

Measurements

When you select MEASUREMENT_TBD in the treeview, or any other placeholder that can be replaced with a measurement, the pull-down menu displays the standard measurements followed by the available custom measurement expressions arranged alphabetically. To avoid recursive expressions, the measurement you are currently editing is unavailable, and thus, is omitted from the list.

When you select a measurement from the popup, it replaces the selected placeholder in the expression. You can use measurements to replace any placeholder except for the pin, node, and edge arguments to waveform functions.

Note: To insert driver buffer delay into a custom measurement expression, use a pinRiseDly or pinFallDly PinParameter for the Driver Pin.

The *DefinedMeasurement* popup omits the BufferDelayRise and BufferDelayFall measurements from the Reflection set because these would be evaluated for the Receiver and are likely to produce "NA" as a result. You may create your own custom BufferDelayRise and BufferDelayFall measurements, but be careful to create them as a Driver PinParameter.

Math Functions

The following mathematical functions are available from the pull-down menu for use when creating custom measurement expressions. The Sum, Difference, Product, Quotient, Min, and Max functions are added with two arguments. Use the Add Operand button to add additional arguments to these functions.

Function	Description	Arguments added
Sum	Add two or more values ($a + b + c \dots$).	VALUE_TBD
Difference	Subtract two or more values ($a - b - c \dots$).	VALUE_TBD
Product	Multiply two or more values ($a * b * c \dots$).	VALUE_TBD
Quotient	Divide two or more values ($a / b / c \dots$).	VALUE_TBD
Min	Minimum of 1 or more values.	VALUE_TBD
Max	Maximum of 1 or more values.	VALUE_TBD
Abs	Absolute value of the argument.	VALUE_TBD

Waveform Functions

The following waveform measurement functions are available from the pull-down menu for use when creating custom measurement expressions.

Function	Description	Arguments added
CrossingTime	Nth time at which a waveform crosses a voltage threshold, within the rise or fall portion of a cycle.	PIN_TBD NODE_TBD THRESH_VOLT_TBD EDGE_TBD CROSSING_NUMBER_TBD
RangeCrossingTime	Nth time at which a waveform crosses a voltage threshold, during a time window.	PIN_TBD NODE_TBD START_TIME_TBD STOP_TIME_TBD THRESH_VOLT_TBD EDGE_TBD CROSSING_NUMBER_TBD
VoltageAtTime	Voltage on a waveform at the given time.	PIN_TBD NODE_TBD TIME_TBD

Allegro SI SigXplorer Reference

Context-Sensitive Menus

RangeMaxVoltage	The maximum voltage seen on a waveform during a time window.	PIN_TBD
		NODE_TBD
		START_TIME_TBD
		STOP_TIME_TBD
RangeMinVoltage	The minimum voltage seen on a waveform during a time window.	PIN_TBD
		NODE_TBD
		START_TIME_TBD
		STOP_TIME_TBD
StateMaxVoltage	The maximum voltage seen on a waveform during the high or low state.	PIN_TBD
		NODE_TBD
		STATE_TBD
StateMinVoltage	The minimum voltage seen on a waveform during the high or low state.	PIN_TBD
		NODE_TBD
		STATE_TBD

Pin Parameter Function

The PinParameter function is available from the pull-down menu for use when creating custom measurement expressions. Use this function to obtain numeric data for pins.

Function	Description	Arguments added
PinParameter	Lookup a pin or IOCell parameter in the context of a pin.	PIN_TBD PARAMETER_TBD

Simulation Functions

The following simulation functions are available from the pull-down menu for use when creating custom measurement expressions. Use them to return data about the simulation. These functions take no arguments.

Function	Description	Arguments added
PulseRiseStartTime	Absolute time at which the rise portion of the current cycle starts.	<none>
PulseFallStartTime	Absolute time at which the fall portion of the current cycle starts.	<none>
PulseStopTime	Absolute time at which the fall portion of the current cycle ends, and the next cycle begins.	<none>
SimulationDuration	Absolute time at which the simulation ends.	<none>

Editing a Custom Measurement

Use one of the following methods to invoke the Measurement Expression Editor to modify an existing custom measurement expression.

- In the Measurement tab, click in the Description cell for the named custom measurement expression you want to edit. Then, click the icon at the right end of the Description cell.
– or –
- In the Measurement tab, right click in the Name or Description cell for the named custom measurement expression you want to edit. Then, select Edit from the pull-down menu.

The editor opens, displaying the selected expression in the Expression Treeview window. The Check and Clear Item buttons are active.

Editing the expression name and description

1. In the Expression Name field, enter a new name under which to save the modified expression.

When you save the modified expression, the unmodified original expression will still exist under the old expression name.

2. In the Description field, briefly describe the modified expression.

Changing a number value

1. Right-click on the number to change and select Number from the pull-down menu.
2. Enter the new number in the dialog box and click *OK*.

The number changes to the new value in the expression.

Changing other values

1. Right-click on an item to change (for example, an edge name or a cycle number) and select a new item value from the pull-down menu.

The item changes to the new value in the expression.

2. If the new item includes subitems, right-click on each new subitem and give it a value.

Clear and replacing an item

1. Select the item and then click *Clear Item*.

The selected item and all its sub items are deleted and replaced with the VALUE_TBD placeholder. Right-click on VALUE_TBD and select the new item from the pull-down menu.

VALUE_TBD is replaced with the selected item and any subitems it requires. Right-click on each new subitem and give it a value.

Checking the expression

- At any time during an editing session, you can click Check to verify the expression.

If the check fails, the first invalid item is selected and the applicable message appears in the Status Bar.

Saving the expression

- Click *OK* to save the expression.

The expression is checked before it is saved. If the check fails, a dialog box displays where you choose to continue editing or close the editor without saving your work. The first message is displayed in the Status Bar.

Note: Some operations, such as subtracting a value from an existing measurement are best accomplished by creating a new measurement that references the existing measurement as an argument of a difference function.

Creating a New Custom Measurement

Invoking the Measurement Expression Editor to create a new custom measurement

1. In the Measurement tab, right-click to display a pull-down menu.
2. In the pull-down menu, click *Add* to open the Measurement Expression Editor.

The editor opens empty other than entry for VALUE_TBD in the Expression Treeview window. The Check and Clear Item buttons are active.

Editing the expression name and description

1. In the Expression Name field, enter a name under which to save the new expression.
2. In the Description field, briefly describe the expression.

Constructing the expression

1. Right-click on VALUE_TBD and select the new item from the pull-down menu.

The VALUE_TBD placeholder is replaced with the selected item and any subitems it requires.

2. If the new item includes subitems, right-click on each new subitem and give it a value.

Checking the expression

- At any time during an editing session, you can click Check to verify the expression.

If the check fails, the first invalid item is selected and the applicable message displayed in the Status Bar.

Saving the expression

- Click *OK* to save the expression.

The expression is checked before it is saved. If the check fails, a dialog box displays where you choose to continue editing or close the editor without saving your work. The first message is displayed in the Status Bar.

Importing a Custom Measurements File

Use *File – Import Measurement* to import a custom measurement expressions file created for another topology file to the current topology.

Importing an existing Custom Measurement File

1. Choose *File – Import Measurement*.

The Import Measurements file browser opens set to display custom measurements (.dat) files.

2. Enter the file name in the File Name field or select the file from the list box.
3. Click Open.

To narrow the displayed list of files:

1. In the Files of type field:
 - a. Select Custom Measurements File (*.dat) to display all custom measurement expression files in the directory.
 - b. Click All Files (*.*) to display all files in the directory.

The list box changes to reflect your choice.

2. In the File name field:
 - a. Choose a file name from the list box (or enter it).
 - b. Enter a file name containing wildcard characters to narrow the list of files displayed in the list box.

The list box changes to reflect your choice.

Using the Change Directory checkbox

Use Change Directory to designate which directory is set as the working directory. Change Directory is deselected by default.

- Select Change Directory to designate as the working directory the directory where you are opening the file.
- Deselect Change Directory to keep the current working directory.

Searching for a file

1. Use the Look in Pulldown to navigate the directory structure.
 - ☐ On Unix, you can navigate the local directory structure.
 - ☐ On Windows, you can also navigate networked drives.
2. Select the directory where you want to open (or save) the file.
3. Select the file to open (or save).

The file name displays in the filename field
4. Click Open (or Save) to open the file.

Creating a Custom Measurements File

Use *File – Export Measurement* to create a custom measurements text file containing the custom measurement expressions created for this topology file.

Creating a Custom Measurement File

1. Choose *File – Export Measurement*.

The Export Measurements file browser opens set to display custom measurements (.dat) files.
2. Enter the file name in the File Name field.
3. Click Save.

Narrowing the displayed list of files

1. In the Files of type field:

a. Select Custom Measurements File (*.dat) to display all custom measurement expression files in the directory.

b. Select All Files (*.*) to display all files in the directory.

The list box changes to reflect your choice.

2. In the File name field:

a. Choose a file name from the list box (or enter it).

b. Enter a file name containing wildcard characters to narrow the list of files displayed in the list box.

The list box changes to reflect your choice.

Using the Change Directory checkbox

Use Change Directory to designate which directory is set as the working directory. Change Directory is deselected by default.

- Select Change Directory to designate as the working directory the directory where you are opening the file.
- Deselect Change Directory to keep the current working directory.

Searching for a file

1. Use the Look in Pulldown to navigate the directory structure.

- ☐ On Unix, you can navigate the local directory structure.
- ☐ On Windows, you can also navigate networked drives.

Select the directory where you want to open (or save) the file.

2. Select the file to open (or save).

The file name displays in the Filename field

3. Click Open (or Save) to open the file.

Setting the IOCell buffer model parameter

For single simulations, associate a single IOCell simulation model with the IOCell topology element. For simulation sweeping, associate multiple IOCell simulation models with the IOCell topology element.

1. Select the IOCell buffer model for the IOCell topology element:

- ☐ In the Topology Canvas, click to select the IOCell buffer model to modify.

-or-

- ☐ In the Parameters tab of the spreadsheet, click to select the spreadsheet Value cell containing the IOCell buffer model to modify.

2. The IOCell simulation model highlights and the icon displays in the Value cell.

3. Click the icon.

The Set Buffer Parameters dialog box appears for the type of parameter selected.

4. In the Model Name Pattern field, enter an expression including wildcards to narrow the list of models displayed in the Available Models list box.

The Available Models list box changes to display the available IOCell simulation models.

Associating a simulation model with the topology element

1. Click to select a model in the Available Models list box.

2. Click to move the model to the Selected Models list box.

When you select a model, the Library field changes to display the device model library where the IOCell simulation model is stored.

3. Click *OK*.

The IOCell simulation models appear in both the Topology Canvas and the Parameters tab of the spreadsheet.

Removing a simulation model from the topology element

1. Select a model in the Selected Models list box.

2. Click the arrow to move the model to the Available Models list box.

When you select a model, the Library field changes to display the device model library where the IOCell simulation model is stored.

3. Click *OK*.

The IOCell simulation models appear in both the Topology Canvas and the Parameters tab of the spreadsheet.

Reverting to the default simulation model

1. Click *Default Selection*.

The Selected Models list box changes to reflect the IOCell simulation model associated with the IOCell topology element by default.

2. Click *OK*.

The IOCell simulation models appear in both the Topology Canvas and the Parameters tab of the spreadsheet.

Set Parameters

[Dialog box](#) | [Procedures](#)

Dialog Box

Set Parameters Dialog Box

Single Value area

Use the Single Value area to enter a numeric value for the value.

Option	Description
Single Value	Click <i>Single Value</i> to enter the parameter as a simple, numerical value.
Value	Enter a number.

Linear Range area

Use the Linear Range area to enter the numeric parameter values as a range of numbers.

Option	Description
Linear Range	Click Linear Range to enter the parameter value as a range of numbers.
Start Value	Enter the minimum list value.
Stop Value	Enter the maximum list value.
Count	Displays the number of values that will be simulated.
Step Size	Enter a number specifying the increment between values in the range.

Allegro SI SigXplorer Reference

Context-Sensitive Menus

Multiple Values area

Use the Multiple Values area to enter the numeric parameter values as a list of numbers.

Option	Description
Multiple Values	Click Multiple Value to enter the parameter value as a range of numbers.
Multiple Values	Enter a new list value in this field.
Insert Value	Inserts the number in the Multiple Values field into the list of values.
Delete Value	Deletes a selected value from the list box.
TextEdit	Opens a text editor where you can construct a list of values.
List Box	Displays the list of values.

Expression area

Use the Expression area to enter the numeric parameter values as an expression.

Option	Description
Expression	Click Expression to enter the parameter value as an expression.
Expression	Enter a new expression in this field.
Variables list box	Displays available variables.
Append Var	Appends the variable selected in the Variables list box to the expression.
TextEdit	Opens a text editor where you can construct the expression.

Procedures

Modifying a Topology Element Reference Designator (RefDes)

The RefDes is displayed above the topology element symbol in the Topology Canvas.

1. Click the RefDes above the symbol in the Topology Canvas.

The Spreadsheet Parameters tab opens with the RefDes Name field highlighted and the associated Value cell ready for editing.

2. Enter the new text in the Name cell and click *Enter*.

The RefDes changes in the Topology Canvas as well as in the Spreadsheet.

Set Buffer Parameters

Dialog box

Used to change the IOCell simulation model associated with an IOCell topology element part model.

Set Buffer Parameter Dialog Box

Available Models area

Use the Available Models area to list available IOCell buffer models.

Option	Description
Model Name Pattern	Use wildcards to limit the display of IOCell models in the list box.
Available Models	Lists the IOCell models available for assignment to the IOCell topology element.

Selected Models area

Use the Selected Models area to list one or more IOCell buffer models associated with the topology element.

Option	Description
Default Selection	Use wildcards to limit the display of IOCell models in the list box.
Selected Models	Lists the IOCell models assigned to the IOCell topology element.

Allegro SI SigXplorer Reference

Context-Sensitive Menus

Other Options	Description
-> and <-	Moves models between the list boxes.
Library	Displays the device model library where an IOCell model is stored.

Modifying the Stimulus for an IOCell

[Dialog box](#) | [Procedures](#)

IOCell Stimulus Edit Dialog Box

- **Pulse** Holds the Data terminal high for one-half clock cycle, then low for one-half cycle, then repeats this pattern for the duration of the simulation. Holds the Enable terminal high for the duration of the simulation.
- **Rise** Transitions the Data terminal from low to high, then holds it high for the duration of the simulation. Holds the Enable terminal high for the duration of the simulation.
- **Fall** Transitions the Data terminal from high to low, then holds it low for the duration of the simulation. Holds the Enable terminal high for the duration of the simulation.
- **Custom** Overrides the predefined stimuli settings with a user-defined custom stimulus.
- **Quiet Hi** Holds the Data terminal in a quiescent (non-switching) high state. Holds the Enable terminal high for the duration of the simulation.
- **Quiet Lo** Holds the Data terminal in a quiescent (non-switching) low state. Holds the Enable terminal high for the duration of the simulation.
- **Tristate** Holds the Enable terminal low for the duration of the simulation.

Use the Terminal Info area to define Terminal Information for the IOCell.

From the Terminal Name pull-down menu, select the input terminal to which to apply the custom stimulus. The choices can include Clock, Data, and Enable.

- **Clock** – For Clocked IOCell MacroModels, the Clock Terminal Type activates the Clocked Stimulus Type and all the editing fields in the Stimulus Editor.
- **Data** – The Data Terminal Type activates the Periodic, Asynchronous, and Synchronous Stimulus Types and all the editing fields in the Stimulus Editor.
- **Enable** – The Enable Terminal Type activates the Periodic, Asynchronous, and Synchronous Stimulus Types and all the editing fields in the Stimulus Editor.

From the Stimulus Type pull-down, select a stimulus type. For Clocked IOCell MacroModels, select Clocked. For other signals, select from Periodic, Asynchronous, and Synchronous.

Allegro SI SigXplorer Reference

Context-Sensitive Menus

- **Periodic** – For IOCell Data and Enable terminals, the Periodic stimulus type repeats the bit pattern and frequency you supply in the Stimulus Editing Area for the duration of the simulation.
- **Asynchronous** – For IOCell Data and Enable terminals, the Asynchronous stimulus type toggles the terminal state for each time point you supply in the Stimulus Editing Area as a list of discrete transition times.
- **Synchronous** – For IOCell Data and Enable terminals, the Synchronous stimulus type clocks the bit pattern you supply at the rising edge, falling edge, or at both edges of the data-clock frequency. You supply all information in the Stimulus Editing Area.
- **Clocked** – For IOCells with a Clock terminal (Clocked IOCell MacroModels), the Clock stimulus type allows you to define parameters for a local reference clock, including the definition of a clock skew at each Clocked IOCell MacroModel.

From the Stimulus Name pull-down, select a saved, named stimulus, or enter a new stimulus name under which to save the custom stimulus you are defining.

Editing the Stimulus

In the Stimulus Editing area, define information specific to the Stimulus Type you specified in the Terminal Info area.

Stimulus Type	Information Displayed in the Stimulus Editing Area
Periodic	Frequency, Pattern, Jitter, Tr (0-100%) (read-only), and Tf (0-100%) (read-only).
Async	Init, Switch Times (ns), Jitter, Tr (0-100%) (read-only), and Tf (0-100%) (read-only).
Sync	Frequency, Init, Switch At, Pattern (including random), Tr (0-100%) (read-only), and Tf (0-100%) (read-only).
Clocked	Frequency, Init, Rise, Fall, %Duty, and Jitter.

For pre-defined Stimuli, the Stimulus Editing area displays the following read-only information. Frequency, Initial State, Rise, Fall, %Duty, and Jitter.