cadence™

Allegro® Constraint Manager Reference

Series L, XL, and GXL

Product Version 16.2
November 2008

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

Contents

File Menu Commands

File – Import – Constraints

Use this command to import the dictionary and constraints file (.dcf), archived on disk. The dictionary and constraints file is a snapshot of constraint information. It may include any user-defined properties, ECSets and their constraints, and net-related objects and their constraints (including CSet references). The dictionary and constraints file is proprietary to Cadence Design Systems, Inc., and, as such, is not available for editing.

Important

This command is not implemented when launching Constraint Manager from Allegro® Design Entry HDL or Allegro® System Architect. We plan to implement this in a future release.

Note: If you are using Constraint Manager, when launched from a Series L PCB editor, and you import constraints from a board designed with a Series XL PCB editor, choosing *Overwrite all constraints, Merge constraints,* or *Replace constraints* affects constraints in the *Routing* workbook only; constraints that may exist in the *Signal Integrity* and *Timing* workbooks remain unchanged.

You can choose from the following options when importing constraints. All options generate a report.

Merge constraints

Preserves data in the design and reads in new data from the .dcf file. With *Merge* enabled, Constraint Manager adds new constraints, new objects (including ECSets), and new constraints on objects imported from the .dcf file to the design, as well as cross-section information. Objects and constraints in the design that are not in the .dcf file remain in the design, unchanged. If constraint values differ between the design and the .dcf file, the constraint value in the .dcf file prevails over the constraint value in the design.

If an object in the design is constrained, but the object in the .dcf file is not constrained, Constraint Manager preserves that object and its constraints in the design. For example, if a differential pair in the design has a relative propagation delay and the differential pair in the .dcf file is not constrained, on import the differential pair in the design retains its relative propagation delay setting.

Replace Constraints

Overwrites .only those objects in the design that are constrained in the `.dcf` file. With *Replace Constraints* enabled, Constraint Manager adds new constraints, new objects (including ECSets), and new constraints on objects imported from the `.dcf` file to the design. Objects and constraints in the design that are not in the `.dcf` file remain in the design, unchanged. If constraint values differ between the design and the `.dcf` file, the constraint value in the `.dcf` file prevails over the constraint value in the design.

If an object in the design is constrained but the object in `.dcf` file is not, Constraint Manger removes the constraints from the design. For example, if a differential pair in the design has a relative propagation delay setting and the differential pair in the `.dcf` file is not constrained, on import the differential pair in the design loses its relative propagation delay setting.

Overwrite all constraints

Clears data (ECSets, objects, constraints on objects) in the design and reads in new data from the `.dcf` file. With *Overwrite All constraints* enabled, Constraint Manager adds new constraints, new objects (including ECSets), and new constraints on objects imported from the `.dcf` file to the design. Constraint Manager clears objects in the design that are not in the `.dcf` file.

Run DRC and update Shapes

When enabled, Constraint Manager first imports constraints, and then runs design rule checks and updates dynamic shapes; when disabled, no design rule checks or shape updates occur on constraint import.

Report only

Generates a report of the constraint import without executing the import.

File – Import – Technology File

Use this command to import either a legacy tech file (`.tech`) or a new Technology Constraint File `.tcf` file into your design.

For additional information about tech files, see the *Using Technology Files* chapter in the *Defining and Developing Libraries User Guide* in your product documentation.

Dialog Boxes

Import a technology file (.tcf) Dialog Box

File name	Specifies the name of the tech file, either the XML `.tcf` file or the legacy `.tech` file. Browse to find the tech file or click *Library* to display the Technology File Library Dialog Box.
Files of Type	Select either the new `.tcf` file or the legacy `.tech` file.
Import Modes	You can import the tech file in one of three modes: *Merge*, *Replace*, or *Overwrite*. *Overwrite* mode is enabled by default.
Merge constraints	Click this button to enable *Merge* mode. Using this mode, the tool adds or changes objects and constraints in the design, but does not delete objects or its constraints in the design. For example, if the design has a physical constraint set (CSet) defined but the tech file does not define the physical CSet, the tool does not remove the CSet and its constraints from the design.
Replace constraints	Click this button to enable *Replace* mode. Using this mode, the tool substitutes objects and their constraints in the design with existing objects of the same name from the tech file. If no object of the same type with the same name exists in the design, then the tool ignores the object in the tech file.

Overwrite all constraints	Click this button to enable *Overwrite* mode. Using this mode, the tool updates the design to match the contents of the tech file. For example, if the tech file contains only physical and spacing information, then the tools processes only those object types. If the design has a physical CSet defined but the tech file does not define it, the tool deletes the physical CSet and its constraints from the design. The tool does not delete CSets and net classes if they are referenced by any design objects. The tool always deletes Net Class-Class, Region, Region-Class, and Region-Class-Class objects from the design if the object does not exist in the tech file as these objects do not reference design objects, for example, nets.
Report only	Check this box to produce a text description of the import results without performing the import task. The report is similar to the report generated by the *File – Import – Constraints* command.
Run DRC and update Shapes	When enabled, Constraint Manager first imports constraints, and then runs design rule checks and updates dynamic shapes; when disabled, no design rule checks or shape updates occur on constraint import.
Library	Click this box to display the Technology File Library Dialog Box and choose a file from those that exist in the directories defined by the TECHPATH environment variable. TECHPATH is an existing environment variable (enved command) which is supported by Constraint Manager.
Help	Displays help for this command.

Technology File Library Dialog Box

Filter filenames	Controls which technology files appear in the list. Only those technology files whose names match the characters in this field are listed.
OK	Click *OK* or double-click on the file to select it and return to the Import a technology file (.tcf) Dialog Box.

Cancel	Cancels the operation and returns you to the Technology File Library Dialog Box.
Help	Displays help.

Procedure

1. Open a design into which you want to import design data.
2. Choose *File – Import – Technology File* from the menu bar.

   The Import a technology file (`.tcf`) dialog box appears.
3. Enter the file name for a `.tech` or `.tcf` file. Either browse to find the file

   -or-

   click *Library* to find a tech file that exists in the directories defined by the TECHPATH environment variable. Double-click on the file or click *OK* in the Technology File Library dialog box.
4. Click the type of *Import mode* in the Import a technology file (`.tcf`) dialog box.
5. Click *OK* to import the file and dismiss the dialog box.

File – Import – Electrical CSets

Procedures

Use this command to import a selected on-disk topology template into Constraint Manager. The imported template becomes an Electrical CSet which can be referenced by net-related objects that share the same electrical characteristics (see Mapping Templates and ECSets to Net-related Objects in the *Constraint Manager User Guide.*

Tip

An Electrical CSet is a named collection of electrical constraints and their default values.

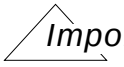
When Constraint Manager is launched from Design Entry HDL:

- Topological constraints are not applied. The reference is maintained, however. Topological mapping occurs when the constraint information is fed-forward to the design.
- The *Referenced Electrical CSet* column is colored yellow, indicating that you must run the *Audit – Electrical CSets* command.

See also: *Objects – Constraint Set References*.

Procedures

Importing a topology file



Important

By default, custom measurements are not included with an imported Electrical CSet. To override this behavior, you must enable the *Update existing or create new Custom Measurement worksheet* option.

1. Choose *File – Import – Electrical CSet*.
2. Select one or more topology files from the browser list.
3. Click *Open*.

   Constraint Manager imports the topology template as an ECset. The Electrical CSet is located in the *Objects* column of the *Electrical Constraint Set* folder.

Tip

If the *Automatic Topology Update* checkbox is enabled (Tools – Options), the refreshed template information is immediately applied to the net-related objects that reference an Electrical CSet; otherwise, you must choose Tools – Update Topology to apply the changes.

Assigning the imported Electrical CSet to a net-level object

1. Select an object in one of the worksheets in the *Net* folder.
2. Do one of the following:
   - Choose *Objects – Electrical CSet References*.

     - or -

   - Right-click and choose *Electrical CSet References* from the pop-up menu.

   The Electrical CSet References dialog box appears.
3. From the drop-down menu, choose the desired Electrical CSet.
4. Click *OK* to apply the assignment and close the dialog box.

   The object inherits the constraint values of the selected Electrical CSet.

File – Import – Analysis Results

Use this command to read a results file that contains previously saved analysis information.

A Design Entry HDL schematic contains logic, but not physical information about the design. You can use this command to import a snapshot of the design and verify analysis results with constraints.

Procedure

1. Choose *File – Import – Analysis Results*.
2. Select an analysis results (`.acf`) file.
3. Click *Open*.

   Constraint Manager reads the analysis results file and updates all constraints and margins.

File – Import – Worksheet File

Use this command to read a tab-delimited (`.txt`), comma-delimited (`.csv`), or space-delimited (`.psn`) ASCII text file into the active worksheet.

Note: The first *token* of every line must contain the object type and name (`type:name`). Header information is not supported.

Important

If a constraint object (such as *Match Group* or *Pin Pair*) exists in an exported worksheet, they are not automatically created in the imported worksheet. On import, Constraint Manager updates the values only when the constraint object exists in both the exported and the imported worksheets.

Procedure

1. Choose *File – Import – Worksheet File*.

   The Import Worksheet File dialog box appears.

2. Click the *files of type* drop-down menu to filter on a file type.
3. Select a file.
4. Click *Open*.

   Constraint Manager populates the active worksheet with imported values.

File – Import – Worksheet Customization

Use this command to import a worksheet customization file from another design.

The worksheet customization file (.wcf) contains columns that you will add to predefined (default) worksheets and new, custom worksheets.

Procedure

1. Choose *File – Import – Worksheet Customization*.

   The *Import Worksheet Customization* dialog box appears.

2. Navigate to the directory where the worksheet customization file resides.
3. Select the customization file.
4. Click *Open*.

   Constraint Manager imports the worksheet customization file adding workbooks, worksheets, columns, and labels where appropriate

Tip

You can tailor your worksheets to suit your corporate requirements by using the CDS_SITE environment variable.

If you export or copy your customization file (.wcf) to the folder pointed to by the CDS_SITE environment variable, all users in the corporation who have the environment variable set to the same folder automatically get the customization file loaded for all their designs.

Example:

```
CDS_SITE = c:\my_cust
```

Create the following directories . . .

```
c:\my_cust\cdssetup\consmgr
c:\my_cust\cdssetup\consmgr\consmgr.wcf
```

File – Export – Text File

Use this command to export the contents of the active worksheet to an ASCII text file.

Procedure

1. Choose *File – Export – Text File*.

   The Export Text File dialog box appears.

2. Select a file.

3. Click *Save*.

   Constraint Manager exports the contents of the active worksheet to a text file.

File – Export – Worksheet File

Use this command to export a tab-delimited (`.txt`), comma-delimited (`.csv`), or space-delimited (`.psn`) ASCII text file based on the active worksheet.

Important

If a constraint object (such as *Match Group* or *Pin Pair*) exists in an exported worksheet, they are not automatically created in the destination worksheet. On export, Constraint Manager updates the values only when the constraint object exists in both the exported and the imported worksheets.

Procedure

1. Choose *File – Export – Worksheet File*.

   The *Export Worksheet File* dialog box appears.

2. Click the *save as type* drop-down menu to filter on a file type.

3. If you want to export all data in all rows, check the *Expand all rows* checkbox; otherwise, Constraint Manager exports only the data that is visible (not collapsed or hidden).

4. Click *Save*.

   Constraint Manager exports and saves the contents of the active worksheet.

File – Export – Constraints

Use this command to export a dictionary and constraints (`.dcf`). The dictionary and constraints file is a snapshot of electrical constraint information. It may include any user-defined properties, ECSets and their constraints, and net-related objects and their constraints (including Electrical CSet references). The dictionary and constraints file is proprietary to Cadence Design Systems, Inc., and, as such, is not available for editing.

Important

This command is not implemented when launching Constraint Manager from Allegro® Design Entry HDL or Allegro® System Architect. We plan to implement this in a future release.

Procedure

1. Choose *File – Export – Constraints*.

   The Export Constraints dialog box appears.

2. Click *Save* to archive the current session or choose to overwrite an existing dictionary and constraints file.

   Constraint Manager exports the constraint information for the current session, overwriting the contents of the existing dictionary and constraints file.

File – Export – Technology File

Use this command to create a tech file (`.tcf`). For additional information on tech files, see the *Using Technology Files* chapter in the *Defining and Developing Libraries User Guide* in your product documentation.

The file can include the current design's cross-section, constraints, user property definitions, and design units. You can use the *Contents* option of this command to control the contents of the file. This `.tcf` file supports all the information which is currently available in the `.tech` file.

Example

If you create a tech file using only the physical and spacing constraints, the following information appears in the file:

- Physical and spacing constraint sets and their constraints
- Physical and spacing net classes with constraint set (CSet) references as well as physical and spacing constraint overrides

Export a technology file (.tcf) Dialog Box

File name	Specifies the name of the tech file that you are creating.
Contents	
User-defined constraint definitions	Check this box to export all user-defined property definitions to the tech file.
Electrical constraints	Check this box to export all electrical constraint sets to the tech file.
Physical and spacing constraints	Check this box to export all physical and spacing constraint-related objects (constraint sets, Net Classes, Class-Class, Regions, Region-Class, and Region-Class-Class) to the tech file. If you check this box, the *Cross-sectio*n box is automatically checked.
Cross-section	Check this box to export the design’s cross-section to the tech file. The cross-section includes all the conductor, surface, dielectric, and bonding _wire layers and their characteristics.

OK	Click this button to create the tech file using the parameters that you applied.
Cancel	Click this button to cancel the command.
Help	Displays help for this command.

Procedure

1. Open a design from which you are exporting design data.
2. Choose *File – Export – Technology File* from the menu bar.

   The Export a technology file (`.tcf`) dialog box appears.
3. Enter a file name for the tech file.
4. Complete the parameters that define the contents of the tech file.
5. Click *OK* to start the tech file creation process and dismiss the dialog box.

   If the technology file already exists on disk, a dialog box asks you if you want to overwrite it.

File – Export – Electrical CSets

Use this command to save a topology template (`.top`) file for an existing Electrical CSet. You can import the topology template file into another design or into SigXplorer for further exploration.

Procedures

You can export individual, or multiple, ECSets.

Exporting a single Electrical CSet

1. Choose *File – Export – Electrical CSets*.

   The Export an Electrical CSet File dialog box appears.

2. From the Electrical CSet drop-down menu, select an Electrical CSet.

   Constraint Manager adds the name of the Electrical CSet that you selected to the *File Name* field.

3. Accept the Electrical CSet name that you selected, or specify a different name.

4. If desired, specify a revision number.

5. Navigate to the desired directory.

6. Click *Save*.

   Constraint Manager saves the Electrical CSet as a topology file. Later, if you import the topology (see *File – Import – Electrical CSets* on page 12), Constraint Manager compares the revision number when you update the topology (see *Tools – Update Topology* on page 192).

Exporting all ECSets

1. Choose *File – Export – Electrical CSets*.

   The Export an Electrical CSet File dialog box appears.

2. If desired, specify a revision number.

3. Click *Export All*.

4. Navigate to the desired directory.

5. Click *Save*.

File – Export – Analysis Results

Use this command to export the results of the most recent analysis session in Constraint Manager. You may want to share the results with a colleague who is working on an identical design. You may also want to send results to the schematic where it can be imported.

Procedure

1. Choose *File – Export – Analysis Results*.

   The Export Actuals dialog box appears.

2. Navigate to the desired directory.
3. Click *Save*.
4. Constraint Manager saves the results to a results file (`.acf`).

File – Export – Worksheet Customization

Use this command to export a worksheet customization file to disk for use in another design.

The worksheet customization file (`.wcf`) contains columns that you added to predefined (default) worksheets and new, custom worksheets.

Procedure

1. Choose *File – Export – Worksheet Customization*.

   The *Export Worksheet Customization* dialog box appears.

2. Navigate to the directory where you want to save the worksheet customization file.
3. Click *Save*.

Constraint Manager exports the worksheet customization file adding workbooks, worksheets, columns, and labels where appropriate.

Tip

You can tailor your worksheets to suit your corporate requirements by using the `CDS_SITE` environment variable.

File – File Viewer

Use this command to display log (`*.log`), report (`*.rpt`), text (`*.txt`) or data (`*.dat`) files from your current working directory.

Procedure

1. Choose *File – File Viewer.*
2. Refine your selection using the *Files of Type* and *File Name* fields.
3. Click *Open*.

   The report appears.

File – Print

Use this command to print the active worksheet.

File – Print Preview

Use this command to preview how a worksheet will appear when printed. You can zoom, display the previous or next, or display a single- or adjacent- worksheets.

File – Print Setup

Use this command to specify a printer, paper source, and page orientation for worksheets printed with the *File – Print* command.

File – Page Setup

Use this command to specify margins, page order, grids, color, and page orientation.

File – Close

Use this command to close Constraint Manager. Constraint assignments and modifications are saved in the PCB-, package-, or schematic-database.

Tip

File – Close is available when you launch Constraint Manager from a PCB Editor or APD. The *File – Exit* command is available when you launch Constraint Manager in stand-alone mode or from Design Entry HDL.

File – Exit

Use this command to close Constraint Manager. Constraint Manager saves any pending constraint modifications to the dictionary and constraints file (see *File – Export – Constraints*)

Tip

File – Close is available when you launch Constraint Manager from a PCB editor or APD. *File – Exit* is available when you launch Constraint Manager in stand-alone mode or from Design Entry HDL.

Edit Menu Commands

Edit – Undo

Use the multi-level Undo command to step back through a history of recent operations.

Edit – Redo

Use the Redo command to reapply the most-recent Undo(ne) operation.

Edit – Cut

Use this command to cut the value from a cell. The value remains in a paste buffer until a subsequent cut or copy command is issued.

Edit – Copy

Use this command to copy (and paste) the value from one cell to other cells.

Edit – Paste

Use this command to paste the value, from a cell that was copied or cut, into the selected cell.

Edit – Paste Special

Use this command in conjunction with a cell that contains a formula. Paste Special lets you paste the formula itself, or the formula's value, into another cell.

Edit – Find

Dialog Box | Procedures

Use this command to locate an object in a worksheet, a column in the *Worksheet Selector*, or an attribute value in a cell.

Tip

The toolbar has a type-in field where you can specify a search string to quickly locate an object in the active worksheet. This field keys off of the last data type selected in the pull-down menu of the *Find* field.

Find and Replace Dialog Box

Use this field . . .	**To . . .**
Find (pull-down menu)	Choose one of the following options from the pull-down menu: ■ *Object Name* To locate a row based on an object's name. ■ *Column Name* To locate a column (or the internal property name) in the *Worksheet Selector*. ■ *Attribute value* To locate a value in a cell of the active worksheet and replace the value with the string entered in the *Replace attribute value* field.
Text	Enter a search string based on the selection made in the *Find* field. This field accepts regular expressions.
Exact match only (checkbox)	Find a search string consisting of an exact match.
Expand hierarchy (checkbox)	Expand the worksheet hierarchy to show the worksheet containing the value specified in the *Text* field.

Use this field . . .	To . . .
Replace attribute value	Replace the string entered in the *Text* field. If this field is blank, *Replace* behaves the same as *Find Next*. **Note:** Active only when you choose *Attribute* from the *Find* pull-down menu. Check *Replace inherited values* to set overrides on inherited values that match the *Text* field. When unchecked, *Replace* ignores inherited values.

Procedures

Locating an object in the Objects column

1. Choose *Edit – Find*.

   The *Find and Replace* dialog box appears.

2. In the *Find* field, choose *Object name* from the pull-down menu.
3. In the *Text* field, enter a string.
4. If you want to search for an exact match of an object, choose *Exact match only*.
5. If the object is a child of a parent object (such as an Xnet in a bus), choose *Expand Hierarchy*.
6. Click *Find Next*.

   Constraint Manager highlights the object.

   Note: Press F3 to quickly find subsequent matches.

7. Click *Close* to dismiss the dialog box.

Locating a column in the Worksheet Selector

1. Choose *Edit – Find*.

   The *Find and Replace* dialog box appears.

2. In the *Find* field, choose *Column name* from the pull-down menu.
3. In the *Text* field, enter a string.

4. If you want to search for an exact match of a column or attribute name, choose *Exact match only*.
5. Click *Find Next*.

   Constraint Manager enters *Customize* mode and highlights a matching column in the *Worksheet Selector*.

   Note: Press F3 to quickly find subsequent matches.
6. Click *Close* to dismiss the dialog box.

Replacing an attribute in the active worksheet

1. Choose *Edit – Find*.

   The *Find and Replace* dialog box appears.
2. In the *Find* field, choose *Attribute value* from the pull-down menu.
3. In the *Text* field, enter a string.
4. If you want to search for an exact match of an entire cell, choose *Exact match only*.
5. If the object is a child of a parent object (such as an Xnet in a bus), choose *Expand Hierarchy*.
6. In the *Replace attribute value* field, enter a replacement string.
7. Click *Find Next*.

   Constraint Manager highlights the cell.

   Note: Press F3 to quickly find subsequent matches.
8. Choose *Replace* (or *Replace All*).
9. Click *Close* to dismiss the dialog box.

Edit – Find Next

Use this command to locate the next occurrence of an object in a worksheet, a column in the *Worksheet Selector*, or an attribute value in a cell. You can also access this command from the *context* menu (right-click) or by pressing F3.

This command works in conjunction with the Edit – Find command.

Edit – Find Previous

Use this command to locate the previous occurrence of an object in a worksheet, a column in the *Worksheet Selector*, or an attribute value in a cell. You can also access this command from the *context* menu (right-click) or by pressing `SHIFT+F3`.

This command works in conjunction with the Edit – Find command.

Edit – Toggle Bookmark

Use this command to bookmark any design element that you select in the *Objects* column.

Procedure

1. Click on an element in the *Objects* column.

   Do one of the following:

2. Choose *Edit – Toggle Bookmark*.

   - or -

3. Right-click and choose *Bookmark – Object Bookmark* from the pop-up menu.

   A square appears to the left of the object to aid you in locating the object. The bookmark follows the object across worksheets.

This command works in conjunction with the Edit – Next Bookmark and Edit – Previous Bookmark commands.

Edit – Next Bookmark

Use this command to locate the next bookmarked element in the *Objects* column.

This command works in conjunction with the Edit – Toggle Bookmark command.

Procedure

1. Click on a bookmarked element in the *Objects* column.

   Do one of the following:

2. Choose *Edit – Next Bookmark*.

   - or -

3. Right-click and choose *Next Bookmark* from the pop-up menu.

Edit – Previous Bookmark

Use this command to locate the previous bookmarked element in the *Objects* column.

This command works in conjunction with the Edit – Toggle Bookmark command.

Procedure

1. Click on a bookmarked element in the *Objects* column.

   Do one of the following:

2. Choose *Edit – Previous Bookmark*.

   - or -

3. Right-click and choose *Previous Bookmark* from the pop-up menu.

Edit – Go to Source

Use this command to locate the parent object that owns the inherited Electrical CSet of the selected child object.

Go to Source is available only if the cell value is not blue (not set in that cell). Using the command takes you to the source from which the current cell inherited its value. The Source itself should be blue (or set on the object). Inheritance can come from an Electrical CSet, Match Group, Differential Pair, Bus, Net or Xnet. The typical color for an inherited value is black, but the value will be green, red, or yellow if you are in a margin cell. In Margin cells the *Go to Source* command takes you to the worst-case margin. If you hover over a cell with an inherited value, the status bar will indicate the source of the value. Overrides are blue, as you set a value in the cell, and; therefore, it is now no longer a source.

Procedures

Finding the parent object

1. Click inside the cell of the *Referenced Electrical CSet* column, adjacent to the row containing the child object. You may have to expand the parent object to access the child object.
2. Click *Cancel* to dismiss the *Electrical CSet References* dialog box.
3. Do one of the following:

   - Choose *Edit — Go to source*.

     - or -

   - Right-click and choose *Go to source* from the pop-up menu.

   Constraint Manager highlights the parent object that contains the assigned Electrical CSet (inherited by the child object).

Finding the worst-case margin

1. Click on the object at any level in the object hierarchy.
2. Do one of the following:

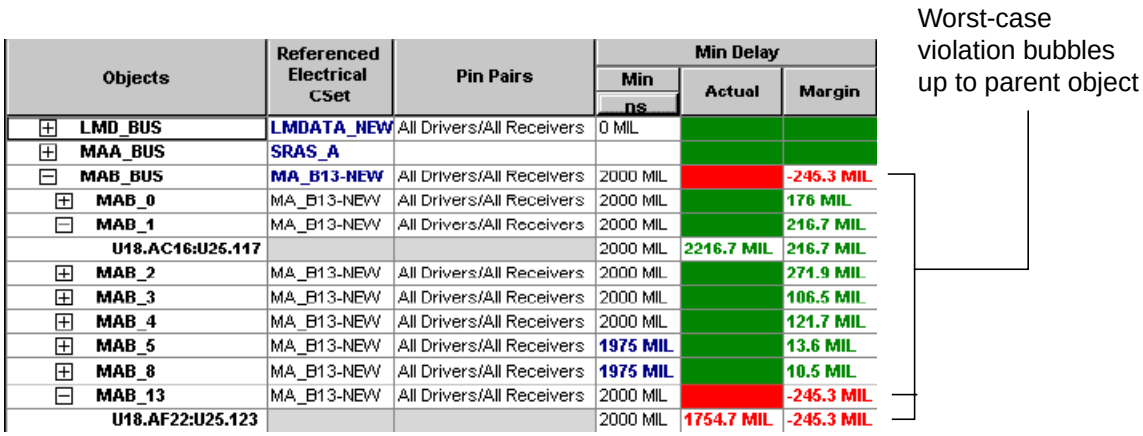
   - Choose *Edit — Go to source*.

     - or -

   - Right-click and choose *Go to source* from the pop-up menu.

   Constraint Manager expands the object, as necessary, and highlights the child object that contains the worst-case margin.

   In the following illustration, the bus object (MAB_BUS) expands to show the worst-case violation. In this case, the pin pair on net MAB_13.



Objects	Referenced Electrical CSet	Pin Pairs	Min Delay		
			Min (ns)	Actual	Margin
⊞ LMD_BUS	LMDATA_NEW	All Drivers/All Receivers	0 MIL		
⊞ MAA_BUS	SRAS_A				
⊟ MAB_BUS	MA_B13-NEW	All Drivers/All Receivers	2000 MIL		-245.3 MIL
⊞ MAB_0	MA_B13-NEW	All Drivers/All Receivers	2000 MIL		176 MIL
⊟ MAB_1	MA_B13-NEW	All Drivers/All Receivers	2000 MIL		216.7 MIL
U18.AC16:U25.117			2000 MIL	2216.7 MIL	216.7 MIL
⊞ MAB_2	MA_B13-NEW	All Drivers/All Receivers	2000 MIL		271.9 MIL
⊞ MAB_3	MA_B13-NEW	All Drivers/All Receivers	2000 MIL		106.5 MIL
⊞ MAB_4	MA_B13-NEW	All Drivers/All Receivers	2000 MIL		121.7 MIL
⊞ MAB_5	MA_B13-NEW	All Drivers/All Receivers	1975 MIL		13.6 MIL
⊞ MAB_8	MA_B13-NEW	All Drivers/All Receivers	1975 MIL		10.5 MIL
⊟ MAB_13	MA_B13-NEW	All Drivers/All Receivers	2000 MIL		-245.3 MIL
U18.AF22:U25.123			2000 MIL	1754.7 MIL	-245.3 MIL

Edit – Change

The function of the *Change* command differs by domain.

Electrical Domain

Use this command to enter values into a cell. You can enter values directly into a cell or you can use the *Edit – Change* command. This command presents a dialog box applicable to the cell type. Novice users may find it easier to enter cell values through a dialog box rather than typing directly in a cell as the dialog box fields guide you through all the parameters for the cell type.

Physical, Spacing, and Same Net Spacing Domains

You can use *Change* in a Constraint Set folder to specify layer variances on a CSet. These variances are inherited by the object that is assigned that CSet in the Net folder. You cannot set layer variances on a CSet in the Net folder; however, you can change the default value of that CSet, but that change affects all layers.

Figure 2-1 CSet variances by layer

Constraint Set Folder View

Type	Objects	Line Width	
		Min	Max
		mil	mil
Dsn	⊟ tom_k_movie_regio	5.00	0.00
PCS	⊞ DEFAULT	5.00	0.00
PCS	⊟ 4MIL	4.00:4.00:4.0...	0.00
Lyr	TOP	4.00	0.00
Lyr	GND	4.00	0.00
Lyr	SIG1	4.00	0.00
Lyr	SIG2	4.00	0.00
Lyr	VCC	4.00	0.00
Lyr	VCC1	4.00	0.00
Lyr	SIG3	3.00	0.00
Lyr	SIG4	2.00	0.00
Lyr	GND1	4.00	0.00
Lyr	BOTTOM	4.00	0.00

Net Folder View

Type	Objects	Line Width
		Min
		mil
Net	⊟ L11	
Lyr	TOP	4.00
Lyr	GND	4.00
Lyr	SIG1	4.00
Lyr	SIG2	4.00
Lyr	VCC	4.00
Lyr	VCC1	4.00
Lyr	SIG3	3.00
Lyr	SIG4	2.00
Lyr	GND1	4.00
Lyr	BOTTOM	4.00

OK Cancel Help

Type	Objects	Line Width
		Min
		mil
Net	⊟ L11	8.00
Lyr	TOP	8.00
Lyr	GND	8.00
Lyr	SIG1	8.00
Lyr	SIG2	8.00
Lyr	VCC	8.00
Lyr	VCC1	8.00
Lyr	SIG3	8.00
Lyr	SIG4	8.00
Lyr	GND1	8.00
Lyr	BOTTOM	8.00

OK Cancel Help

Procedure

1. Click in a cell.
2. Do one of the following:
   - Choose *Edit – Change*.

     - or -

   - Right-click and choose *Change* from the pop-up menu.

   A dialog box specific to the cell type appears.
3. Enter values in each field.
4. Click *OK*.

Edit – Clear

Use this command to clear the contents of the selected cells.

Procedure

1. Click in a cell that contains a value.
2. Do one of the following:
   - ❑ Choose *Edit – Clear*.

     - or -
   - ❑ Right click and choose *Clear* from the pop-up menu.

     - or -
   - ❑ Press `Backspace`.

     - or -
   - ❑ Press `Spacebar`.

     - or -
   - ❑ Press `Delete`.

   Constraint Manager clears the value from the cell and any dependent cells that may reference the cell.

Edit – Formula

Use this command to customize any property or constraint by adding a formula to it (you do not have to be in worksheet customization mode to use formulas).

Note: User-defined predicates and measurements are closely related to formulas, but they do not have menu choices; therefore, much of the material in this section applies to them as well.

Tip

This section complements the material in Customizing Design Rule Checks in the *Constraint Manager User Guide*.

A formula executes once you add it to a cell. Subsequently, you must force a recalculation (see Edit – Calculate on page 66 or Edit – Calculate All on page 67). Constraint Manager performs the calculation and returns the calculated value to the cell that contains the formula.

Formulas are portable among designs. See

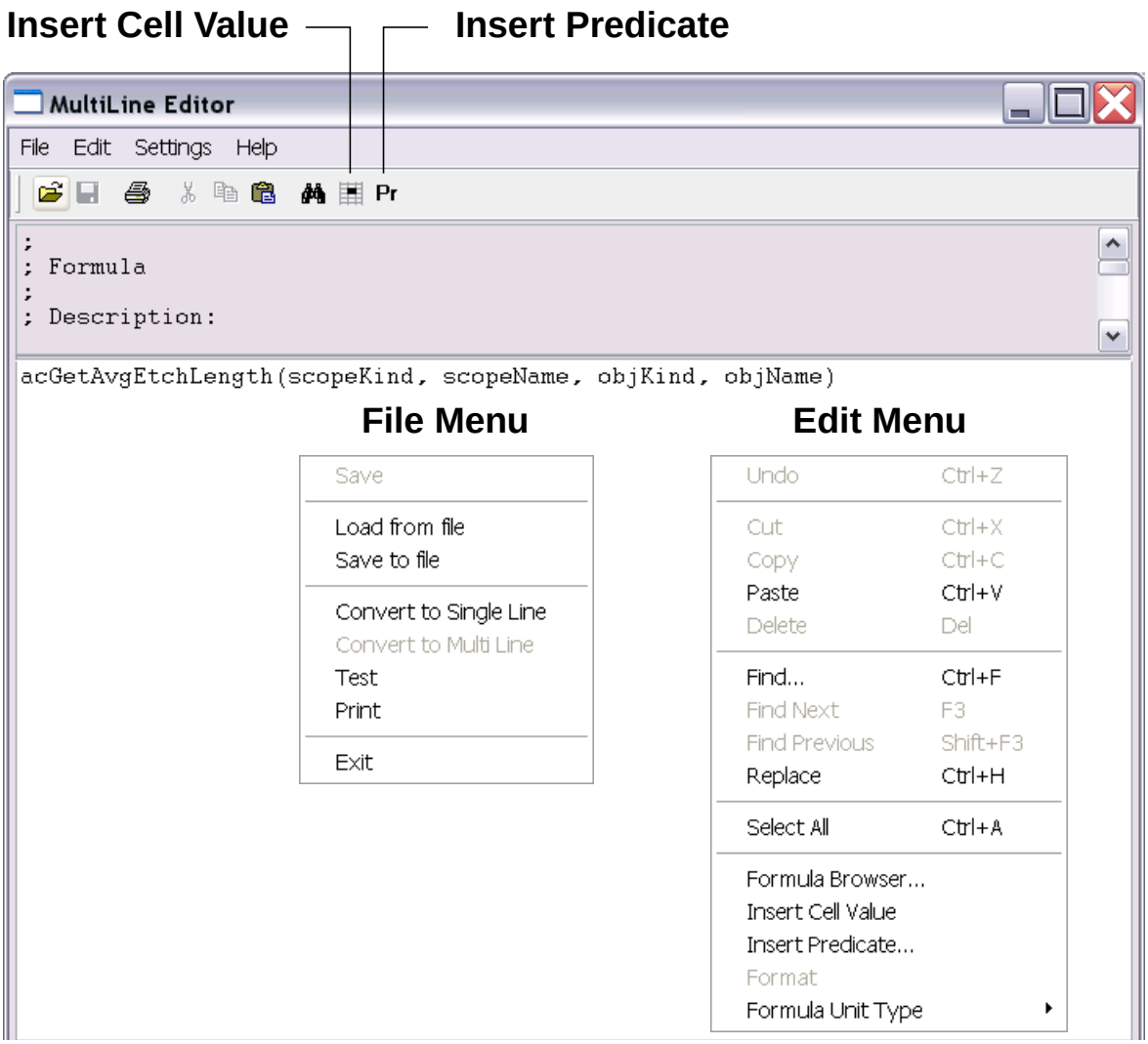
- File – Import – Constraints on page 7
- File – Import – Worksheet Customization on page 16
- File – Export – Constraints on page 19
- File – Export – Worksheet Customization on page 25

You build a formula by selecting cells and inserting predicates. The calculation is captured using the Cadence SKILL® language. Constraint Manager automatically wraps the SKILL code to ensure that it can be executed.

Working with the Multi-line Editor

You use the multi-line Editor to write, debug, and test user-defined formulas, predicates, and measurements.

Figure 2-2 The Multi-line Editor



Multi-line Editor Commands

Use this command . . .	To . . .
Load from File	Loads a user-defined formula, predicate, or measurement that was previously saved to disk.
Save to file	Saves the user-defined formula, predicate, or measurement to a text file for archiving and use with another design
Convert to Single Line	Converts the formula to a single line and switches from Multi-line to Single-line mode.

Use this command . . .	To . . .
Test	Use the *Test* button in the *Single*- or *Multi-line Editor* to check for syntax errors in your user-defined formula, predicate, or measurement. This launches a viewable log file with the following fields: ■ **Code** Displays the user-defined formula-, predicate, or measurement-code for reference. ■ **Debug** Displays the output produced by the special debug predicates that can be used in testing. These predicates are only active during the test command and can be used to display intermediate information to aid in debugging. See Pre-defined Predicates on page 58 for more detailed descriptions of the debug predicates. ■ **Result** Displays the result of the user-defined formula-predicate- or measurement-calculation. For formulas, this is the value that is inserted into the cell if the formula is accepted. ■ **SKILL Lint** Runs a default SKILL Lint check on a user-defined formula, predicate, or measurement. This can spot errors in the SKILL program and give suggestions on how to debug your code.
Formula Browser	Lets you access formulas available within the design (available from both the *Single*- and *Multi-line Editor*).
Insert Cell Value	Inserts the code required to access a cell's value when editing a formula.
Insert Predicate	Inserts a chosen predicate into the formula (see Pre-defined Predicates on page 58 for more detailed descriptions).

Use this command . . .	To . . .
Formula Unit Type	Select a unit type from the pull-right menu. Contents of this menu vary depending on the cell from which you added the formula.

A formula is a SKILL function that automatically receives parameters from Constraint Manager when the formula is calculated.

Formula Parameters

This table contains a list of parameters for formulas.

See Customizing Design Rule Checks in the *Constraint Manager User Guide* for information on using formula parameters.

Table 2-1 Formula Parameters

This parameter . . .	Retrieves the . . .
■ `parentScopeKind` `parentScopeName`	Object `kind` and `name` of the Design or System containing the parent object
■ `parentKind` `parentName`	Object `kind` and `name` of the parent. The parent is populated and used when the object/formula being calculated is a member of a match group (the parent will be its Match Group).
■ `scopeKind` `scopeName`	Object `kind` and `name` of the Design or System containing the object.
■ `objKind` `objName`	Object `kind` and `name` of the current object, which is updated upon formula calculation
■ `attrName`	Name of the attribute for the current cell, which is updated upon formula calculation

These parameters are available for use within the formula. They can be used in any predicate calls that require the same information.

Pre-defined Predicates

Predicates are functions that return data (usually a single value). They are used with formulas and user-defined measurements. You can define your own predicates and you can choose from Constraint Manager's pre-defined predicates.

See Customizing Design Rule Checks in the *Constraint Manager User Guide* for information on using predicates.

Table 2-2 Pre-defined Predicates

Predicate	Scope	Function
`acGetDBID`	All named objects	Returns the database id of an object. Required for accessing most AXL functions, which are available to measurements, predicates, and formulas. Not available as a measurement.
`acGetLength`	Xnets, Nets, and Pin Pairs	Returns the total length for an object. Uses both etch and manhattan length of connections which are not routed.
`acGetEtchLength`	Xnets, Nets, and Pin Pairs	Returns the total etch length for an object
`acGetManhattanLength`	Xnets, Nets, and Pin Pairs	Returns the Manhattan length of an object
`acGetPropDelay`	Pin Pairs	Returns the propagation delay of a Pin Pair. Uses both etch and manhattan delay of connections that are not routed
`acGetEtchPropDelay`	Pin Pairs	Returns the etch-only propagation delay of a Pin Pair

Predicate	Scope	Function
acGetManhattanPropDelay	Pin Pairs	Returns the Propagation delay of a Pin Pair's manhattan ratsnest connections
acGetPercentManhattan	Xnets, Nets, and Pin Pairs	Returns the ratio of actual length divided by manhattan length
acGetAvgLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the average length of a group
acGetTotalLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the total length of a group, including manhattan ratsnest length of unrouted segments
acGetAvgEtchLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the average etch length of a group
acGetTotalEtchLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the total etch length of a group
acGetAvgManhattanLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the average manhattan length of a group
acGetTotalManhattanLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the total manhattan length of a group
acGetAvgPropDelay	Match Groups containing Pin Pairs	Returns the average propagation delay of a group
acGetTotalPropDelay	Match Groups containing Pin Pairs	Returns the total propagation delay of a group, including manhattan ratsnest delay of unrouted segments for pin pairs of a Match Group
acGetAvgEtchPropDelay	Match Groups containing Pin Pairs	Returns the average etch propagation delay of a group

Predicate	Scope	Function
acGetTotalEtchPropDelay	Match Groups containing Pin Pairs	Returns the total etch propagation delay for the pin pairs of a Match Group
acGetAvgManhattanPropDelay	Match Groups containing Pin Pairs	Returns the average manhattan propagation delay for the pin pairs of a Match Group
acGetTotalManhattanPropDelay	Match Groups containing Pin Pairs	Returns the average manhattan propagation delay of a group
acGetMinLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the minimum length of a group
acGetMinEtchLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the minimum etch length of a group
acGetMinManhattanLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the minimum manhattan length of a group
acGetMinPropDelay	Match Groups containing Pin Pairs	Returns the minimum propagation delay of a group
acGetMinEtchPropDelay	Match Groups containing Pin Pairs	Returns the minimum etch propagation delay of a group
acGetMinManhattanPropDelay	Match Groups containing Pin Pairs	Returns the minimum manhattan propagation delay of a group
acGetMaxLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the maximum length of a group
acGetMaxEtchLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the maximum etch length of a group
acGetMaxManhattanLength	Buses, Classes, Differential Pairs, and Match Groups	Returns the maximum manhattan length of a group
acGetMaxPropDelay	Match Groups containing Pin Pairs	Returns the maximum propagation delay of a group

Predicate	Scope	Function
acGetMaxEtchPropDelay	Match Groups containing Pin Pairs	Returns the maximum etch propagation delay of a group
acGetMaxManhattanPropDelay	Match Groups containing Pin Pairs	Returns the maximum manhattan propagation delay of a group
acIsRouted		Returns TRUE if an object is fully routed; nil otherwise. The corresponding measurement will populate the associated *Actual* with the strings *Routed* or *Unrouted*.
acIsPlaced		Returns TRUE if the components related to an object are all placed; nil otherwise. The corresponding measurement will populate the associated *Actual* with the strings *Placed* or *Unplaced*.
acGetParentName		Gets a Net Class for a net. This predicate does not have a corresponding measurement
acPutValue		Populates an attribute for an object. Should only be used in measurements.
acPutValue_p		Populates an attribute for an object/parent pair. Should only be used in measurements
acCreateResultName		Creates a name for a result object. Should only be used in measurements.
acAddResult		Associates a result object with a parent, which is usually the object that is being measured. Should only be used in measurements.

Predicate	Scope	Function
`acGetValue`		Gets the value of a specified cell. Requires information about the object, scope, parent object and scope, and attribute. Other versions of this predicate are available that require only a subset of this information; the rest is determined from the calling object. ■ `acGetValue_a` Requires only the attribute information ■ `acGetValue_o` Requires only the object information ■ `acGetValue_oa` Requires the object and attribute information ■ `acGetValue_po` Requires the parent and object information ■ `acGetValue_poa` Requires the parent, object, and attribute information ■ `acGetValue_so` Requires the scope and object information ■ `acGetValue_soa` Requires the scope, object, and attribute information

Predicate	Scope	Function
`acGetValueUnitString`		Returns a string containing the units used in the value specified by the parameters. For example, if the parameters are set up to access a Min Prop Delay constraint value of "10 ns," this function returns "ns." Most predefined properties in Constraint Manager do not contain the unit string; therefore, the function just returns an empty string.
`acDebug_print*`		These predicates correspond to the SKILL print functions, taking the same parameters. They only have effect when running under the Test command and print to the DEBUG section of the Test output. ■ `acDebug_print` corresponds to SKILL print ■ `acDebug_println` corresponds to SKILL println ■ `acDebug_printf` corresponds to SKILL printf ■ `acDebug_pprint` corresponds to SKILL pprint

Procedures

Adding a formula to a cell

1. Click in a cell that contains an existing constraint or property, and then

   - choose *Edit – Formula*

     -or-

   - right-click and choose *Formula* from the pop-up menu.

   Note: If the cell already contains a formula, clicking in the cell invokes formula edit mode.

2. Dismiss the informational dialog box.

   The *Single-line Editor* appears. If the formula was last edited in the *Multi-line Editor*, it will appear instead.

3. Build the formula using a combination of cell selection, operands, and predicates.

4. Choose a unit type that matches the cell's contents from the *Select Unit Type* drop-down menu. Depending on the cell, you might not have to choose a *Unit Type*.

5. Add a description.

6. Press *Test* to check for syntax errors.

7. Click *OK* to add the formula to the cell.

Editing an existing formula

1. Click in a cell that contains an existing formula.

   The *Single-line Editor* appears. If the formula was last edited in the *Multi-line Editor*, it will appear instead.

2. Edit the formula using a combination of cell selection, operands, and predicates.

3. Press *Test* to check for syntax errors.

4. Click *OK* to add the modified formula to the cell.

Deleting a formula from a cell

1. Hover your cursor over a cell that contains a formula.

   If you mistakenly clicked, the *Single-line Editor* appears. If the formula was last edited in the *Multi-line Editor*, it will appear instead.

2. Choose

   - *Edit – Clear*

     -or-

   - right-click and choose *Clear* from the pop-up menu.

Copying the contents of a cell that contains a formula

When you copy a formula, it does not maintain a relation to the original source. Edits to either formula (original or copied) are independent.

1. Hover your cursor over a cell that contains a formula.

   If you mistakenly clicked, the *Single-line Editor* appears. If the formula was last edited in the *Multi-line Editor*, it will appear instead.

2. Right click and choose *Copy* from the pop-up menu.
3. Click in a cell where you want to add a formula or the value of a formula.
4. Right-click and choose

   Paste to insert the formula's code

   -or-

   Paste Special, and then choose

   - *Formula* to insert the formula's code

     -or-

   - *Value* to insert the last calculated result of the formula.

Calculating Formulas

See Edit – Calculate on page 66 or Edit – Calculate All on page 67.

Edit – Calculate

Use this command to exercise a formula. For more information on formulas, see

- Edit – Formula on page 53
- Customizing Design Rule Checks in the *Constraint Manager User Guide*

Procedure

Calculating a formula

➤ Hover your mouse cursor over a cell (or a range of cells) that contains a formula, and then

- ❑ choose *Edit – Calculate*

  -or-

- ❑ right-click and choose *Calculate* from the pop-up menu.

Tip

If the formula of the cell that you are calculating depends on the value of an *Actual* cell, ensure that the dependent cell does not contain stale data (see Analyze – Analyze on page 181).

Edit – Calculate All

Use this command to exercise all formulas. For more information, see

- Edit – Formula on page 53
- Customizing Design Rule Checks in the *Constraint Manager User Guide*

If the formula of the cell that you are calculating depends on the value of an *Actual* cell, ensure that the dependent cell does not contain stale data (see Analyze – Analyze on page 181). Also, as some formulas depend on the results of other formulas, *Calculate All* iterates through the formulas (up to three passes) to ensure that all dependencies are updated.

A progress meter tracks the time remaining and provides feedback on aborted, partially calculated, and complete calculations.

Procedure

Calculating a formula

➤ Choose *Edit – Calculate All*.

Halting global calculations

➤ In the *progress meter*, click *Abort*.

Any formulas already calculated keep their new values, but they may be stale if they have dependencies on other formulas.

Objects Menu Commands

Objects – Filter

Procedures

Use this command to alter the display (rows) in the active net-level worksheet, according to a:

- filter setting specified in the *Filter* dialog box (*Objects – Filter*)
- regular expression (non-mathematical) filter setting in the first row of a column
- enumerated selection (value filter) setting in the first row of a column

Note: The fields in the *Filter* dialog box vary depending on which worksheet is active.

Tip

In addition to choosing the *Objects – Filter* command, you can click the first cell (*Type* column) in the top row of a worksheet to invoke the *Filter* dialog box.

Table 3-3 Filter Dialog Box

Use this field . . .	**To . . .**
Object types filter	Control which objects to display. The unchecked objects in this field restrict the objects available for display.
Restore default Object filter	Restore the object type filter settings to the default settings for the active worksheet.
Advanced filters	
Selected nets/xnets only	Show only those nets that are selected in PCB Editor's canvas. **Note:** *Application Select* must be enabled (choose *View – Options*).
Failed only	Show only those objects with constraint violations (*Actual* and *Margin* cells rendered red).
Constrained only	Show only the selected objects for which constraints are set.

Use this field . . .	To . . .
Active partition only	Show only those objects in the active partition (when running with a partitioned design).
Active DRCs	Show the design rule checks that are preserved. See Objects – Waive on page 80 for more information.
Waived DRCs	Show suppressed design rule checks. See Objects – Waive on page 80 for more information.
Reset all filters	Reset advanced filter settings for this dialog box, regular expressions in columns, and enumerated selections.

Procedures

You can use the different filtering techniques alone or in conjunction with each other. Each is explained in a separate procedure.

Tip

In addition to the pull-down menus, many commands have right-click menu and toolbar icon access.

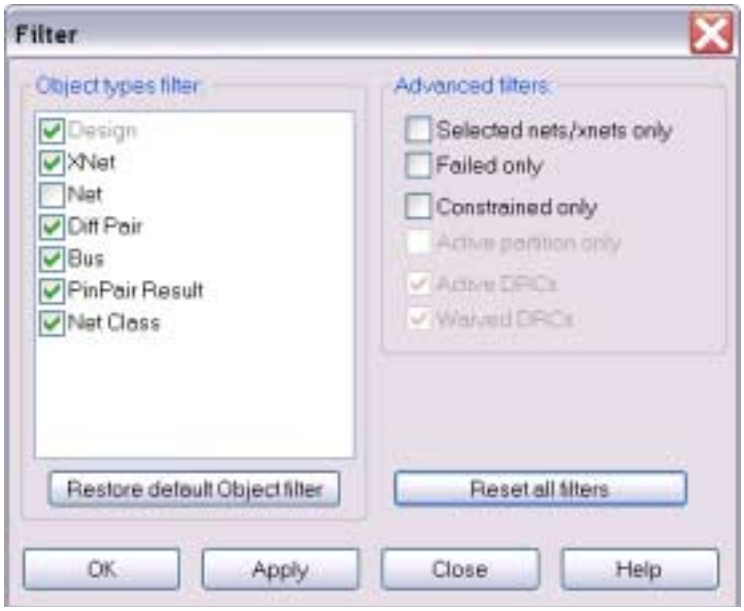
Important

Filters are not dynamic, they must be refreshed when an edit is made to a property or a design rule check runs. See the Objects – Filters re-apply command.

Filtering with the Filter dialog box

1. From an active worksheet, choose *Objects – Filter*.

   The *Filter* dialog box appears with default settings for the active worksheet. See Table 3-3 on page 69 for field descriptions.



2. Uncheck any object types that you want to filter out (hide).
3. Refine your view by specifying advanced filtering criteria, if desired.
4. Click *Apply* to execute the filter settings

   -or-

5. Click *OK* to execute the filter settings and dismiss the *Filter* dialog box.

Tip

By enabling XNet and Net together, you can display individual nets that comprise an extended (electrical) net.

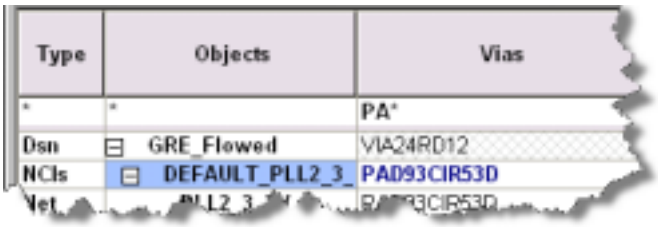
Filtering with regular expressions

The *Filter* handles regular expressions as string matches; they cannot be numerically evaluated.

1. From an active worksheet, click the top row of a non-numeric column, such as *Vias* in the *Physical* domain.

2. Replace the asterisk with a filter string.

   Note: You can discern whether a column contains a regular expression filter by the absence of an asterisk in the top row of the column.



3. Press *Return*.

   Only those objects that match the string that you entered are visible. If a member object of a container object matches the filter, the container object appears. You must manually expand the container to locate the member object that matches the expression.

Examples

Column	Expression	Matching Criteria	Result
Objects	`N*`	All objects that begin with ‘N’	`NET1, NET2, N3, N813, NB`
Vias	`*D*`	All vias that contain ‘D’ anywhere in the string	`SMD25_400, DIEN_FCU, SMD25_48`
Objects	`NET1|NET3`	Only explicit, exact matches	`NET1, NET3, (not NET2)`
Objects	`*[0-9]V*`	Only numbers that precede the letter ‘V’	`12V, 3.3V, 1.8V_OUT`

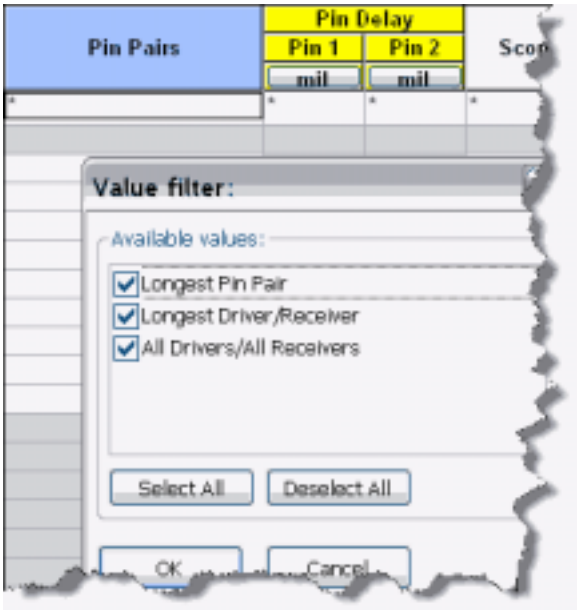
Note: You should precede special characters, such as `[ ]\^$.|?*+( )`, by the escape character `‘\’` for correct literal interpretation.

Filtering with enumerated selection (value filter)

The *Filter* takes advantage of defined lists, such as Object Types, CSets, and other columns with user-selectable values by presenting you with a list of check boxes to control what appears in the selected column.

1. From an active worksheet, click the top row of a non-numeric column (Pin Pairs, Scope, Pad-Pad Connect, Corner Type, to name a few).

   The *Value Filter* appears. This example results from clicking in the top row of the Pin Pairs column.



2. Uncheck any object types that you want to filter out (hide). Use *Select All* or *Deselect All* as necessary.

3. Click *OK*.

   Only those values that are checked appear. *FLTR* appears in the first row of the column with a value filter applied.

Objects – Filters re-apply

Use this command to refresh a stale filter. Filters are not dynamic, they must be refreshed when an edit is made to a property or after a design rule check runs.

Tip

In addition to the pull-down menus, this command has right-click menu and toolbar icon access.

Objects – Select

Use this command to select an object in Constraint Manager and crossprobe to locate that object in the PCB-, or schematic-editor, or in APD.

Tip

Choose an interactive command, such as *Info – Item*, in the layout tool and then choose *Objects – Select* on an object in Constraint Manager to view the operation on the selected object in the layout tool.

Procedure

1. In Constraint Manager, click on an object (Net, Xnet, Pin Pair, Diff Pair, DRC, or Bus).
2. Do one of the following:
   - Choose *Objects – Select*.

     - or -

   - Right-click and choose *Select* from the pop-up menu.

   Constraint Manager highlights the object in PCB-, or schematic-editor, or in APD.

Objects – Select and Show Element

Use this command to select an object in Constraint Manager and crossprobe to locate that object in the PCB-, or schematic-editor, or in APD, and run the Show Element command.

Procedure

1. In Constraint Manager, click on an object (Net, Xnet, Pin Pair, Diff Pair, DRC, or Bus).
2. Do one of the following:
   - Choose *Objects – Select and Show Element*.

     - or -

   - Right-click and choose *Select* from the pop-up menu.

   Constraint Manager highlights the object in PCB-, or schematic-editor, or in APD.

Objects – Deselect

Use this command to deselect an object in the PCB Editor or APD, which was crossprobed (Objects – Select) from Constraint Manager.

Procedure

➤ Do one of the following:

- Choose *Objects – Deselect*.

  - or -

- Right-click and choose *Deselect* from the pop-up menu.

The object is deselected in the PCB Editor or in APD.

Objects – Expand

Use this command to view the children of the selected object.

Procedure

1. Click on an object (*System*, *Design*, *Match Group*, or *Bus*).
2. Do one of the following:
   - ❑ Choose *Objects – Expand*.

     - or -
   - ❑ Right-click and choose *Expand* from the pop-up menu.

     - or -
   - ❑ Click the [+] symbol to the left of the object.

   The children of the object appear.

Objects – Collapse

Use this command to roll-up the children of the selected object into the parent object.

Procedure

1. Click on an object (*System*, *Design*, *Match Group*, or *Bus*).
2. Do one of the following:
   - ❑ Choose *Objects – Collapse*.

     - or -

   - ❑ Right-click and choose *Collapse* from the pop-up menu.

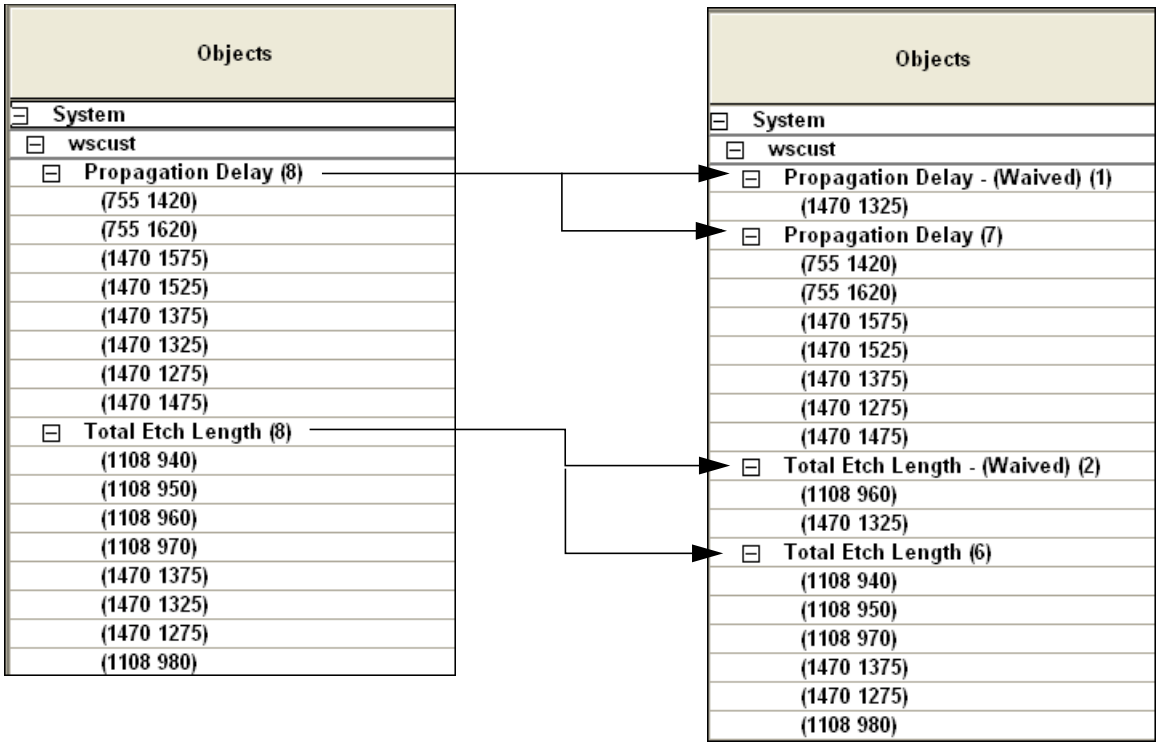
     - or -

   - ❑ Click the [-] symbol to the left of the object.

   The children of the selected object roll-up to the parent object.

Objects – Waive

Use this command to suppress (waive) the display of the design rule violations selected in the *Objects* column of Constraint Manager's DRC workbooks. Waived DRC errors appear with the term *Waived* (in parenthesis), and include the number of waived instances. A DRC bow tie that has *waived* status appears rotated 90 degrees in PCB Editor. Use Objects – Restore on page 82 to restore a waived DRC error.



Procedure

1. Expand the *DRC* workbook.
2. Expand a DRC worksheet (*Electrical*, *Spacing*, *Physical*, *Design*).
3. If necessary, expand the *DRC Group*.

   Individual instances of the DRC error appear and include their physical coordinates.
4. Adjust the *Active DRCs* and *Waived DRCs* filter options accordingly. See Objects – Filter on page 69.
5. Click to select a DRC instance to suppress.
6. Optionally, enter a user-friendly name in the *Comment* column.

7. Do one of the following:

- Choose *Objects – Waive*.

  - or -

- Right-click and choose *Waive* from the pop-up menu. Constraint Manager moves the DRC instance out of the *DRC Group* and into the *Waived DRC Group* of the same base name.

Tip

You can use Objects – Select to cross-probe design rule violations in your layout. You can also use the `no_zoom_to_object` environment variable (in PCB Editor, choose *Setup – User Preferences* and click the *Input* folder) to control the DRC display.

Objects – Restore

Use this command to *restore* the display of the *waived* design rule violations selected in the *Objects* column of Constraint Manager's DRC workbooks. Waived DRC errors appear with the term *Waived* (in parenthesis), and include the number of waived instances. A DRC bowtie that has *restored* status displays in PCB Editor. Use Objects – Waive on page 80 to waive a DRC error.

Procedure

1. Expand the *DRC* workbook.
2. Expand a DRC worksheet (*Electrical*, *Spacing*, *Physical*, *Design*).
3. If necessary, expand the *DRC Group*.

   Individual instances of the DRC appear along with physical coordinates.

   Adjust the *Active DRCs* and *Waived DRCs* filter options accordingly. See Objects – Filter on page 69.
4. Click to select a waived DRC instance to restore.
5. Choose *Objects – Restore.*

   - or -

   Right-click and choose *Restore* from the pop-up menu.

   Constraint Manager moves the DRC instance out of the Waived *DRC Group* and into the *DRC Group* of the same base name. Constraint Manager also clears the *Comment* field.

Objects – Create – Net Class

Use this command to group and constrain nets, Xnets, differential pairs, and buses that share common characteristics and require a similar constraint requirement.

For more information on the *Net Class* constraint object and the constraint system, see Net Class in the *Allegro Platform Constraints Reference*.

Creating a Net Class constraint object

1. From any *Electrical*, *Physical*, *Spacing*, or *Same Net Spacing* worksheet, select one or more nets, Xnets, differential pairs, or buses.
2. Do one of the following:
   - Choose *Objects – Create – Net Class*.

     -or-

   - Right-click and choose *Create – Net Class* from the pop-up menu.
3. Check *Create for physical and spacing* (physical and spacing only), if applicable.

   The same *Net Class* can exist in both the *Physical* and *Spacing* worksheets; the *Electrical* worksheet requires a unique *Net Class*. A *Net Class* created in the *Spacing* domain carries over to the *Same Net Spacing* domain. The converse is also true.
4. Enter a name, or accept the default.
5. Click *OK*.

   Constraint Manager adds a new *Net Class*, which you can identify by *NCls* in the *Type* column.

See also: Objects – Membership – Net Class on page 114.

Objects – Create – Net Class-Class

Use this command to define a relationship among two *Net Classes*.

For more information on the *Net Class-Class* constraint object and the constraint system, see Net Class-Class in the *Allegro Platform Constraints Reference*.

Creating a Net Class-Class constraint object

1. From any *Spacing* worksheet within the *Net Class-Class* folder, select a *Net Class*.

   Note: A *Net Class-Class* is not available in the *Same Net Spacing* domain.

2. Do one of the following:

   - Choose *Objects – Create – Net Class-Class*.

     -or-

   - Right-click and choose *Create – Net Class-Class* from the pop-up menu.

3. Specify the Class-to-Class relationship.
4. Click *OK*.

Constraint Manager adds a new *Net Class-Class*, which you can identify by *NCC* in the *Type* column.

Objects – Create – Region

Use this command to add or modify *Physical*, *Spacing*, or *Same Net Spacing* constraints on all nets that cross the boundaries of the *Region's* shape.

Note: You delimit a *Region* with a geometric shape, or a group of shapes, that you draw on a subclass layer in PCB Editor. You have the flexibility to create the *Region* constraint object or the Region's shape, in any order.

For more information on the *Region* constraint object and the constraint system, see Region in the *Allegro Platform Constraints Reference*.

Creating a Region constraint object

1. From the *Physical*, *Spacing*, or *Same Net Spacing* domain, expand the *Region* workbook.
2. From any *Physical*, *Spacing*, or *Same Net Spacing*-worksheet, do one of the following:
   - Choose *Objects – Create – Region*.

     -or-

   - When hovering over an existing *Region* object, right-click and choose *Create – Region* from the pop-up menu.
3. Optionally, check *Copy constraints from* and specify an existing *Region* if you want to seed this new *Region* from an existing one.
4. Enter a name or accept the default.
5. Click *OK*.

   Constraint Manager adds a new *Region*, which you can identify by *Rgn* in the *Type* column.
6. In PCB Editor,
   a. Choose *Shape* – [*Polygon, Rectangular, Circular*].
   b. Choose *Constraint Region* as the Active Class.
   c. Choose *All*, *Outer*, *Inner Signal*, *Inner Plane*, or *by layer* for a subclass layer.
   d. In the *Assign to region* field, choose an existing *Region* from the drop-down menu, or enter the name of a new *Region*.

e. Draw the shape (you can have multiple shapes of the same name on a layer).

f. Right-click and choose *Done* from the pop-up menu.

Objects – Create – Region Class

Use this command to constrain the members of a *Net Class*—within a *Region*—differently than the original constraints in that *Region*.

For more information on the *Region Class* constraint object and the constraint system, see Region Class in the *Allegro Platform Constraints Reference*.

Creating a Region constraint object

1. From the *Physical*, *Spacing*, or *Same Net Spacing* domain, expand the *Region* workbook.
2. In the *Objects* column, select a *Region*.
3. Do one of the following:
   - Choose *Objects – Create – Region Class*.

     -or-

   - When hovering over an existing Region object, click-right and choose *Create – Region Class* from the pop-up menu.
4. In the left column, choose a *Region* name.
5. In the right column, choose one or more *Net Classes*.
6. Enter a name or accept the default.
7. Click *OK*.

   Constraint Manager adds a new *Region Class*, which you can identify by *RcIs* in the *Type* column.

Objects – Create – Region Class-Class

Use this command to specify unique spacing relationships between two *Net Classes* within that *Region.*

For more information on the *Region Class-Class* constraint object and the constraint system, see Region Class-Class in the *Allegro Platform Constraints Reference*.

Creating a Region Class-Clas constraint object

1. From the *Spacing* domain, expand the *Region* workbook.

   Note: A *Region Class-Class* is not available in the *Same Net Spacing* domain.

2. In the *Objects* column, select a *Region*.
3. Do one of the following:

   - Choose *Objects – Create – Region Class-Class*.

     -or-

   - When hovering over an existing Region object, choose *Create – Region Class-Class* from the pop-up menu.

4. In the left column, choose from the list of defined *Regions*.
5. In the middle-column, choose from the list of defined *Net Classes*.
6. In the right column, choose from the list of defined *Net Classes*.
7. Click *OK*.

Constraint Manager adds a new *Region Class-Class*, which you can identify by *RCC* in the *Type* column.

Objects – Create – Bus

Use this command to group functionally similar nets, Xnets, and differential pairs. Constraints captured on a bus will be inherited by all members of the bus

Tip

For more information, see Buses in the *Allegro® Constraint Manager User Guide*.

Procedures

Creating a bus

1. Select a group of nets, Xnets, or differential pairs.
2. Do one of the following:
   - ❑ Choose *Objects – Create – Bus*

     – or –

   - ❑ Right-click and choose *Create – Bus* from the pop-up menu.
3. Enter a name for the bus.
4. Click *OK*.

Deleting a bus

1. Select a bus.
2. Do one of the following:
   - ❑ Choose *Objects – Delete*

     – or –

   - ❑ Press *Delete*.

   Constraint Manager destroys the bus object and preserves the individual members.

Renaming a bus

1. Select a bus.
2. Do one of the following:
   - Choose *Objects – Rename*

     – or –

   - Right-click and choose *Rename* from the pop-up menu.

Creating an Electrical CSet based on an existing bus

1. Select a bus.
2. Do one of the following:
   - Choose *Objects – Create Electrical CSet.*

     – or –

   - Right-click and *Create – Electrical CSet* from the pop-up menu.

Selecting a bus in the PCB Editor

1. In Constraint Manager, select a bus.
2. Do one of the following:
   - Choose *Objects – Select* (or *Deselect*).

     – or –

   - Right-click and choose Select (or *Deselect*) from the pop-up menu.

   The select object in Constraint Manager is highlighted (or de highlighted) in PCB Editor or in APD.

Examining bus membership

1. Select a bus.
2. Do one of the following:
   - Choose *Objects – Membership – Bus.*

     – or –
   - Right-click and choose *Membership – Bus* from the pop-up menu.

     – or –
   - Right-click and choose *Expand* from the pop-up menu.

Redefining bus membership

1. Select a bus.
2. Do one of the following:
   - Choose *Objects – Membership – Bus.*

     – or –
   - Right-click and choose *Membership – Bus* from the pop-up menu.
3. Add or remove nets, Xnets, or pin pairs as appropriate.
4. Click *OK*.

Objects – Create – Match Group

Use this command to create a Match Group. This command applies only to the Relative Propagation Delay worksheet in the Wiring workbook.

Tip

A *Match Group* can also be created when applying an Electrical CSet. For more information, see Relative/Match Group in the *Allegro® Constraint Manager User Guide*.

Procedures

Creating a Match Group

1. From the *Relative Propagation Delay* worksheet, select the desired nets, Xnets, or pin pairs.
2. Do one of the following:
   - ❑ Choose *Objects – Create – Match Group*.

     - or -

   - ❑ Right-click and choose *Create – Match Group* from the pop-up menu.
3. Enter a name for the *Match Group*.
4. If the seed net or Xnet for the new *Match Group* is a member of en existing *Match Group*, click *Preserve existing membership* if you want the net or Xnet in multiple groups; otherwise, Constraint Manager will remove the member from the existing group.

5. For the *Match Group* members, in the *Relative Propagation Delay* worksheet, specify:

   a. Pin Pairs (*Longest Pin Pair, Longest Driver/Receiver, All Drivers/All Receivers*).

   b. Scope (*Local* or *Global*)

      Note: A common practice with many designs is to define *Match Groups* in an *Electrical CSet*. In addition to *Local*- and *Global*-, you can choose *Bus*- and *Class*- scopes. When the *Electrical CSet* is referenced, the system automatically creates *Match Groups* based upon the *Bus* and *Class* collections in the design. See the Constraint Manager User Guide for complete details on working with *Match Groups*.

   c. *Delta* and *Tolerance* values.

6. Click *OK*.

Deleting a Match Group

1. In the *Objects* column, select a *Match Group*.
2. Do one of the following:

   - ❑ Choose *Objects – Delete*.

     - or -

   - ❑ Right-click and choose *Delete* from the pop-up menu.

     - or -

   - ❑ Click *Delete*.

   Constraint Manager destroys the *Match Group* and keeps preserves individual members.

Renaming a Match Group

1. Select a *Match Group*.
2. Do one of the following:

   - ❑ Choose *Objects – Rename*.

     - or -

- Right-click and choose *Rename* from the pop-up menu.

3. Enter a new name for the *Match Group*.
4. Click *OK*.

Selecting or deselecting a Match Group in the PCB Editor or APD

1. In the *Objects* column of Constraint Manager, click on a *Match Group*.
2. Do one of the following:
   - Choose *Objects – Select* (or *Deselect*).

     - or -

   - Right-click and choose *Select* (or *Deselect*) from the pop-up menu.

   The *Match Group* is selected (or deselected) in PCB Editor or in APD.

Examining Match Group membership

1. Select a *Match Group*.
2. Do one of the following:
   - Choose *Objects – Membership – Match Group.*

     – or –

   - Right-click and choose *Membership – Match Group* from the pop-up menu.

     – or –

   - Right-click and choose *Expand* from the pop-up menu.

Redefining Match Group membership

1. Select a *Match Group*.
2. Do one of the following:
   - Choose *Objects – Membership – Match Group.*

     – or –

   - Right-click and choose *Membership – Match Group* from the pop-up menu.
3. Add or remove nets, Xnets, or pin pairs as appropriate.

4. If the seed net or Xnet for the new *Match Group* is a member of an existing *Match Group*, click *Preserve existing membership* if you want the net or Xnet in multiple groups; otherwise, Constraint Manager will remove the member from the existing group.
5. Click *OK*.

Objects – Create – Ratsnest Bundle

Use this command to group pin pairs, which you want routed together. You can select member pin pairs from the same, or different, net. Setting attributes on a ratsnest bundle affects all member pin pairs of that bundle. You set these attributes in the *Ratsnest Bundle Properties* worksheet in the *Properties* domain.

Tip

For more information, see Ratsnest Bundle in the *Allegro® Constraint Manager User Guide*.

Procedures

Creating a ratsnest bundle

1. In the *Ratsnest Bundle Properties* worksheet, select one or more pin pairs.
2. Do one of the following:
   - Choose *Objects – Create – Ratsnest Bundle*

     – or –
   - Right-click and choose *Create – Ratsnest Bundle* from the pop-up menu.

   The *Ratsnest Bundle* dialog box appears.
3. Enter a name for the bundle.
4. Click *OK*.

   The bundle appears in the *Objects* column, labeled *RBnd*.

   Note: You can create an empty ratsnest bundle and add members later. See the Objects – Membership – Ratsnest Bundle command.

Defining attributes for a ratsnest bundle

1. In the *Objects* column of the *Ratsnest Bundle Properties* worksheet, select a *Ratsnest Bundle*.

2. Define attributes for each cell in the worksheet.

   For functional descriptions of each cell, see the bundle properties command in B commands chapter of the *Allegro PCB and Package Physical Layout Command Reference.*

Deleting a ratsnest bundle

1. Select a *Ratsnest Bundle*.
2. Do one of the following:
   - Choose *Objects – Delete*

     – or –

   - Press *Delete*.

   Constraint Manager destroys the bundle object and preserves the individual pin pair members.

Renaming a ratsnest bundle

1. Select a *Ratsnest Bundle*.
2. Do one of the following:
   - Choose *Objects – Rename*

     – or –

   - Right-click and choose *Rename* from the pop-up menu.

Cross-probing a ratsnest bundle

1. In Constraint Manager, select a *Ratsnest Bundle*.
2. Do one of the following:
   - Choose *Objects – Select* (or *Deselect*).

     – or –

   - Right-click and choose Select (or *Deselect*) from the pop-up menu.

   The selected *Ratsnest Bundle* in Constraint Manager becomes highlighted (or de highlighted) in PCB Editor or in APD.

Examining ratsnest bundle membership

See Objects – Membership – Ratsnest Bundle on page 124.

Redefining ratsnest bundle membership

See Objects – Membership – Ratsnest Bundle on page 124.

Objects – Create – Pin Pair

Use this command to capture specific pin-to-pin constraints for a net or an Xnet. You can also use pin pairs to capture generic pin-to-pin constraints for ECSets. Generic pin pairs are used to automatically define net- or Xnet-specific pin pairs when the Electrical CSet is referenced. Once established, a pin pair is associated with an Electrical CSet.

Tip

Pin Pairs are created automatically by applying an Electrical CSet containing the following Net- or Xnet-level constraints:

- Propagation Delay
- Relative Propagation Delay
- Impedance Rule
- Min Switch
- Max Settle

For more information, see Pin Pairs in the *Allegro® Constraint Manager User Guide*.

Procedures

Creating a pin pair

1. In the *Switch/Settle Delay, Setup/Hold*, *Impedance*, *Min/Max Propagation Delay*, or *Relative Propagation Delay* worksheet in the Electrical domain (or Net worksheets in the Physical or Spacing domains), select an Xnet or net.
2. Do one of the following:
   - Choose *Objects – Create – Pin Pair*.

     - or -

   - Right-click and choose *Create – Pin Pair* from the pop-up menu.

   The *Create Pin Pair* dialog box appears containing all pins on the selected net.
3. Match up a pin in the Driver column with a mate in the Receiver column.
4. Click *Apply* or *OK*.

Deleting a pin pair

1. In the *Objects* column, select a pin pair.
2. Do one of the following:
   - ❑ Choose *Objects – Delete*.

     - or -

   - ❑ Right-click and choose *Delete* from the pop-up menu.

     - or -

   - ❑ Click *Delete*.

   Constraint Manager destroys the pin pairing and preserves individual pins.

Selecting or deselecting a pin pair in the PCB Editor or in APD

1. In the *Objects* column of Constraint Manager, click on a pin pair.
2. Do one of the following:
   - ❑ Choose *Objects – Select* (or *Deselect*).

     - or -

   - ❑ Right-click and choose *Select* (or *Deselect*) from the pop-up menu.

   The pin pair is selected (or deselected) in PCB Editor or in APD.

Objects – Create – Differential Pair

Procedures

Use this command to create a *user-defined* differential pair object.

Note: Use SigXplorer to create *model-defined* differential pairs.

Tip

To learn more about working with Differential Pair constraints, see

- Best Practices: Working with Differential Pairs
- Differential Pair Constraint Data Sheets in the *Allegro Platform Constraints Reference*
- Differential Pairs in the *Allegro® Constraint Manager User Guide*

Differential Pair Create Dialog Box

Use this field . . .	**To . . .**
Filter drop-down menu	Focus a selection to nets, Xnets, or differential pair objects. When you choose nets or Xnets, any associated differential pair objects display as well.
Filter type in field	Enter a string (and wildcard characters) to further focus on the object types chosen with the filter drop-down menu.
Left column	List all available nets, Xnets, or differential pair objects in the design. Use the filter drop-down menu and type-in field to limit the list. **Note:** If a net or Xnet is a member of an existing differential pair object, it is not an eligible member for a new differential pair object.
Selections (right column)	List the members of the differential pair objects chosen in the left column or individual nets or Xnets moved from the left column.
Diff Pair Name	Enter a name for a user-defined differential pair. You cannot create a model-defined differential pair in Constraint Manager.

Use this field . . .	To . . .
Modify	Implement a modification to a differential pair name or its membership. This button is inactive for model-defined differential pair objects.
Delete	Remove the selected net or Xnet member of the differential pair (right column), effectively destroying the differential pair. The net or Xnet is preserved in the design.
Auto Setup	Invokes the differential pair automatic setup dialog box (described next).
Create	Create the differential pair object with the two nets or Xnets in the selections column as its members.

Differential Pair Automatic Setup Dialog Box

Use this field . . .	To . . .
Filter drop-down menu	Focus a selection to *Nets*, *Xnets*, or *Xnets to be split*.
Filter type in field	Enter a string (and wildcard characters) to further focus on the object types chosen with the filter drop-down menu.
Left column	List all available *Nets* or *Xnets* in the design, including *Xnets to be split*. Use the filter drop-down menu and type-in field to limit the list.
Selections (right column)	Lists auto-generated differential pair objects and their member nets or Xnets.
Prefix	Specify a string to precede the base name of the auto-generated differential pair object.
+ Filter	Specify a character that differentiates a common net or Xnet base name. This identifies the *non-inverting* net or Xnet of the auto-generated differential pair object.
- Filter	Specify a character that differentiates a common net or Xnet base name. This identifies the *inverting* net or Xnet of the auto-generated differential pair object.

Use this field . . .	To . . .
Create	Create differential pair objects by ■ Examining the nets and Xnets in the filter list (on the left). ■ Mating nets and Xnets with a common name, differentiated with a post fix (+/- Filter). ■ Prefixing the auto-generated differential pair object name with a string (Prefix), if specified. ■ Populating the column on the right with auto-generated differential pair objects.

Important

If you connect a resistor between the inverting and non-inverting signals in an external parallel termination network, and the resistor has an attached signal model, both nets become an Xnet. For Auto-setup to work in this configuration, you must select *Xnets to be split* from the drop-down menu.

Procedures

Creating a differential pair object

1. In the *Objects* column, click on a net or Xnet that is not a member of a differential pair object.
2. Do one of the following:
   - ❑ Choose *Objects – Create – Differential Pair*.

     - or -

   - ❑ Right-click and choose *Create – Differential Pair* from the pop-up menu.

   Note: You can also select a net or Xnet (and optionally its mate) that is not a member of a differential pair object to seed the new differential pair.

   The *Create Differential Pair* dialog box appears.
3. In the *Filter* drop-down menu, choose *net* or *Xnet*.
4. If necessary, enter a string in the filter type-in field to focus the search.
5. Click on the net or Xnet and click **>** to move it to the selected column (right).

Note: A differential pair object consists of a net or Xnet and its mate. You cannot choose a mate that is part of another differential pair object.

6. Click on the mate net or Xnet and click **>** to move it to the selected column (right).
7. Click *Create*.
8. Click *Close* to dismiss the dialog box.

Auto-creating differential pair objects

1. In the *Objects* column, click on a net or Xnet that is not a member of a differential pair object.

Important

If two nets or Xnets — each comprised of two Xnets joined by a series resistor — are connected by a parallel termination resistor, the auto-generate function considers the circuit as a single Xnet. Therefore, to auto-generate differential pairs for these Xnets, you must select *Xnets to be split* from the drop-down menu.

Alternatively, you could (1) temporarily remove the signal models from the resistors, (2), auto-generate to create the differential pair objects and (3), reassign signal models to the resistors.

2. Do one of the following:

   - Choose *Objects – Create – Differential Pair*.

     - or -

   - Right-click and choose *Create – Differential Pair* from the pop-up menu.

   The Create Differential Pair dialog box appears.

3. Click *Auto Setup*.

   The *Differential Pair Automatic Setup* dialog box appears.

4. In the *Filter* drop-down menu, choose *net, Xnet,* or *Xnets to be split*.
5. If necessary, enter a string in the filter type-in field to focus the search.
6. In the *Prefix* field, enter a string that you want to precede the base name of the auto-generated differential pair object.

7. In the *+Filter* field, enter a character that differentiates a common net or Xnet base name.

   This identifies the *non-inverting* net or Xnet of the auto-generated differential pair object.

8. In the *-Filter* field, enter a character that differentiates a common net or Xnet base name.

   This identifies the *inverting* net or Xnet of the auto-generated differential pair object.

   Note: With values in both the *+Filter* and *-Filter*, the selections field lists all the differential pairs to generate. If *net* or *Xnet* is selected in the *Filter* drop-down menu, Constraint Manager populates the list by matching all net or Xnet names against the *+Filter* and *-Filter*. If you choose *Xnets to be split*, the list is determined by checking only the nets that compose each Xnet. Review this list and remove any Xnets for which you do not want to auto-generate a differential pair by highlighting them and clicking *Remove*.

9. Click *Create*.

   Constraint Manager attempts to create all of the differential pairs still appearing in the list. A log file lists all differential pairs created, and it includes any errors encountered.

10. Click *Close* repeatedly to dismiss each dialog box.

Deleting a differential pair object

1. In the *Objects* column, click on a user-defined differential pair object.

   Note: You cannot delete a model-defined differential pair object in Constraint Manager.

2. Click *Delete*.

   A confirmer appears.

3. Click *Yes*.

   Constraint Manager destroys the differential pair object and preserves the member nets and Xnets.

Analyzing a differential pair object

1. Choose *Analyze – Analysis Modes*.
2. Enable *All differential pair checks*.
3. In the *Objects* column. click on a differential pair object.
4. Choose *Analyze – Analyze*.

Extracting a differential pair object into SigXplorer

1. In the *Objects* column. click on a differential pair object.
2. Choose *Tools – SigXplorer*.

   If the differential pair object is model-defined, both net or Xnet members of the differential pair are extracted. If the differential pair object is user-defined, only one member of the differential pair object is extracted.

Objects – Create – Electrical CSet

Use this command to create an *Electrical CSet*. An electrical constraint set is a collection of constraints, and their default values, which reflect a particular design requirement. You can capture any, or all, electrical constraints, including topology-related information, in an *Electrical CSet*.

You define generic rules, such as ECSets, under the *Electrical Constraint Set* object folder. These generic rules can subsequently be applied to net-related objects.

You can also define an *Electrical CSet* based on the characteristics of a net or Xnet. Defining net-derived rules lets you create (or clone) rules based on the electrical characteristics of the physical net in your design.

Tip

Refer to Constraint Sets in the *Allegro® Constraint Manager User Guide* for more information on ECSets.

Procedures

Creating an empty Electrical CSet

1. In the *Electrical Constraint Set* folder or in the *Net* folder, click on a workbook, or a worksheet within a workbook.
2. Do one of the following:
   - Choose *Objects – Create – Electrical CSet*.

     - or -

   - Right-click and choose *Create Electrical CSet* from the pop-up menu.

   The *Create Electrical CSet* dialog box appears.
3. Ensure that *copy constraints from* is deselected.
4. Enter a name for the Electrical CSet.
5. Click *OK*.

   Constraint Manager creates the Electrical CSet where it can be selected in the *Objects* column.

6. Specify the desired constraint parameters.
7. Assign the Electrical CSet to net-related objects (see "Objects – Constraint Set References" on page 128).

Creating an Electrical CSet based on another Electrical CSet

1. In the *Objects* column, click on an existing Electrical CSet.
2. Do one of the following:
   - ❑ Choose *Objects – Create – Electrical CSet*.

     - or -

   - ❑ Right-click and choose *Create Electrical CSet* from the pop-up menu.

   The *Create Electrical CSet* dialog box appears.
3. Ensure that *copy constraints from* is selected.
4. Enter a name for the Electrical CSet.
5. Click *OK*.

   Constraint Manager creates a new Electrical CSet with the definitions of the seed Electrical CSet.
6. Change constraint parameters as desired.
7. Assign the Electrical CSet to net-related objects (see "Objects – Constraint Set References" on page 128).

Creating an Electrical CSet based on a physical net

1. In the *Nets* folder, click on a workbook, or a worksheet within a workbook.
2. Click on a net.
3. Do one of the following:
   - ❑ Choose *Objects – Create – Electrical CSet*.

     - or -

   - ❑ Right-click and choose *Create Electrical CSet* from the pop-up menu.

   The Create Electrical CSet dialog box appears.

4. Ensure that *copy constraints from* is selected.
5. Enter a name for the Electrical CSet.
6. Click *OK*.

   Constraint Manager creates the Electrical CSet where it can be selected in the *Objects* column.
7. Specify the desired constraint parameters.

   Assign the Electrical CSet to net-related objects (see "Objects – Constraint Set References" on page 128).

Objects – Create – Physical CSet

Use this command to create a *Physical CSet*. A physical constraint set is a collection of constraints, and their default values, which reflect a particular design requirement. All *Designs* begin with the *DEFAULT Physical CSet*, which is pre-populated with constraints and their values. You cannot delete the *DEFAULT Physical CSet*, nor can you remove any constraints from it; you can, however, change the constraint values within it.

You define physical rules under the *Physical Constraint Set* object folder. These rules can subsequently be applied to net-related objects.

You can create a *Physical CSet* to suit your specific needs by cloning the *DEFAULT,* or another, *Physical CSet*, renaming it, and redefining its constraint values.

 *Tip*

Refer to Constraint Sets in the *Allegro® Constraint Manager User Guide* for more information on CSets.

Creating a Physical CSet

1. In the *Objects* column, click on an existing *Physical CSet*.
2. Do one of the following:
   - ❑ Choose *Objects – Create – Physical CSet*.

     - or -

   - ❑ Right-click and choose *Create Physical CSet* from the pop-up menu.

   The *Create Physical CSet* dialog box appears and the seed constraint is identified by *copy constraints from*.
3. Enter a name for the *Physical CSet*.
4. Click *OK*.

   Constraint Manager creates a new *Physical CSet* with the values of the seed object.
5. Change constraint values as desired.
6. Assign the *Physical CSet* to net-based objects (see "Objects – Constraint Set References" on page 128).

Objects – Create – Spacing CSet

Use this command to create a *Spacing CSet*. A spacing constraint set is a collection of net-to-net constraints, and their default values, which reflect a particular design requirement. All *Designs* begin with the *DEFAULT Spacing CSet*, which is pre-populated with constraints and their values. You cannot delete the *DEFAULT Spacing CSet*, nor can you remove any constraints from it; you can, however, change the constraint values within it.

You define spacing rules under the *Spacing Constraint Set* object folder. These rules can subsequently be applied to net-related objects.

You can create a *Spacing CSet* to suit your specific needs by cloning the *DEFAULT,* or another, *Spacing CSet,* renaming it, and redefining its constraint values.

Tip

Refer to Constraint Sets in the *Allegro® Constraint Manager User Guide* for more information on CSets.

Creating a Spacing CSet

1. In the *Objects* column, click on an existing *Spacing CSet*.
2. Do one of the following:
   - ❑ Choose *Objects – Create – Spacing CSet*.

     - or -

   - ❑ Right-click and choose *Create Spacing CSet* from the pop-up menu.

   The *Create Spacing CSet* dialog box appears and the seed constraint is identified by *copy constraints from*.
3. Enter a name for the *Spacing CSet*.
4. Click *OK*.

   Constraint Manager creates a new *Spacing CSet* with the values of the seed object.
5. Change constraint values as desired.

Assign the Spacing CSet to net-based objects (see “Objects – Constraint Set References” on page 128).

Objects – Create – Same Net Spacing CSet

Use this command to create a *Same Net Spacing CSet*. A same net spacing constraint set is a collection of spacing constraints on the same net, and their default values, used to control spacing checks among objects on the same net. All *Designs* begin with the *DEFAULT Same Net Spacing CSet*, which is pre-populated with constraints and their values. You cannot delete the *DEFAULT Spacing CSet*, nor can you remove any constraints from it; you can, however, change the constraint values within it.

You define same net spacing rules under the *Same Net Spacing Constraint Set* object folder. These rules can subsequently be applied to net-related objects.

You can create a *Same Net Spacing CSet* to suit your specific needs by cloning another *Same Net Spacing CSet,* renaming it, and redefining its constraint values.

Important

The *Same Net Spacing* domain supports by-layer constraint modes. See Same Net Spacing DRC Modes in the *Constraint Manager User Guide*.

Tip

Refer to Constraint Sets in the *Allegro® Constraint Manager User Guide* for more information on CSets.

Creating a Same Net Spacing CSet

1. In the *Objects* column, click on an existing *Same Net Spacing CSet*.
2. Do one of the following:
   - ❑ Choose *Objects – Create – Same Net Spacing CSet*.

     - or -

   - ❑ Right-click and choose *Create Same Net Spacing CSet* from the pop-up menu.

   The *Create Same Net Spacing CSet* dialog box appears and the seed constraint is identified by *copy constraints from*.
3. Enter a name for the *Same Net Spacing CSet*.
4. Click *OK*.

   Constraint Manager creates a new *Same Net Spacing CSet* with the values of the seed object.

5. Change constraint values as desired.

Assign the Same Net Spacing CSet to net-based objects (see "Objects – Constraint Set References" on page 128).

Objects – Membership – Net Class

Use this command to redefine the membership of an existing *Net Class*.

Procedure

1. In the *Objects* column, select a *Net Class*.

   To aid you in locating a *Net Class*, use the *Edit – Find* and *Objects – Filter* commands.

   A *Net Class* in the *Physical* domain may also be located in the *Spacing* domain; the converse is also true. A *Net Class* that was created in the *Electrical* domain is unique to that domain.

2. Choose *Objects – Membership – Net Class*.

   The *Net Class Membership* dialog box appears.

3. Optionally, use the *Object Type* drop-down menu in the in the *Net Class Membership* dialog box to filter your selection.

4. Use the left and right arrows to populate the *Existing Members* column to suit your needs.

5. Click *OK*.

Objects – Membership – Bus

Procedures

Use this command to examine or redefine the members (bits) of a bus.

Tip

Refer to the *Allegro® Constraint Manager User Guide* for more information on bus objects.

Bus Membership Dialog Box

When you select a bus member to invoke this command, you get a dialog box that lets you assign the member to a different bus or none. You can also examine all members that comprise the bus. When you select a bus object, you get the dialog box described in the following table.

Use this field . . .	**To . . .**
Filter drop-down menu	Focus a selection to nets or Xnets.
Filter type in field	Enter a string (and wildcard characters) to further focus on the object types chosen with the filter drop-down menu.
Left column	List all available nets or Xnets in the design. Use the filter drop-down menu and type-in field to limit the list. **Note:** If a net or Xnet is a member of an existing differential pair object, it is not an eligible member for a new differential pair object.
Members (right column)	List the current members of the bus.

Procedures

Examining a bus

1. In the *Objects* column, click on a bus.

2. Do one of the following:

   - Choose *Objects – Membership – Bus*.

     - or -

   - Right-click and choose *Membership– Bus* from the pop-up menu.

   The Bus Membership dialog box appears with the bus members listed in the right column.

Redefining bus membership

1. In the *Objects* column, click on a bus.
2. Do one of the following:

   - Choose *Objects – Membership – Bus*.

     - or -

   - Right-click and choose *Membership – Bus* from the pop-up menu.

   The Bus Membership dialog box appears with the bus members listed in the right column.
3. Use the left and right arrows to redefine the bus as desired.

Objects – Membership – Match Group

Procedures

Use this command to examine or redefine the members of a *Match Group*.

Tip

Refer to the *Allegro® Constraint Manager User Guide* for more information on Match Group objects.

Match Group Membership Dialog Box

When you select a *Match Group* member to invoke this command, a dialog box appears that lets you assign the member to a different group (or groups) or no *Match Group*. You can also examine all members that comprise the *Match Group*. When you select a *Match Group* object, you get the dialog box described in the following table.

Use this field . . .	**To . . .**
Filter drop-down menu	Focus a selection to nets, Xnets, or pin pairs.
Filter type in field	Enter a string (and wildcard characters) to further focus on the object types chosen with the filter drop-down menu.
Left column	List all available nets, Xnets, or pin pairs in the design.
	Use the filter drop-down menu and type-in field to limit the list.
Members (right column)	List the current members of the *Match Group*.
Preserve existing membership (checkbox)	When checked, preserves the member net or Xnet in the original *Match Group* in addition to its membership in a new *Match Group*.
	When unchecked, removes the member net or Xnet in the original *Match Group* when you specify membership in a new *Match Group*.

Procedures

Examining a Match Group

1. In the *Objects* column, click on a *Match Group*.
2. Do one of the following:

   - Choose *Objects – Membership – Match Group*.

     - or -

   - Right-click and choose *Membership – Match Group* from the pop-up menu.

   The *Match Group* Membership dialog box appears with the bus members listed in the right column.

Redefining Match Group membership

1. In the *Objects* column, click on a Match Group.
2. Do one of the following:

   - Choose *Objects – Membership – Match Group*.

     - or -

   - Right-click and choose *Membership – Match Group* from the pop-up menu.

   The *Match Group* Membership dialog box appears with the *Match Group* members listed in the right column.
3. Use the left and right arrows to redefine the *Match Group* as desired.
4. If the seed net or Xnet for the new *Match Group* is a member of an existing *Match Group*, click *Preserve existing membership* if you want the net or Xnet in multiple groups; otherwise, Constraint Manager will remove the member from the existing group.
5. Click *OK*.

Objects – Membership – Differential Pair

Procedures

Use this command to examine or modify the members of a differential pair object.

Tip

Refer to the *Allegro® Constraint Manager User Guide* for more information on differential pair objects.

You can examine or change the members of a *user-defined* differential pair object in Constraint Manager. You can only examine the members of a *model-defined* differential pair in Constraint Manager.

Tip

You can change members of a model-defined differential pair by editing the IBIS device model in PCB Editor, APD, or SigXplorer. This change is then reflected in Constraint Manager in real time or when importing a topology template from SigXplorer.

Differential Pair Membership Dialog Box

Use this field . . .	**To . . .**
Filter drop-down menu	Focus a selection to nets, Xnets, or differential pair objects. When you choose nets or Xnets, any associated differential pair objects display as well.
Filter type in field	Enter a string (and wildcard characters) to further focus on the object types chosen with the filter drop-down menu.
Left column	List all available nets, Xnets, or differential pair objects in the design. Use the filter drop-down menu and type-in field to limit the list.
Selections (right column)	List the members of the differential pair objects chosen in the left column or individual nets or Xnets moved from the left column.

Use this field . . .	To . . .
Diff Pair Name	Enter (or *Modify*) a name for a user-defined differential pair. You can also rename a differential pair with the *Objects – Rename* command. You cannot rename a model-defined differential pair in Constraint Manager.
Modify	Implement a modification to a differential pair name or its membership. This button is inactive for model-defined differential pair objects.
Create	Create the differential pair object with the two nets or Xnets in the selections column as its members.
Delete	Remove the selected net or Xnet member of the differential pair (right column), effectively destroying the differential pair. The net or Xnet is preserved in the design.
Auto Setup	Invokes the differential pair automatic setup dialog box (described next).

Differential Pair Automatic Setup Dialog Box

Use this field . . .	To . . .
Filter drop-down menu	Focus a selection to nets or Xnets.
Filter type in field	Enter a string (and wildcard characters) to further focus on the object types chosen with the filter drop-down menu.
Left column	List all available nets or Xnets in the design. Use the filter drop-down menu and type-in field to limit the list.
Selections (right column)	Lists auto-generated differential pair objects and their member nets or Xnets.
Prefix	Lets you specify a string to precede the base name of the auto-generated differential pair object.

Use this field . . .	To . . .
+ Filter	Lets you specify a character that differentiates a common net or Xnet base name. This identifies the *non-inverting* net or Xnet of the auto-generated differential pair object.
- Filter	Lets you specify a character that differentiates a common net or Xnet base name. This identifies the *inverting* net or Xnet of the auto-generated differential pair object.
Create	Creates differential pair objects by ■ Examining the nets and Xnets in the filter list (on the left). ■ Mating nets and Xnets with a common name, differentiated with a post fix (+/- Filter). ■ Prefixing the auto-generated differential pair object name with a string (Prefix), if specified. ■ Populating the column on the right with auto-generated differential pair objects.

Procedures

Changing the members of a differential pair

In Constraint Manager, you can change only the members of a user-defined differential pair.

1. Do one of the following:
   - ❑ Choose *Objects – Differential Pair Membership*.

     - or -

   - ❑ In the Objects column, click on a differential pair, then right-click and choose *Membership – Differential Pair* from the pop-up menu.
2. In the *Filter* drop-down menu, choose *Diff Pair*.
3. If necessary, enter a string in the filter type-in field to focus the search.
4. Click on the differential pair object and click **>** to move it to the selected column (right).

   The differential pair name along with its member nets or Xnets appears.

5. Use the **<** and **>** keys to move nets in and out of the differential pair selection column as desired.

Tip

A differential pair object consists of a net or Xnet and its mate. You cannot choose a mate that is part of another differential pair object.

6. Click *Modify*.
7. Click *Close* to dismiss the dialog box.

Using differential pair membership auto setup

See Objects – Create – Differential Pair on page 101.

To delete a differential pair object

1. In the *Objects* column, click on a user-defined differential pair object.

   Note: You cannot delete a model-defined differential pair object in Constraint Manager.

2. Click *Delete*.

   A confirmer appears.

3. Click *Yes*.

   Constraint Manager destroys the differential pair object and preserves the member nets and Xnets.

Renaming a differential pair object

1. In the *Objects* column, click on a differential pair object.
2. Do one of the following:

   - Choose *Objects – Rename*.

   - or -

   - Right-click and choose *Rename* from the pop-up menu.

3. Enter a new name.
4. Click *OK*.

Analyzing a differential pair object

1. Choose *Analyze – Analysis Modes*.
2. Enable “All differential pair checks.”
3. In the *Objects* column. click on a differential pair object.
4. Choose *Analyze – Analyze*.

Extracting a differential pair object into SigXplorer

1. In the *Objects* column. click on a differential pair object.
2. Choose *Tools – SigXplorer*.

   If the differential pair object is model-defined, both net or Xnet members of the differential pair are extracted. If the differential pair object is user-defined, only one member of the differential pair object is extracted.

Objects – Membership – Ratsnest Bundle

Use this command to examine, or redefine, the pin pair members of a ratsnest bundle. A pin pair cannot be a member of more than one ratsnest bundle.

Tip

Refer to the *Allegro® Constraint Manager User Guide* for more information on Ratsnest Bundle objects.

Procedures

Examining membership of a ratsnest bundle

1. In the *Objects* column of the *Ratsnest Bundle Properties* worksheet (in the *Properties* domain), click on a member pin pair of the expanded ratsnest bundle.

   The *Ratsnest Bundle* dialog box appears.

2. Click *Current Members*.

   The *Current Members* dialog box appears showing the member pin pairs of the parent bundle.

3. Click *Close*.

Changing membership of a ratsnest bundle

1. In the *Objects* column of the *Ratsnest Bundle Properties* worksheet (in the *Properties* domain), click on a member pin pair of the expanded ratsnest bundle.

   The *Ratsnest Bundle* dialog box appears.

2. From the drop-down menu, click to select from the available bundles in the design.

3. Click *OK*.

   Constraint Manager reassigns the pin pair to the bundle that you selected.

Objects – Remove

Use this command to remove selected nets, Xnets, or pin pairs from a bus or a *Match Group*.

Procedure

Removing a member

1. Select and expand a bus or *Match Group*.
2. Select a net, Xnet, or pin pair member.
3. Do one of the following:
   - Choose *Objects – Remove*.

     - or -

   - Right-click and choose *Remove* from the pop-up menu.

Objects – Rename

Use this command to rename a bus, *Match Group*, differential pair, or Xnet.

Procedure

Renaming a bus, Match Group, differential pair, or Xnet

1. In the *Objects* column, select a bus, *Match Group*, differential pair, or Xnet.
2. Do one of the following:
   - Choose *Objects – Rename*.

     - or -

   - Right-click and choose *Rename* from the pop-up menu.

   Note: By default, the Xnet inherits the name of the member net whose name is the lowest in the alphabet.

Objects – Delete

Use this command to delete a bus, *Match Group*, user-defined differential pair, pin pair, or Electrical CSet.

Procedures

Deleting a bus, Match Group, differential pair, or pin pair

1. In the *Objects* column, select a bus, *Match Group*, user-defined differential pair, or pin pair.
2. Do one of the following:
   - ❑ Choose *Objects – Delete*.

     - or -

   - ❑ Right-click and choose *Delete* from the pop-up menu.

   Constraint Manager destroys the object and leaves the members intact.

Deleting an Electrical CSet

1. In the *Objects* column in the *Electrical CSet* folder, select an Electrical CSet.
2. Do one of the following:
   - ❑ Choose *Objects – Delete*.

     - or -

   - ❑ Right-click and choose *Delete* from the pop-up menu.
3. If the Electrical CSet is referenced by objects in the design, then acknowledge the confirmer message.

   Constraint Manager removes the Electrical CSet.

Objects – Constraint Set References

Use this command to associate a Constraint Set (CSet) with a net object.

Mapping CSets to Net-related Objects

You can instruct Constraint Manager to make an association between a CSet and a net object. This binding is the conduit that transfers design intent (constraints) to that net object. With *Physical*- and *Spacing* CSets, the association is layer-based. With *Electrical* CSets, there are caveats.

Mapping Electrical CSets

Constraint Manager intelligently maps the constraint, pin pair, and scheduling information, imported from a topology template or defined in an *Electrical CSet*, to a candidate Xnet that matches the topological characteristics of the referenced *Electrical CSet*. If the candidate Xnet contains the same number of pins but does not match all of the topological characteristics of the *Electrical CSet*, Constraint Manager maps the constraints that it can and renders the *Referenced Electrical CSet* column red. Examine the *Electrical CSet Apply* report to aid you in resolving conflicts.

The Mapping Process

Topology mapping ensures that you can apply the topology template (Electrical CSet) to a class of Xnets which can accept the desired schedule and pin pair constraints. Constraint Manager makes several passes in determining topological mapping criterion, with each pass being less restrictive.

One of the more common failures occurs when The candidate Xnet does not match the number of pins in the topology or the *Electrical CSet*. Constraint Manager still makes the association between the candidate net and the referenced *Electrical CSet*, but does not transfer any constraint, pin pair, or scheduling information.

With the introduction of *Optional Pins* in SigXplorer, this is now less of an issue.

Tip

You can influence each pass by specifying the following. See

- Mapping Modes on page 130
- Optional Pins in the *SigXplorer Command Reference*

Constraint Manager examines the following entities as it makes each pass:

Entity	Description	Example
Refdes	Component Identifier	`U1, U3, U5-7`
Pin Number	Pin of Component	`U1.3, U3.2`
Device Value	Value of Discretes	`10 Ohms, 22 Pf`
Diff Pair Type	Inverting, Non-inverting	`CH1_1553+, CH1_1553-`
Buffer Model	Model assigned to device	`CDSDefaultInput,MyBuffer`
Pinuse	Buffer Type	`Input, Output, Bi, Unspec, Power, Ground`
Pinuse (relaxed)	Maps an I/O- or connector-pin to any driver or receiver.	

Pin Mapping Passes

Constraint Manager makes up to seven passes in determining pin mapping.

Pass 1	Pass 2	Pass 3	Pass 4
1. Refdes	1. Refdes	1. Device Type	1. Device Type
2. Pin Number	2. Device Type	2. Device Value	2. Device Value
3. Device Type	3. Device Value	3. Diff Pair Type	3. Diff Pair Type
4. Device Value	4. Diff Pair Type	4. Pinuse	4. Pinuse
5. Diff Pair Type	5. Pinuse	5. Buffer Model	
6. Pinuse			

Pass 5	Pass 6	Pass 7
1. Refdes 2. Pin Number 3. Diff Pair Type	1. Refdes 2. Diff Pair Type	1. Pinuse (less restrictive) In the less-restrictive pinuse pass, any I/O- or connector-connector pin maps to any driver or receiver pin. 2. Diff Pair Type

Mapping Modes

Although not usually required, you can change the mapping modes to guide the mapping process.

When you set the Mapping Mode to . . .	. . .Constraint Manager maps pins by this precedence . . .	. . . and makes these passes See Pin Mapping Passes on page 129.
■ *Pinuse*	1. Pinuse and Buffer Model	3, 4, 7
■ *Refdes*	1. Refdes and Pin Number 2. Pinuse and Refdes	1, 2, 5, 6
■ *Pinuse and Refdes*	1. Refdes and Pin Number 2. Pinuse and Refdes 3. Pinuse and Buffer Model 4. Pinuse	1, 2, 3, 4, 5, 6, 7

You specify the mapping mode in either SigXplorer (choose *Set – Constraints* and click the *Wiring* tab) or in Constraint Manager (in the *Wiring* worksheet of the *Routing* workbook at the *Electrical CSet* folder level). The mapping mode is stored with the topology file.

With the mapping mode set to its default (Pinuse and Refdes), all mapping possibilities are considered.

Illegal Topologies

The following results in a mismatch:

- A trace or via element in the topology; only T-lines can connect components. You can choose *Edit – Transform – for Constraint Manager* before choosing *File – Update Constraint Manager* from SigXplorer to convert the trace to an ideal transmission line.
- A terminator connected to multiple pins on a component.
- More than one terminator on a node.
- More than one component pin connected to a node; T-lines must separate component pins.
- A voltage source that is not connected to a discrete component.
- Any disconnected components.
- Sweepable ranges on discrete components; only a single value is allowed.
- A T-point connected to fewer than three T-lines; Every T-line end must connect to a T-point or a pin.
- Pins in each net (in the Xnet) do not match with the pins in each net in the topology.

Successful Mapping

If the mapping is successful, the net object inherits constraints from the Electrical CSet as follows:

- Pin Pairs with switch/settle constraints appear as children of the Xnet or Net in the *Switch/Settle Delays* worksheet of the *Timing* workbook.
- Pin Pairs with propagation constraints appear as children of the Xnet or Net in the *Min/Max Propagation Delays* worksheet of the *Routing* workbook.
- Pin Pairs with impedance constraints appear as children of the Xnet or Net in the *Impedance* worksheet of the *Routing* workbook.
- *Match Group*s appear in the *Relative Propagation Delays* worksheet of the *Routing* workbook.
- The schedule constraint appears in the *Wiring* worksheet of the *Routing* workbook.
- The net object automatically inherits all non topology-specific electrical constraints.

Procedures

Assigning (or reassigning) a Constraint Set to a Net Object

1. In any Net-based worksheet, select a net, Xnet, *Match Group*, differential pair, pin pair, or class object, then do one of the following:

   - In the *Objects* column, choose *Objects – Constraint Set References*.

     - or -

   - In the *Objects* column, right-click and choose *Constraint Set References* from the pop-up menu.

     - or -

   - In the *Referenced Constraint Set* column, choose an CSet from the drop-down menu.

2. For *Physical-* and *Spacing* CSets, use the left and right arrows to populate the *References* column.

   Note: Choosing a Constraint Set from the *Referenced Electrical CSet* column lets you bypass this dialog box.

3. From the drop-down menu, choose a Constraint Set from the list.

4. Click *OK*.

Tip

You can assign the same CSet to multiple objects; you can associate each object to only one CSet.

Removing a Constraint Set from an object

1. In the *Objects* column of any *Net* worksheet, select a net, Xnet, *Match Group*, differential pair, pin pair, or class object.

2. Do one of the following:

   - Choose *Objects – Constraint Set References*.

     - or -

   - Right-click and choose *Constraint Set References* from the pop-up menu.

3. From the drop-down menu, choose *None* from the list.
4. Click *OK*.

Tip

A quick way to de-reference a CSet from an object is to select the object, then right-click and choose *Clear* from the *Referenced CSet* pop-up menu.

Objects – Report

Use this command to specify report settings for objects and constraints. The Analysis Settings and Analysis Modes dialog boxes can also be accessed through this command.

Procedures

Report Dialog Box

Use this field . . .	To . . .
Select Worksheets	Specify which worksheets to include in the report. This field is hierarchical, starting at the Electrical CSet or *Net* folder, down to the workbook, down to the individual worksheet. Only checked worksheets appear in the report.
Selected Objects	Filter based on Electrical CSet, Bus, Diff Pair, Xnet, or Net from the pull-down menu.
Filter (left column)	List all available objects in the design. Use the *selected objects* filter to limit the list. Use wildcard characters to further limit the list. Use the right arrow to move objects to the right column for inclusion in the report.
Filter (right-column)	List objects included the report. Use the *selected objects* filter to limit the list. Use wildcard characters to further limit the list. Use the left arrow to move objects to the left column for exclusion from the report.

Use this field . . .	To . . .
Analysis Status	Select one of the following options from the drop-down menu: ■ *Any* Reports objects that pass analysis with no constraint violations, objects that pass analysis with constraint violations, and objects that fail to analyze. ■ *Violations Only* Reports only objects that pass analysis with constraint violations. ■ *Failures Only* Reports only objects that fail to analyze.
Run Analysis (checkbox)	Run analysis on the objects selected for the report. *Actual* and *Margin* values are not calculated until the object is analyzed.
Constrained Objects Only (checkbox)	Limits the report to objects that have an associated Electrical CSet or an override.
Settings	Set simulator preferences. See *Analyze – Settings* for more information.
Analysis Modes	Set design rule checks. See *Analyze – Analysis Modes* for more information.

Procedure

Generating a report

1. Choose *Objects – Report*.

   The *Report* dialog box appears.

2. Focus the report using filters as follows:
   - Select Worksheet
   - Selected Objects
   - Wildcard queries
   - Analysis Status
   - Constrained Objects Only
3. Optionally, specify simulator settings and design rule checks, then analyze.
4. Click *OK*.

   Constraint Manager displays the report online and saves it to the `consmgr.rpt` file.

Column Menu Commands

Column – Analyze

Procedures

Use this command to compute results for the selected column. Constraint Manager analyzes each cell in the column and returns a value called the *actual*. Constraint Manager then compares the *actual* value with the *constraint* value, that you set, to compute the *margin*.

See Constraint Analysis in the *Allegro® Constraint Manager User Guide* for more information.

Constraint Manager:

- Uses color to display analysis results. The default colors are:
  - Green

    when the actual returned is within the limits of the constraint
  - Red

    when the actual returned exceeds the limits of the constraint
  - Yellow

    when analysis cannot be performed. You should position the cursor over the *actual* cell and examine the status window for analysis errors.
- Rolls up the worst-case margin to a higher level object. This lets you to view a collapsed worksheet and immediately realize where violations exist.
  - Pin Pair results roll up to the Xnet or Net.
  - Xnet/Net results roll up to their parent Bus, Electrical Diff Pair, or Relative/Match Group.

- Invalidates (removes values) analysis results whenever you:
  - Move components or route nets in the board layout
  - Change a constraint value
  - Change analysis settings
  - Exit and re-enter Constraint Manager

Tip

When Constraint Manager invalidates analysis results, it removes values from the *actual* and *margin* cells. Although the values are removed, certain cells retain their pass/fail color status. To re-populate the worksheets with values, you have to resimulate (choose *Analyze – Analyze* or *Column – Analyze*).

Procedure

Analyzing a column

1. In the worksheet selector, click the desired workbook.
2. Click the tab of the desired worksheet.
3. Click the column heading that you want to analyze.
4. Comply with the steps in the Analysis Checklist.
5. Do one of the following:
   - Choose *Column – Analyze*.

     - or -

   - Right-click and choose *Analyze* from the pop-up menu.

   Constraint Manager computes the *Actual* values and compares them to the set constraints to derive the *Margins*.

Column – Sort

Use this command to order (ascending or descending) a column's values.

Tip

Constraint Manager compares objects at each level in the hierarchy and sorts them independently.

Procedure

1. Click in a column header of the active worksheet.
2. Do one of the following:
   - ❑ Choose *Column – Sort*

     - or -

   - ❑ Right-click and choose *Sort* from the pop-up menu.

   Constraint Manager orders the values in all columns of the active worksheet opposite (ascending or descending) the ordering before you issued the command.

View Menu Commands

View – Options

Use this command to control many of the user interface elements.

View Options Dialog Box

Use this field	To
Main Window	Increase the available screen area for viewing the worksheets. You can hide the tool bar, status bar, worksheet selector, and color legend. Alternatively, you can un-dock, and reposition, the color legend. *Important* The status bar provides key feedback on many operations. We recommend that you do not hide it.

Use this field	To
Synchronize Rows	Synchronize the object display across all rows in all worksheets. When enabled, *Synchronize Rows* recognizes the row modifications that you make in the *active* worksheet — including filters, scroll position, and object expansion — and synchronizes all worksheets that you access with these same display settings. Constraint Manager aligns rows across open worksheets based on the object hierarchy of the active worksheet. If the hierarchy differs among worksheets, Constraint Manager aligns rows across all worksheets as close as possible to the object hierarchy of the active worksheet. **Note:** Depending on the size of the design, Constraint Manager may defer row synchronization until you click a new worksheet to make it active. Constraint Manager preserves the state of *Synchronize Rows* (ON by default) when you exit the tool.  *Tip* When *Synchronize Rows* is *ON* globally, you can disable row synchronization on an individual worksheet by clicking in the *Objects* column label, then right-clicking and choosing *Stay Synchronized* (uncheck the option) from the pop-up menu.
Tool Tips	Show tool tips.
Row Numbers	Show row numbers.
Object Type Dividers	Show a thick horizontal line that separates object types. You can specify a color from the Dividers drop-down menu, located in the Colors section of this dialog box.
Cell Selection Highlights	Show a background tint applied to the row and column header of the selected cell.
Application Select	Control the auto-scrolling feature in worksheets. When selected (the default setting), all selections in the application from which Constraint Manager was launched will automatically scroll and select that object in Constraint Manager's open worksheets. If disabled, the object is not selected and the worksheet does not scroll automatically.

Use this field	To
Single-Click Editing	Control the manner with which you edit a cell's contents. If checked (the default mode), when you click in a cell, the insertion point appears at the location within the cell's contents where you clicked. This makes it easier to edit one or more digits within the cell. If unchecked, when you click in a cell, all digits within the cell are highlighted. This makes it easier to replace the entire contents of the cell, the same way Microsoft® Office Excel works. You can also double-click a cell in unchecked mode to edit one or more digits within the cell.
Show Values Inherited from Design	Clear this checkbox to stop showing values inherited from the *Design* level but not from other structural parents (Classes, Buses, etc.).
Font	Specify a font family for text in workbooks and worksheets.
Restore Defaults	Use Constraint Manager's default settings on start up. Constraint Manager remembers the last state of your work session when you exit the tool. The next time you invoke Constraint Manager, everything appears in the state in which you left it, including window size and position, worksheet display, column size and visibility, analysis color settings, tool option settings, filter settings, and much, much more.
Names	Specify net display in physical or logical (canonical) format.
Pass	Specify the cell color for resulting analysis values (actuals) that meet the specified constraint setting.
Fail	Specify the cell color for resulting analysis values (actuals) that do *not* meet the specified constraint setting.

Use this field	To
Analysis error	Specify the cell color when analysis can not be performed. The *status* window displays the cause of the error. Possible failures may result when trying to analyze with: ■ *only* Design Entry HDL and Constraint Manager running. A PCB editor or APD must be running to access the analysis engine. ■ unplaced components ■ unrouted nets (on stub length and impedance checks) ■ incorrect (or missing) signal models ■ missing trace information ■ inactive DRC mode
Directly set	Specify the cell color of an Electrical CSet assignment or a net-related constraint override set on an object
Dividers	Specify the color of dividers that separate object groupings. This is used with the Object Type Dividers checkbox, located in the *Workbooks* section of this dialog box.
Cell Selections	Specify the color of cell selections. Use this with the Cell Selection Highlights checkbox, located in the Workbooks section of this dialog box.
Reset to defaults	Restore the default cell color scheme: ■ Pass = green ■ Fail = red ■ Analysis error = yellow ■ Directly set = blue If you have modified color options (pass, fail, analysis error, directly set), these settings will be recalled when you uncheck *use defaults*.

Use this field	To
Use colors	Turn off cell colors. At times, it may be easier on your eyes to examine cells without colors. Simply uncheck the *use colors* checkbox to turn off cell colors.
Color Palette	Customize colors or use Constraint Manager's defaults.
Use Tabbed View	Enable tabbed selection of participating designs in a system.  *Tip* Once visible, you can right-click on a tab to change its orientation or to synchronize tabs across worksheets.
Synchronize Tabs	Synchronize all open workbooks associated with the design represented by the active tab.
Tabs on . . .	Specify where (top, left, bottom, right) to display tabs in Constraint Manager.

View – Hide Column

Use this command to collapse a selected worksheet column to hide its contents.

See also: *View – Show All Columns*.

Procedure

Hiding a column

1. Click the head of the column that you want to hide.
2. Do one of the following:
   - Choose *View – Hide Column*.

     -or-
   - Right-click and choose *Hide Column* from the pop-up menu.

     -or-
   - Drag the right border of the column to the left until the column is hidden.

View – Show All Columns

Use this command to expand all hidden worksheet columns.

See also: *View – Hide Column*.

Tip

To show only a single hidden column, double-click the divider at the location of the hidden column. The cursor changes slightly when moved over the divider of a hidden column.

View – Expand All Rows

Use this command to expand all hidden (collapsed) worksheet rows in the active worksheet.

See also: *View – Collapse All Rows*.

Expanding all rows

1. Click in any row of the worksheet that you want to expand.
2. Choose *View – Expand All Rows*.

   Constraint Manager expands all rows in the active worksheet.

View – Collapse All Rows

Use this command to hide (collapse) all hidden worksheet rows in the active worksheet.

See also: View – Expand All Rows.

Collapsing all rows

1. Click in any row of the worksheet that you want to hide.
2. Choose *View – Collapse All Rows*.

   Constraint Manager collapses all rows in the active worksheet.

View – Refresh

Use this command to reload (redraw) the current worksheet in focus.

View – Always on Top

Use this command to keep Constraint Manager in the forefront of all open applications. This command is not available in Constraint Manager when launched from Unix or Linux.

Audit Menu Commands

Audit – Constraints

Procedure | Example

Use this command to generate a report of net-level overrides and constraint inconsistencies.

Tip

A net inherits the default values of its assigned Electrical CSet. A net-level override lets you specify a different constraint value, on a case-by-case basis, while maintaining the default value defined in the Electrical CSet.

The audit includes the following checks:

- Min values that exceed Max values
- Values less than zero
- Completeness violations
- Group membership violations
- Relative group violations
- Paired parallelism lengths and gap
- Setup and hold relative to clock period
- Net-related overrides
- Diff pair violations when one member inherits a constraint that is different than the same constraint on its mate

Procedure

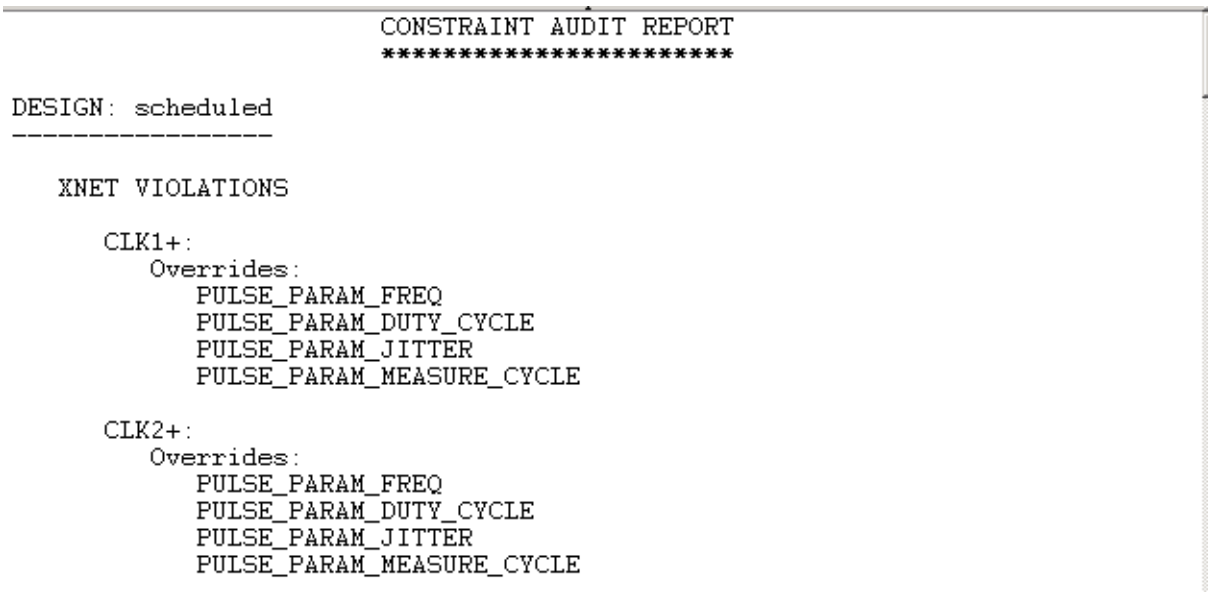
1. Choose *Audit – Constraints*.

   The Audit Constraints dialog box appears.

2. Specify a directory and a file name, or accept the default: *constraintsaudit*`.rpt`.

3. Click *Save*.

Example



```
                        CONSTRAINT AUDIT REPORT
                        ***********************

DESIGN: scheduled
-----------------

   XNET VIOLATIONS

      CLK1+:
         Overrides:
            PULSE_PARAM_FREQ
            PULSE_PARAM_DUTY_CYCLE
            PULSE_PARAM_JITTER
            PULSE_PARAM_MEASURE_CYCLE

      CLK2+:
         Overrides:
            PULSE_PARAM_FREQ
            PULSE_PARAM_DUTY_CYCLE
            PULSE_PARAM_JITTER
            PULSE_PARAM_MEASURE_CYCLE
```

Audit – SI Setup

Use this command to run the SigNoise Setup report.

Audit – Obsolete Objects

Dialog Box | Procedure | Example

Use this command to generate a report of objects that must be reconciled between Constraint Manager and the PCB-, package-, or schematic-databases. Constraint Manager displays a *No Obsolete Objects* message as appropriate.

For example, if you use Constraint Manager to constrain an object in Design Entry HDL, that object will be stored in Design Entry HDL's constraint view of the HDL library. If you later delete that object in Design Entry HDL, that constraint will still be in Constraint Manager until it is reconciled with the obsolete objects audit.

This command is used subsequent to importing a dictionary and constraint file (*File – Import – Constraints*) or when the connectivity is disjoint between the component or net in Design Entry HDL and the corresponding Constraint Manager object.

Note: The *Audit – Obsolete Objects* command is *not* available when running Constraint Manager in stand-alone mode.

Audit Obsolete Objects Dialog Box

Use this field	To
Type	Filter on an object type (bus, net, Xnet)
Obsolete Objects	List all objects that no longer exist in the Allegro or Design Entry HDL database, yet exist in Constraint Manager
Existing Objects	List all objects that exist in the PCB-, package-, or schematic-database, and in Constraint Manager
Delete	Remove objects, listed in the *obsolete objects* list, from the PCB-, package-, or schematic-database.
Merge	Assign all properties and constraints from the object selected in the *obsolete object* list to the object selected in the *existing object* list (properties and constraints on the existing object will not be overwritten).

Procedure

1. Choose *Audit – Obsolete Objects*.

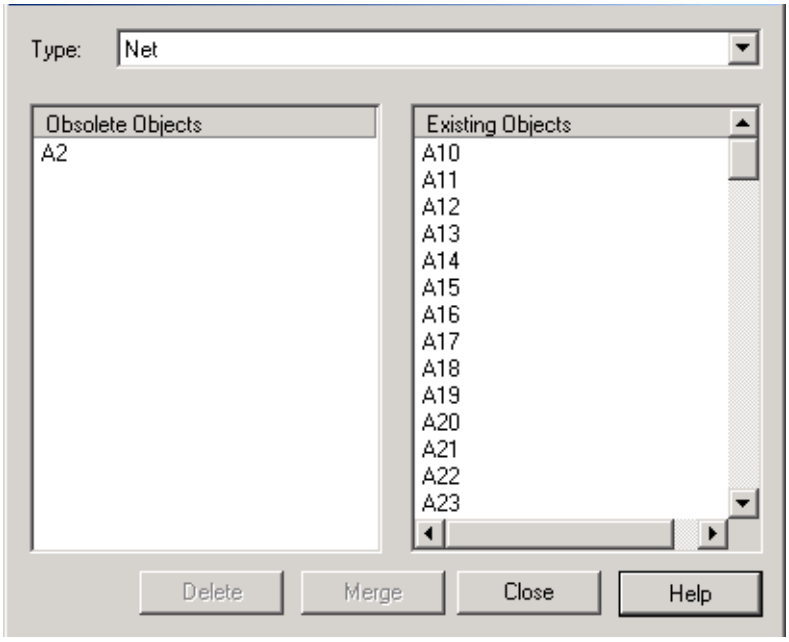
   If there are not any obsolete objects, a confirming message appears.

   If there are obsolete objects, the Audit Obsolete Objects dialog box appears.

2. Click the drop-down menu to filter on an object type (Bus, Net, Xnet).
3. Delete or Merge the obsolete objects.
4. Click *Close*.

Example

This example shows how deleting a member of a bus in the board layout tool, Net A2, affects the remaining bits of the bus as reconciled in Constraint Manager.



Audit – Electrical CSets

Procedures

Use this command to generate a report on the current ECSets in the design and the status of all net-related objects that reference them. The heading of the Electrical CSet audit report summarizes the total number of ECSets and those that contain errors.

The status reports the inheritance for each constraint defined in the Electrical CSet including:

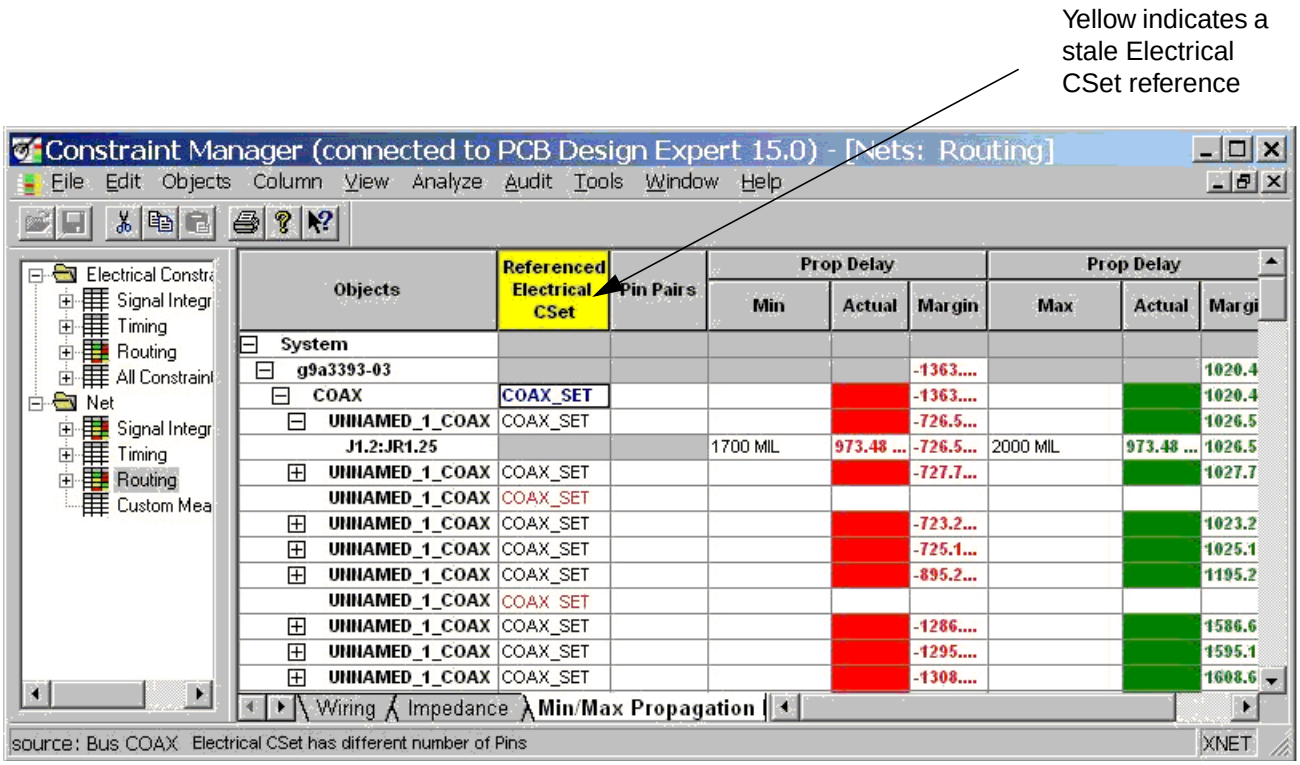
- Any mismatch of the topological characteristics between a net-related object and the Electrical CSet (in which case, the constraint from the Electrical CSet is not inherited but the reference is still made).
- The net-related object that inherits the Electrical CSet constraint.

See Objects – Constraint Set References on page 128 for more information about referencing ECSets and topology mapping criteria.

Apply Status

The *Referenced Electrical CSet* column of any net-related worksheet is colored yellow when:

- Constraint Manager is running with Design Entry HDL
- Automatic topology update is turned off in Constraint Manager (choose *Tools – Options*).



Once you audit the electrical constraint sets (choose *Audit – Electrical CSets*) the *Referenced Electrical CSet* column label changes from yellow to a neutral color. You can also right-click and choose *Audit Electrical CSet* from the pop-up menu.

Any Electrical CSet apply errors will be reflected in the worksheet by coloring the Electrical CSet name in the *Referenced Electrical Cset* cell red. You can refer to the *Audit* report to determine what the error is or you can re-run the *Audit* of the Net or Xnet that has the error.

Procedures

Auditing all ECSets

1. Choose *Audit – Electrical CSets*.

   The Audit Electrical CSets *Report* dialog box appears.

2. Specify a directory and a file name, or accept the default: *ecsetaudit*`.rpt`.

3. Click *Save*.

4. Examine the audit report.
5. Resolve topology mismatches.

   See Objects – Constraint Set References on page 128 for more information about referencing ECSets and topology mapping criteria.

To audit a single Electrical CSet reference

➤ In the *Referenced Electrical CSet* cell, right-click and choose *Audit Electrical CSet* from the pop-up menu.

The audit report for that Electrical CSet appears.

Audit – Topology Templates

Use this command to migrate deprecated properties to Electrical CSet references used by Constraint Manager.

In pre-14.0 designs, electrical constraints were captured using three entities: the topology template (specified with the TOPOLOGY_TEMPLATE property), the topology assignment (specified with the ASSIGN_TOPOLOGY property), and the constraint set (specified with the ELECTRICAL_CONSTRAINT_SET property).

In post-14.0 designs, only the Electrical CSet association is supported; the TOPOLOGY_TEMPLATE and the ASSIGN_TOPOLOGY properties are no longer required. All topology information is now contained in the Electrical CSet (specified with the ELECTRICAL_CONSTRAINT_SET property).

The topology templates audit removes the TOPOLOGY_TEMPLATE, TOPOLOGY_TEMPLATE_REVISION, and ASSIGN_TOPOLOGY references from net-related objects.

Audit Old Template Dialog Box

Use this field	To
Topology template values (drop-down menu)	List all TOPOLOGY_TEMPLATE and ASSIGN_TOPOLOGY property values in the design. Once a property value is selected from the drop-down menu, all nets which have the same template value will be listed
Update to use Electrical Cset	
Import (radio button)	Import a new template. The field beside the radio button will be populated with a Topology Template (.top file) name if one can be found on disk. Constraint Manager uses the TOPOLOGY_TEMPLATE_PATH environment variable to search for a template file with the same name as the property value
Browse	Find the appropriate template file if Constraint Manager cannot locate it.

Use this field	To
Existing (radio button)	Use an existing Electrical CSet. This option will be the default if the design contains an Electrical CSet with the same name as the property value
Overwrite existing constraints (checkbox)	Controls whether constraint values in the Electrical CSet will overwrite any existing net-related constraints when an Electrical CSet is re-applied.  *Important* Enabling *overwrite existing constraints* is necessary when migrating pre-14.0 designs. This will ensure that all net-related overrides—created by the pre-14.0 topology template mapping software—are removed.
Apply	Migrate all listed nets to reference the imported or existing Electrical CSet. Once migrated, nets will have their TOPOLOGY_TEMPLATE and ASSIGN_TOPOLOGY properties deleted. Constraint Manager displays a *properties up to date* message as appropriate.

Window Menu Commands

Window – New Window

Use this command to duplicate the content of the active worksheet in a new window. This lets you focus your view on different objects, or different columns, in the same worksheet.

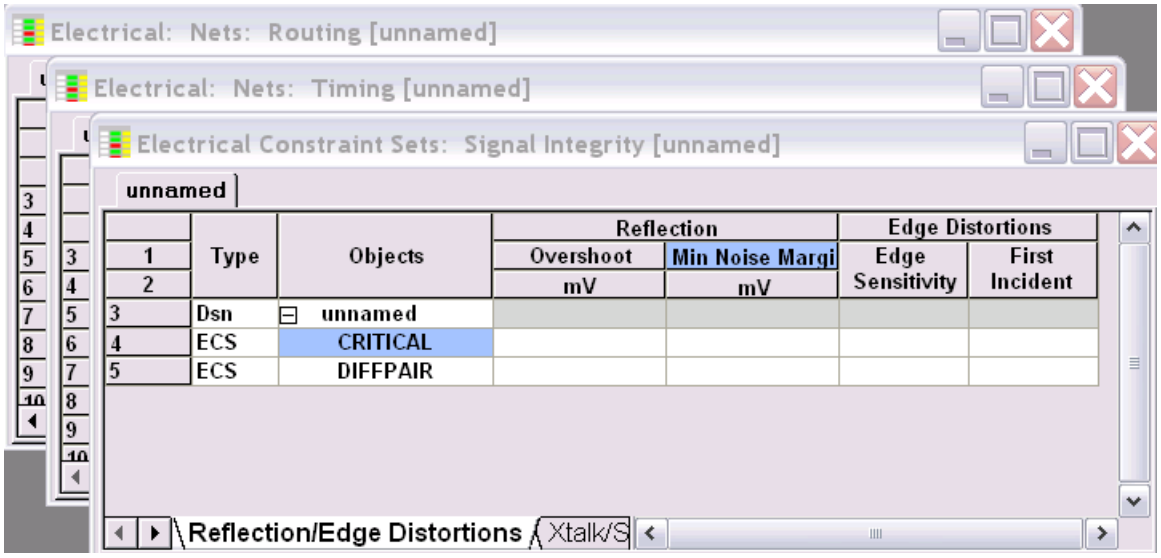
Window – Cascade

Use this command to view all open worksheets, arranged one-behind-the-other. Constraint Manager orders worksheets so that each is selectable with a click of the mouse. The active window occupies the foreground and is identifiable by an active (selected) border.

Procedure

➤ Choose *Window – Cascade*.

All open worksheets reposition, one in front of the other.



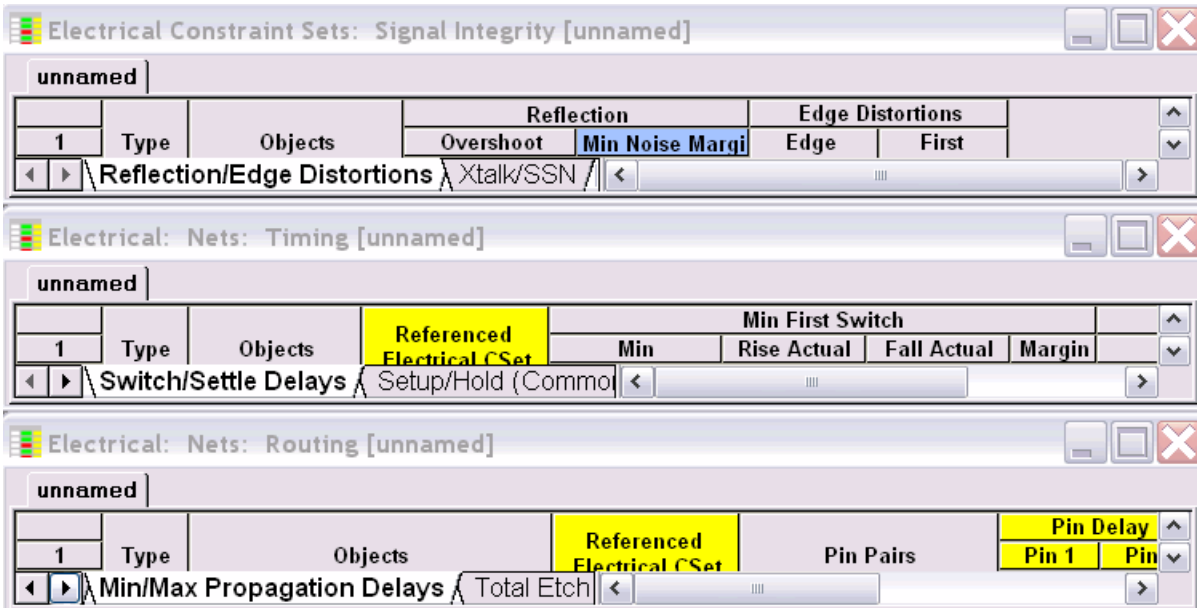
Window – Tile

Use this command to view all open worksheets simultaneously by tiling. Each open worksheet resizes to accommodate the dimensions of the Constraint Manager workspace.

Procedure

➤ Choose *Window – Tile*.

All open worksheets resize, one atop the other.



Window – Arrange Icons

Use this command to arrange *minimized* worksheets at the bottom of the Constraint Manager workspace.

Window – Close All

Use this command to close all open windows in all worksheets.

Window – Next Worksheet Tab

Use this command to cycle forward among worksheets in a workbook.

Window – Previous Worksheet Tab

Use this command to cycle backward among worksheets in a workbook.

Analyze Menu Commands

Analyze – Initialize

Use this command to configure a multi-board (System) design or to manage cases (simulation sessions)

Constraint Manager uses the same simulation environment as your PCB editor or in APD.

Refer to the following topics in the *Allegro® SI User Guide* for more information on:

- Transmission Line Simulation Setup
- Multi-Board Designs

Analyze – Settings

Procedures

Use this command to set up options and preferences for analysis. These settings apply to subsequent analysis sessions.

Important

When you change values in a worksheet, you may have to re-analyze (choose *Analysis – Analysis*) to refresh the analysis results.

Analysis Settings Dialog Box

Driver/Receiver FTS mode	Specifies one or more simulation modes (*Fast*, *Typical*, *Slow*, *Fast*/*Slow*, *Slow*/*Fast*). Determines different buffer and device characteristics to use in simulation circuits. For example, VI Curves and package parasitics. Default is *Typical*. If *FTS* mode settings result in multiple simulations, Constraint Manager displays only the worst-case results.
Reflection	Specifies the type of simulation and measurements to use for getting reflection results. *Pulse* is the default.

Crosstalk	*Aggressor Switch* Specifies the aggressor switching mode (Odd or Even). The default is Odd. *Aggressor* Specifies the number of crosstalk simulations required for generating actuals for the MAX_PEAK_XTALK constraint. When you choose *All Drivers*, crosstalk simulations are run for all Aggressor Drivers; worst case Xtalk is shown in the *Actual* field. For the MAX_XTALK (MAX_INTER/ INTRA_XTALK) constraints, Fastest Driver on Aggressors is used irrespective of this field. *Timing Windows* When selected, specifies that the MAX_XTALK constraint is to be checked against various Timing Group Crosstalk Simulations rather than All Neighbor Crosstalk Simulations. Also specifies that the MAX_PEAK_XTALK constraint shall be checked only against those individual aggressors which are not ignored due to Time Groups. When unselected, the MAX_XTALK constraint is checked against All Neighbor Crosstalk and MAX_PEAK_XTALK constraint is checked against all individual aggressors which fall in specified geometry window and with sufficient coupled length.
Save Waveforms	When selected, specifies that waveforms be saved for every simulation. When unselected, waveforms will not be saved for those simulations.
Save Circuit File	Saves the simulation directory to disk.
Use Custom Stimulus for Custom Measurements	Enables custom stimulus for simulations. You define custom stimulus in SigXplorer.

Preferences

Displays the Analysis Preferences dialog box to set up simulation preferences. Changing any of the case-specific preferences causes the *Case Update* dialog box to display whenever there is simulation data in the current case.

Refer to the following topic in the *Allegro® SI User Guide* for more information.

- Transmission Line Simulation Setup

Procedure

1. Choose *Analyze Settings*.
2. In the Analysis Settings dialog box, specify
   - An FTS mode
   - Reflection and crosstalk simulation parameters
3. Optionally, click *Save Waveforms*.
4. Optionally, click Use *Custom Stimulus*.
5. Optionally, click preferences to set default simulation parameters.
6. Click *OK*.

Analyze – Analysis Modes

Procedures

Use this command to enable individual design rule checks (DRCs), associated options, and custom measurements.

Important

The *Setup – Constraints – Modes* command in PCB Editor, SiP Editor, and APD performs the same function as the *Analyze – Analysis Modes* command in Constraint Manager.

Refer to the Allegro Platform Constraints Reference for detailed information on individual constraints.

Analysis Modes Dialog Box

The *Analysis Modes* dialog box contains the *modes* for the *Electrical, Physical, Spacing, Same Net Spacing, BondWire,* and *Design* rule checks and their options, as well as *Custom Measurements*.

Common Controls

All On	Interactively checks for design rule violations. ■ Design rule violations appear in the appropriate worksheet cell (in red) for all objects that have offending constraints in an assigned CSet ■ Design rule violations also appear in reports (choose *Objects –Report*) ■ A DRC bowtie marker appears in the board layout canvas
All Off	Disables design rule checking to improve system performance.
All Batch	Checks for design rule violations only when a batch command (electrical-only) is performed in the layout tool.

Online DRC

Temporarily disables design rule checking.

- This is useful when you need to make compute-intensive placement or routing modifications and you do not want to hinder system performance.

Tip

The *Setup – Enable Online DRC* command in PCB Editor, SiP Editor, and APD performs the equivalent function.

- When you turn DRC back on (check the On-line DRC checkbox), the constraint status is stale; you must analyze in Constraint Manager or specify a DRC update in the layout tool to refresh the design rule checks.

Electrical, Physical, Spacing, Same Net Spacing, and BondWire Modes tabs

Controls whether a specific design rule check is triggered based on changes to the layout.

Tip

To enable or disable constraint checks by layer in the *Same Net Spacing* domain, use the *Enable DRC by Layer* switch in the Options worksheet. See Same Net Spacing DRC Modes in the *Constraint Manager User Guide*.

Design Options Tab

Specifies plane, soldermask, mechanical hole, and testpoint parameters for enabled *Design Modes* checks.

BondWire Options Tab

Species physical and spacing parameters for enabled *BondWire Modes* checks.

Electrical Options Tab

Click the *Electrical Options* tab to selectively enable the inclusion of etch, pin delay, or via delay in length calculations.

- *DRC Unrouted*

  When *DRC unrouted* is checked—for either *Propagation Delay* or *Relative Propagation Delay* — Constraint Manager performs the rule check using manhattan distances for ratsnest connections. This governs the respective DRC rule check, which you specify by clicking the *DRC Modes* tab.

- *Pin Delay*

  When enabled, includes the delay associated with the interconnect that extends from a component pin to the die pad. This includes *min/max/relative* prop delays and differential pair *phase tolerance* constraint checks. You must also enable the appropriate design rule check, which you specify by clicking the *DRC Modes* tab.

  You can access the *Pin Delay* column in the *Relative Propagation Delay*-, *Propagation Delay*-, or *Differential Pair*-worksheets. The *Pin Delay* column has two fields (*Pin1* and *Pin2*) that contain default values, which are derived from a component library or a board database. You can override the defaults by entering your own values.

  You enter constraints in worksheet cells using a *unit* of either *time* or *length*. You can enter a value in the *Propagation Velocity Factor* field to convert *Pin Delay* values to match the *units* entered in the worksheet cells.

  When you hover your mouse pointer over an *Actual* cell, the status line indicates whether pin delay is included in the result.

- *Z Axis Delay*

  When enabled, includes the delay associated with a via that extends between connecting signal layers. You must also enable the appropriate design rule check, which you specify by clicking the *DRC Modes* tab. Constraint Manager derives the Z axis delay length from the board thickness.

  When enabled, Constraint Manager includes via delay in length columns displayed in the *Relative Propagation Delay*-, *Propagation Delay*-, or *Differential Pair*-worksheets.

  Z Axis Delay calculations use a *unit* of either *time* or *length*. If a constraint that uses Z Axis Delay is given in delay units, the *Propagation Velocity Factor* converts the actual length of the Z Axis Delay to the appropriate delay units.

  When you hover your mouse pointer over an *Actual* cell, the status line indicates whether via delay is included in the result.

- *Same Net Xtalk and Parallelism Checks*

  When enabled, performs DRC calculations for crosstalk and parallelism on themselves. When disabled, crosstalk and parallelism checks are made only between one net to every other net.

- *Resolve Differential Pair properties (overrides) before Constraint Regions*

  Preserves the constraint resolution (precedence) of differential pair overrides. When enabled, differential pair overrides have a higher precedence than constraint regions. This property is automatically applied during uprev if any differential pair has any one of the following properties attached:

  - DIFFP_PRIMARY_GAP
  - DIFFP_NECK_GAP
  - MIN_LINE_WIDTH
  - MIN_NECK_WIDTH

  Note: You may not get the desired result if this option is enabled and differential pair Line and Gap constraints are applied by constraint region. The purpose of this property is to preserve the DRC status of an upreved design. Enabling this option is not recommend for new designs. You should review your constraints and eliminate the need for this option.

Custom Measurements tab

Click the *Custom Measurements* tab to selectively enable custom measurements.

Tip

You define custom measurements in SigXplorer Expert.

Measurements appear as children in the tree structure with the parent object representing the Electrical CSet containing the custom measurements set. Only checked measurements appear in analysis results.

The checkbox adjacent to the parent object also serves as a toggle switch for all measurements in the Electrical CSet: *all on* (when checked) or *all off* (when unchecked).

Tip

By default, custom measurements are not included with an imported *Electrical CSet*. To override this behavior, you must enable the *Update existing or create new Custom Measurement worksheet* option in the File – Import – Electrical CSets command.

Procedures

DRC modes procedures

1. Choose *Analyze – Analysis Modes*.

   The *Analysis Modes* dialog box appears.

2. Click the desired *DRC modes* tab.

3. Click *On* to enable individual design rule checks (or choose *All Batch* in Electrical).

Tip

Use the buttons at the bottom to quickly enable/disable all DRCs in each column (*All On / All Off /All Batch*).

4. Check *On-line DRC*.

5. Click the *Electrical Options* tab if you want to include the distance between unrouted connections, pin delay, or via structures in delay calculations.

6. Click the *Design Options* tab if you want to include plane, soldermask, mechanical hole, and testpoint options.

7. Click *OK*.

   The design rule that you enabled will be compared to constraints that you defined in an Cset and assigned to objects.

Custom Measurements procedures

You define custom measurements in SigXplorer, save them in a topology file, and import them into Constraint Manager (*File – Import – Electrical CSet*). You then assign the Electrical CSet to a net object (*Objects – Electrical CSet References*).

Tip

By default, custom measurements are not included with an imported Electrical CSet. To override this behavior, you must enable the *Update existing or create new Custom Measurement worksheet* option in the File – Import – Electrical CSets command.

1. Choose *Analyze – Analysis Modes*.

   The *Analysis Modes* dialog box appears.

2. Click the *Custom Measurements* tab.
3. Click the checkbox adjacent to the custom measurement that you want included in analysis.
4. Click *OK*
5. Choose *Analyze – Analyze*.

Analyze – Analyze

Procedure

Use this command to analyze the selected object.

Tip

Alternatively, you can choose *Objects – Report* to specify an object to analyze, set analysis filter criterion, specify analysis modes, and specify simulator settings.

Objects in the *Signal Integrity*, *Timing*, and *Custom Measurements* worksheets must be analyzed to calculate the *actual* value. Constraint Manager then compares the *actual* to the set constraint to derive the *margin*.

Tip

Constraint Manager returns analyzed results using a color scheme (green for actuals within the specified constraint limit; red for actuals that exceed the specified constraint limit; yellow for failed analysis).

Procedure

1. In the *Signal Integrity*, *Timing*, or *Custom Measurements* worksheet, click on a Net, Xnet, Bus, or Diff Pair.
2. Optionally, choose *Analyze – Settings* to configure the simulator.
3. Optionally, choose *Analysis – Modes* to specify custom measurements.
4. Choose *Analyze – Analyze*.

Tip

Refer to the analysis checklist in the *Allegro® Constraint Manager User Guide* for more information.

Note: Certain signal integrity and timing simulations yield multiple results (fast, typical, slow, for example). You can also generate multiple results for different measurement locations (at the pin or die pad). Constraint Manager rolls up the worst-case result and displays that value in the *Actual* column. You can right-click over the value in an *Actual* cell and select *Simulations…* to display all simulated results. The result you choose replaces the worst-case *Actual* and recalculates the *Margin*.

Analyze – Show Worst Case

Use this command to show the worst case result of an analysis session that requires multiple simulations — such as *Fast/Typical/Slow*.

Procedure

➤ Choose *Analyze – Show Worst-case*.

Note: Certain signal integrity and timing simulations yield multiple results (fast, typical, slow, for example). You can also generate multiple results for different measurement locations (at the pin or die pad). Constraint Manager rolls up the worst-case result and displays that value in the *Actual* column. You can right-click over the value in an *Actual* cell and select *Simulations…* to display all simulated results. The result you choose replaces the worst-case *Actual* and recalculates the *Margin*.

Tools Menu Commands

Tools – SigXplorer

Procedures

Use this command to launch SigXplorer to explore circuit topologies.

In SigXplorer, you create (or extract from the board layout), simulate, and analyze the circuit topology. You then save the topology as a template (`.top`) file. You can then import the topology template into Constraint Manager as an Electrical CSet.

You can capture the following in a topology template:

- user-defined pin ordering (topology scheduling)
- termination strategy (and location on net)
- electrical constraints
- user-defined properties
- custom measurements
- constrained custom measurements (user-defined constraints)
- custom stimulus

Tip

If you select a bus in Constraint Manager, SigXplorer extracts the first bit of the bus. When you later update the topology in Constraint Manager, the topology changes are promoted to all bits of the bus.

Procedures

Extracting a topology from an existing design

1. In the *Object* column, click the object whose topology you want to extract.

2. Do one of the following:

   - Choose *Tools – SigXplorer*.

     -or-

   - Right-click and choose *SigXplorer* from the pop-up menu.

   SigXplorer launches and displays the circuit topology.

3. In SigXplorer, edit, simulate, and analyze the topology as necessary.
4. Choose *File – Update* to apply the topology changes to the corresponding Electrical CSet in Constraint Manager.

   Constraint Manager refreshes all objects that reference the corresponding Electrical CSet.

Developing a topology for a global rule set

1. In SigXplorer, edit, simulate, and analyze the topology as necessary.
2. Choose *File – Save As*.

   SigXplorer saves the topology in the directory that you specified.

3. In Constraint Manager, choose *File – Import – Electrical CSet*.
4. Select the topology that you just saved.

   Constraint Manager imports the topology as an Electrical CSet, which you can later assign to a net-level object. Constraint Manager stores the Electrical CSet as an object in the Electrical CSet folder.

   The object in Constraint Manager must match the electrical- and topological-characteristics of the topology that you just developed. See "File – Import – Electrical CSets" on page 12 for topology mapping criteria.

Tools – SigWave

Use this command to launch SigWave to view the waveform related with the selected *Actual* value for an object in Constraint Manager. If the selected *Actual* does not contain waveform information, Constraint Manager prompts you to resimulate. Consult the *Allegro® SI SigWave Command Reference* and the *Allegro® SI SigWave User Guide* for information about using SigWave.

Tip

The analysis engine computes a value (*actual*) and compares this to the value specified in the Electrical CSet. The difference between the analysis value and the specified constraint value is the *margin*. Both *actuals* and *margins* are returned to the cells in the appropriate worksheets.

Procedure

1. In the row of the object whose waveform you want to view, click in the *Actual* cell.
2. Do one of the following:
   - Choose *Tools – SigWave*.

     -or-

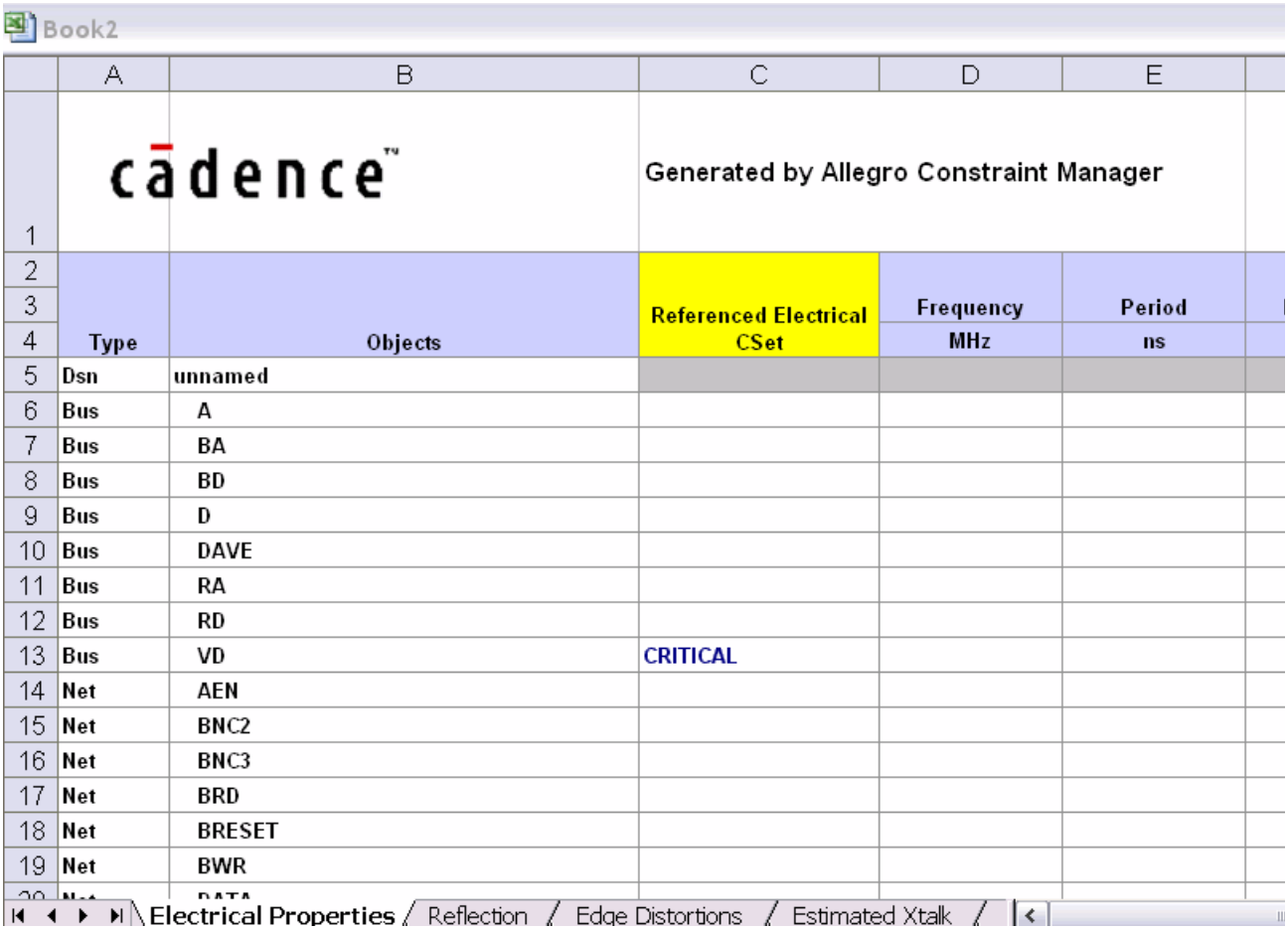
   - Right-click and choose *View Waveform* from the pop-up menu.

   SigWave launches and displays the waveform, which is stored in the *Actual* cell. If Constraint Manager prompts you to resimulate, follow the procedures for the *Analyze – Analyze* command.

Tools – Excel

Use this command to export the active worksheet or workbook to Microsoft® Office Excel on Windows. This command complements Constraint Manager's reporting capability by letting you take advantage of Excel's comprehensive library of data analysis routines. Only worksheet cells—not the *Worksheet Selector*—export to Excel.

Figure 9-3 A Constraint Manager worksheet exported to Excel



Book2

	A	B	C	D	E
1	cadence™		Generated by Allegro Constraint Manager		
2					
3			Referenced Electrical	Frequency	Period
4	Type	Objects	CSet	MHz	ns
5	Dsn	unnamed			
6	Bus	A			
7	Bus	BA			
8	Bus	BD			
9	Bus	D			
10	Bus	DAVE			
11	Bus	RA			
12	Bus	RD			
13	Bus	VD	CRITICAL		
14	Net	AEN			
15	Net	BNC2			
16	Net	BNC3			
17	Net	BRD			
18	Net	BRESET			
19	Net	BWR			

Electrical Properties / Reflection / Edge Distortions / Estimated Xtalk /

Procedure

1. Using Constraint Manager's *Worksheet Selector*, open a workbook and worksheet.
2. Choose *Tools – Excel*.

   The *Data* dialog box appears.

3. Choose *Active Worksheet* or *Active Workbook*.
4. Click *OK*.

   The active worksheet opens in Excel, or the active workbook appears with tabs to choose among worksheets that belong to the active workbook.

   Note: Workbooks may take a few minutes to export. Constraint Manager's *Status Bar* indicates the progression of the task, as each worksheet in the active workbook is processed, as a percentage of completion. To cancel the operation, press Esc.

Tools – Options

Use this command to specify options that govern Electrical CSet extraction and application.

Options Dialog Box

Checkbox Option	Function
Electrical CSet Extraction	
Include Routed Interconnect	Includes traces and vias in the extraction of a net-related object into SigXplorer. This is useful for creating a topology that accurately represents how a net is routed. **Note:** You cannot apply a topology with trace and via models as an Electrical CSet in Constraint Manager. In SigXplorer, you must choose choose *Edit – Transform for Constraint Manager*. *Tip* See Table 1 on page 190 for information about using the *Include Routed Interconnect* and *Schedule Based on Routed Interconnect* options. **Note:** Enabling *Include Routed Interconnect* automatically enables *Schedule Based on Routed Interconnect*.
Schedule Based on Routed Interconnect	SigXplorer derives connections from the trace. SigXplorer derives propagation delay and impedance from traces. Any unrouted segment derives its impedance and propagation velocity from the default settings. *Tip* See Table 1 on page 190 for information about using the *Include Routed Interconnect* and *Schedule Based on Routed Interconnect* options.

Table 1: Interconnect Extraction Scenarios

Include Routed Interconnect	Schedule Based on Routed Interconnect	Circuit Parameters
ON	Forced ON	■ *Schedule* — where routed, based on trace; where unrouted, based on user-defined or default schedule ■ *Propagation delay (Td)* — where routed, based on trace length; where unrouted, based on Manhattan length ■ *Impedance* — where routed, based on trace impedance; where unrouted, based on default impedance ■ *Extraction* — includes traces and vias; must be transformed for Constraint Manager
OFF	OFF	■ *Schedule* — where routed and unrouted, based on user-defined or default schedule ■ *Propagation delay (Td)* — where routed, based on trace length; where unrouted, based on Manhattan length ■ *Impedance* — where routed, based on trace segment with the highest impedance; where unrouted, based on default impedance ■ *Extraction* — includes ideal transmission line (no traces, no vias); can update Constraint Manager directly
OFF	ON	■ *Schedule* — where routed, based on trace; where unrouted, based on user-defined or default schedule ■ *Propagation delay (Td)* — where routed, based on trace length; where unrouted, based on Manhattan length ■ *Impedance* — where routed, based on trace impedance; where unrouted, based on default impedance ■ *Extraction* — includes ideal transmission line (no traces, no vias); can update Constraint Manager directly

Checkbox Option	Function
Electrical CSet Apply	
Rip up etch when mapping topology	Controls whether etch (clines and vias) is removed when an Electrical CSet is reapplied and the schedule of the net changes.
Automatic topology update	Controls how topology-related constraints are reapplied ■ When the design changes (component placement, signal model updates) – or – ■ When an Electrical CSet is initially referenced When *enabled*, changes are applied as the design changes. When *disabled*, changes can be applied by choosing *Tools – Update Topology*. If you change state from disabled to enabled, Constraint Manager presents you with a confirming message stating that it will refresh stale nets and Xnets with updated topology data. **Note:** Disabling automatic topology update may be necessary when design changes are frequent and complex ECSets are referenced.
Overwrite existing constraints	Controls whether constraint values in the Electrical CSet will overwrite any existing pin-pairs (or scheduling) when an Electrical CSet is reapplied. Inherited constraints at the net-level will not be overwritten.  *Important* Enabling *overwrite existing constraints* is necessary when migrating pre14.0 designs using the *Audit – Topology Properties* command. This will ensure that all net-related overrides— created by the pre14.0 topology template mapping software—are removed.

Tools – Update Topology

Use this command to apply a refreshed topology template subsequent to importing a topology with the *File – Import Electrical CSet* command. If *Automatic Topology Update* is enabled (*Tools – Options*), the *Tools – Update Topology* command does not have to be executed.

Procedure

1. Choose *File – Import Electrical CSet*.
2. Use your browser to locate and select a topology (`.top`) file.

   Constraint Manager presents a confirmer window.
3. Click *Yes*.

   Constraint Manager imports the topology file as an Electrical CSet.
4. Choose *Tools – Update Topology*.

   Constraint Manager applies the refreshed topology to any object that references the corresponding Electrical CSet.

Tools – Uprev Topology

Use this command to automatically import topology files which have a higher revision than the existing ECSets in the design.

If the *Automatic Topology Apply* checkbox is enabled (*Tools – Options*), Constraint Manager applies the new information in the ECSets to all objects which reference them.

The initial search for the template uses the directories specified in the argument to the TOPOLOGY_TEMPLATE_PATH environment variable.

Procedure

1. Choose *Tools – Uprev Topology*.
2. Click *Yes* to the confirmer message.
3. Do one of the following:
   - Ensure that the *Automatic Topology Apply* checkbox is enabled (*Tools – Options*)

     -or-
   - Choose *Tools – Update Topology*.

Tools – Update DRC

Use this command to delete all DRC markers in the layout and re-compute DRC in the layout for all constraints that have a DRC mode of *On*. The command adds new DRC markers where errors are detected. The command does not check constraints with DRC mode *Off*.

Tools – Customize Worksheet

Procedures

Use this command to add user-defined or predefined attributes to Constraint Manager's default worksheets, or to create your own customized workbooks and worksheets. You can also use drag-and-drop to reorder columns.

Important

Historically, PCB Editor uses the term *Property*, Constraint Manager uses the term *Constraint*, and Design Entry HDL uses the term *Attribute*. This book uses the term Attribute throughout to describe a Constraint, Property, or Attribute, which is validated or not.

Each attribute that you add to a worksheet requires a new column. Each customized worksheet that you add has an *Objects* column and a *Referenced CSet* column (you can hide the latter). CSet-level customized worksheets do not contain a *Referenced CSet* column.

Important

Certain customized worksheets can contain *Actual* and *Margin* columns. Additionally, you cannot add certain columns, such as *Relative Prop Delay*, *Layer Sets*, and *Ignored Length*.

You add a workbook by selecting the *Object Type* folder in any domain; you add a worksheet by selecting a workbook folder; you add a column by selecting a worksheet. New columns appear to the right of the active worksheet. As with overrides, Constraint Manager renders customized workbooks, worksheets, and columns with a blue tint in the *Worksheet Selector*.

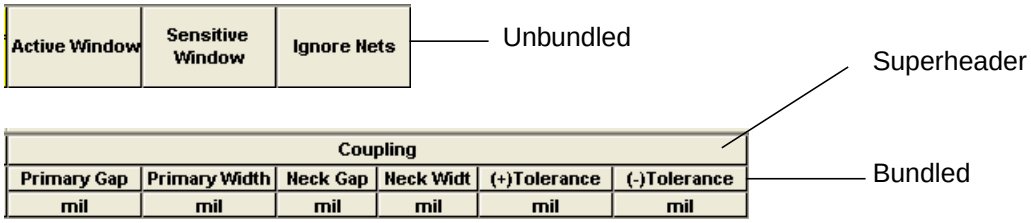
A column superheader icon appears as a rectangle over a circle; a column icon appears as a circle. A predefined column icon is gray; a customized column icon is blue; a hidden column icon is a silhouette of the column icon. See the Customized Worksheets and associated icons figure on page 204 for more information on icons used in Customization mode.

In *Customize* mode (choose *Tools – Customize Worksheet*), Constraint Manager uses special icons in the worksheet selector (see the Customized Worksheets and associated icons figure on page 204).

You use the right mouse button to access most operations in *Worksheet Customization* mode, such as adding, renaming, and deleting workbooks, worksheets, column headers and

columns, and to control their visibility. You can also use drag-and-drop to relocate user-defined columns among worksheets.

Figure 9-4 Column Headers



Procedures

Use the following procedures when working with customized workbooks and worksheets.

Note: You cannot undo actions in *Worksheet Customize* mode, but you can always delete (or hide) any columns that you added.

Adding a customized workbook

1. Choose *Tools – Customize Worksheet* (or right-click and choose *Customize Worksheet* from the pop-up menu) to enable *Worksheet Customization* mode.
2. In the *Worksheet Selector*, click on an *Object Type* folder.
3. Right-click and choose *Add New Workbook* from the pop-up menu.
4. With the workbook name highlighted, enter a unique name for the workbook.

   The workbook appears in the *Worksheet Selector's* tree structure ready for you to add custom worksheets.

Renaming a customized workbook

1. Choose *Tools – Customize Worksheets* (or right-click and choose *Customize Worksheets* from the pop-up menu) to enable *Worksheet Customization* mode.
2. In the *Worksheet Selector*, click the workbook that you want to rename.
3. Right-click and choose *Rename Workbook* from the pop-up menu.
4. With the workbook name highlighted, enter a unique replacement name for the workbook.

Note: You cannot rename a predefined workbook.

Deleting a customized workbook

1. Choose *Tools – Customize Worksheets* (or right-click and choose *Customize Worksheets* from the pop-up menu) to enable *Worksheet Customization* mode.
2. In the *Worksheet Selector*, click the workbook that you want to delete.
3. Right-click and choose *Delete Workbook* from the pop-up menu.

   A confirmation message appears.
4. Acknowledge the message.

   Constraint Manager deletes the workbook; any attributes remain in the dictionary file.

   Note: You cannot delete a predefined workbook.

Adding a customized worksheet

1. Choose *Tools – Customize Worksheets* (or right-click and choose *Customize Worksheets* from the pop-up menu) to enable *Worksheet Customization* mode.
2. In the *Worksheet Selector*, click the customized or predefined workbook to which you want to add a worksheet.
3. Right-click and choose *Add New Worksheet* from the pop-up menu.
4. With the worksheet name highlighted, enter a unique name for the worksheet.

   The worksheet appears in the *Worksheet Selector's* tree structure ready for you to add columns and column headers.

Renaming a customized worksheet

1. Choose *Tools – Customize Worksheets* (or right-click and choose *Customize Worksheets* from the pop-up menu) to enable *Worksheet Customization* mode.
2. In the *Worksheet Selector*, click the worksheet that you want to rename.
3. Right-click and choose *Rename Worksheet* from the pop-up menu.
4. With the workbook name highlighted, enter a unique replacement name for the worksheet.

   Note: You cannot rename a predefined worksheet.

Deleting a customized worksheet

1. Choose *Tools – Customize Worksheets* (or right-click and choose *Customize Worksheets* from the pop-up menu) to enable *Worksheet Customization* mode.
2. In the *Worksheet Selector*, click the worksheet that you want to delete.
3. Right-click and choose *Delete Worksheet* from the pop-up menu.

   A confirmer appears.
4. Click *Yes* to acknowledge the confirmer.

   Constraint Manager deletes the worksheet; any *Attributes* remain in the dictionary file.

   Note: You cannot delete a predefined worksheet.

Adding an attribute to a worksheet

Each attribute that you add to a worksheet requires a new column.

1. Choose *Tools – Customize Worksheets* (or right-click and choose *Customize Worksheets* from the pop-up menu) to enable *Worksheet Customization* mode.
2. In the *Worksheet Selector*, expand the desired workbook.
3. Click the desired worksheet.
4. With the worksheet selected, right-click and choose *Add Column* from the pop-up menu (if you are adding a bundled attribute as part of a group, first choose *Add Column Header* followed by *Add Column*).
5. Ensure that you enable (check) the appropriate *Object Type* for the worksheet.

Tip

If you want to add a column with the same name as an existing column, append a space to the new column name. Internally, this makes the column name unique, although it appears to match the name of an existing column.

6. Select *Predefined* or *User-defined* from the from the drop-down list and follow the appropriate steps:

Adding a predefined or user-defined attribute:

a. Choose a predefined attribute.

Optionally, click *Filter* to focus your selection; or, click *View* to study the attribute's parameters.

 *Tip*

If you intend to add a column from a predefined worksheet to a new worksheet, and you do not know what to choose for a default attribute type, right-click in the column of the predefined worksheet and choose *Change*. Then note the internal attribute name that appears in the pop-up menu and follow the preceding steps. You can click *View* to study the details of the selected, predefined attribute before adding it to a column.

b. Optionally, check *Create Actual/Margin* bundle if you want to also add an *Actual* and a *Margin* column. You can also add a column header name as described in Step 4.

This option is not available for all attributes.

c. Click *OK*.

Adding a user-defined attribute:

a. Click *Create*.

The *Create Attribute Definition* dialog box appears.

b. In the Name field, specify a unique name for the new attribute.

c. In the *Data Type* field, specify an attribute's data type.

All data types, except *string,* include a starting and ending field for specifying a range of values.

d. In the *Treat As* drop-down menu, select one of the following:

- For a user-defined property, choose *Property.*
- For a user-defined constraint, choose *Min Constraint, Max Constraint, Target +/- Tolerance, or Target Tolerances.*

- For an attribute that will display a measurement result, choose *Actual*. You then click the link button to associate a measurement with the *Actual*.

Tip

See Customizing Design Rule Checks in the *Constraint Manager User Guide* for more information on the *Treat As* menu options.

e. In the *Objects* field, enable one or more of the following checkboxes:

- Design
- Part Instance
- Gate Instance
- Pin
- Xnet
- Net
- CSet
- Diff Pair
- Bus
- Pin Pair
- Match Group
- Design Instance
- Net Class

Tip

The choices available in the *Create Attribute Definition* dialog box vary depending on what tool you used to launch Constraint Manager and which objects exist in your design.

f. In the *Range* field, enter start- and end-points.

g. In the *Description* field, enter a description for the new attribute.

Note: If you hover your cursor over a column header, the value that you enter in the *Description* field appears in the *status bar*.

h. In the *Transfer To/From Physical* field, enable this checkbox to pass the attribute to PCB Editor (via netrev), or to Allegro® System Architect or Design Entry HDL (via genfeedformat). The *Part Instance, Gate Instance,* and *Pin* options are available only in Constraint Manager, when launched from Allegro® System Architect or Design Entry HDL.

Attributes are flow-enabled in Constraint Manager, when launched from Design Entry HDL or PCB Editor.

i. Click *OK*.

The *Column Add* dialog box appears with the column name and the attribute name that you just defined. You can change the name if desired.

j. Click *OK*.

Constraint Manager adds the new column to the far-right of the worksheet. In the *Worksheet Selector*, the column name precedes the attribute name (the latter delimited by parentheses).

Tip

You can change an attribute's property by selecting it in the *Worksheet Selector*, right-clicking, and choosing *Properties* from the pop-up menu.

Moving or copying a column

Use drag-and-drop to reposition individual columns within a column superheader, reposition a column superheader (and its member columns) within a predefined or a user-defined worksheet, or move (or copy) individual or bundled columns to a predefined or a user-defined worksheet.

You can drag-and-drop a column from a predefined worksheet to a user-defined worksheet, at which point the column becomes user-defined, and you can rename it. You can also drag-and-drop a column from a user-defined worksheet to a predefined worksheet. Constraint Manager always places newly added columns to the right of all predefined columns.

Important

A column's header (or superheader) must be unique when moving it among worksheets. Constraint Manager prohibits you from dragging a column across worksheets if that column already exists in the target worksheet. Furthermore, the attribute associated with the column must also be unique.

Repositioning a column within the same worksheet

1. In the *Worksheet Selector*, click on a column's header (or superheader).

   If you drag a column's superheader, all columns underneath it are affected.

2. Drag to reposition the column in the current worksheet.

   The columns occupy a different position in the worksheet.

Tip

If you want to drag-and-drop a column with the same name as an existing column, append a space to the new column name. Internally, this makes the column name unique, although it appears to match the name of an existing column. The attribute associated with the column must be unique, however.

Moving or copying a column to a different worksheet

1. In the Worksheet Selector, click on a column's header (or superheader).
2. Drag the column in the current worksheet and drop it in a different worksheet. If you drag a column's superheader, all columns underneath it are affected.

   Note: When dragging a column from a predefined worksheet to a user-defined worksheet, Constraint Manager always performs a copy operation; when moving from a user-defined worksheet to a predefined worksheet, dragging the column results in a move operation, pressing CTRL and dragging results in a copy operation.

   If the target is a predefined worksheet, Constraint Manager adds the column to the end of the worksheet. If the target is a user-defined worksheet, Constraint Manager adds the column where you drag-and-drop it.

Deleting a simple column

1. Choose *Tools – Customize Worksheets* (or right-click and choose *Customize Worksheets* from the pop-up menu) to enable *Customization* mode.
2. In the *Worksheet Selector*, click the column that you want to delete.
3. Right-click and choose *Delete Column* from the pop-up menu.

   - or -

   Drag the column outside of the boundaries of the *Worksheet Selector*.

   A confirmation message appears.

4. Click *Yes* to acknowledge the message.

   Constraint Manager deletes the workbook; any attributes remain in the dictionary file.

   Note: You cannot delete a predefined column.

Deleting a complex column

1. Choose *Tools – Customize Worksheets* (or right-click and choose *Customize Worksheets* from the pop-up menu) to enable Customization mode.
2. In the *Worksheet Selector*, click the column's superheader.
3. Do one of the following:

   Click *Delete*.

   -or-

   Right-click and choose *Delete Column's Header* from the pop-up menu.

   -or-

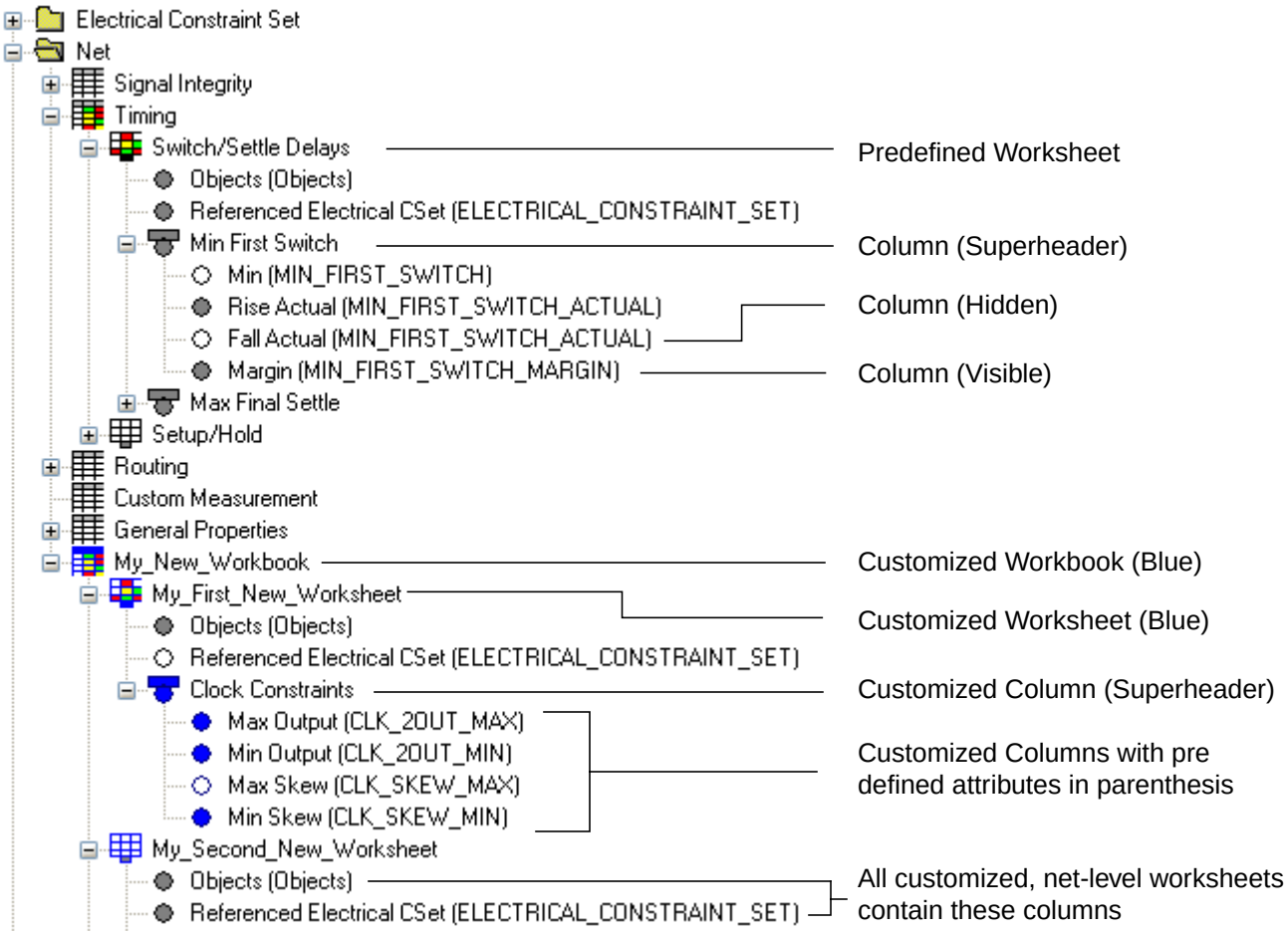
   Drag the column's superheader outside of the boundaries of the *Worksheet Selector*.

   A confirmation message appears.

4. Click *Yes* to acknowledge the message.

   Constraint Manager deletes the column's superheader, individual column headers, and individual columns; any Attributes remain in the dictionary file.

Figure 9-5 Customized Worksheets and associated icons



Tools – Customize Shortcut Keys

Procedures

Use this command to bind (or reassign) a keyboard shortcut key to a command. You can bind one or more keyboard shortcut keys — or key sequences — to each command in Constraint Manager for quick access to, or quick execution of, a command.

Shortcut Keys Dialog Box

Use this field . . .	To . . .
Select a command (list box)	Enable a command to bind to a keyboard shortcut.
Create Shortcut (button)	Invoke the *Assign Shortcut* dialog box, and display keyboard bindings for the selected command (if any exist).
Description (field)	Get a description of the command selected in the *Select a Command* field.
Assigned Shortcuts (list box)	Display keyboard bindings for the selected command (if any exist).
Remove (button)	Remove a selected shortcut (in the *Assign Shortcuts* field), for the highlighted command (in the *Select a Command* field).
OK (Button)	Apply settings and dismiss the *Shortcut Keys* dialog box.
Cancel (button)	Discard settings and retain the *Shortcut Keys* dialog box.
Reset All (Button)	Apply Constraint Manager's default keyboard/ command bindings.

Procedures

Creating a keyboard shortcut

1. Choose *Tools – Customize Shortcut Keys*.

   The *Shortcut Keys* dialog box appears.

2. In the Select a Command field, click on a command.

3. Click the *Create Shortcut* button.

   The *Assign Shortcut* dialog box appears, and shows any bindings that exist for the selected command.

4. Do one of the following:

   - Press a function key.

     -or-

   - Press a letter key.

     To avoid conflicts, when you press a letter key Constraint Manager automatically prepends the *Cntrl* key to create a keyboard combination.

     -or-

   - Simultaneously press the *Alt* key along with a letter key.

   The keyboard key- or key-sequence appears in the *Assigned Shortcuts* field. You can assign more than one shortcut to a command.

5. Click *OK*.

6. Repeat Steps 2 through 5 to bind additional commands, or click *OK* to dismiss the *Shortcut Keys* dialog box.

Unbinding a keyboard shortcut

1. Choose *Tools – Customize Shortcut Keys*.

   The *Shortcut Keys* dialog box appears.

2. In the *Select a Command* field, click on a command.

   Assigned shortcut key bindings appear in the *Assigned Shortcuts* field.

3. Click on a shortcut key binding to select it.
4. Click *Remove*.
5. Repeat Steps 2 through 4 to unbind additional commands, or click *OK* to dismiss the *Shortcut Keys* dialog box.

Restoring default keyboard shortcuts

1. Choose *Tools – Customize Shortcut Keys*.

   The *Shortcut Keys* dialog box appears.
2. Click *Reset All*.

Tools – Customize Toolbars

You can undock a toolbar and reposition it on your work area. You can also hide toolbars individually. If you do not like the default toolbars, you can customize your own, drawing from the pre-defined commands.

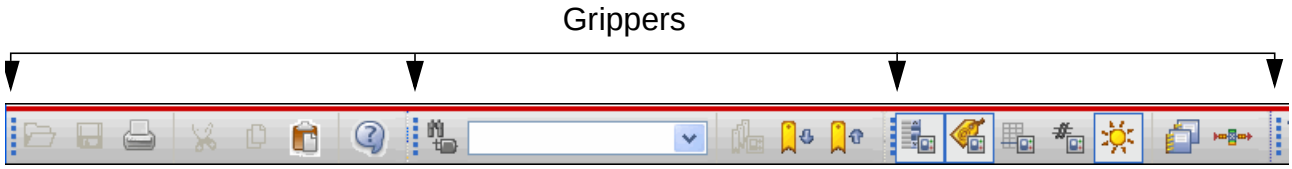
The toolbar ribbon is divided into the following toolbars:

- File
- Edit
- View
- Filter

Procedures

Repositioning a section of the toolbar

➤ Click on a gripper and drag the toolbar to a position that suits your needs.



You can orient the undocked toolbar and have it float or you can position it horizontally or vertically at the perimeter of the worksheets. Reverse this process to dock the back on the toolbar ribbon.

Hiding the toolbar

1. Choose *Tools – Customize Toolbar*.
2. Click the *Toolbars* tab.
3. Uncheck a toolbar that you want to hide.

Adding a custom toolbar

1. Choose *Tools – Customize Toolbar*.
2. Click the *Toolbar* tab.
3. Click *New*.
4. Enter a name for the new toolbar.
5. Click *OK*.

   An empty toolbar appears on your desktop; it will expand as you add command icons.
6. Click the *Commands* tab.
7. Select a category.
8. Drag a command icon to the empty toolbar.
9. Repeat Steps 7 and 8 until the new, custom toolbar is complete.
10. Click *OK*.
11. Dock the new, custom toolbar by grabbing the gripper and dragging it to the toolbar ribbon.

Tip

With the *Toolbars* tab displayed, click *Reset* to set the highlighted toolbar to its default state; choose *View – Options* and click *Reset* to reset custom toolbars.

Help Menu Commands

Help – Tip of the Day

Use this command to learn useful tips on using Constraint Manager. You can cycle through each tip, and you can specify that a different tip appear each time you start Constraint Manager.

Appendix A: Dialog Box Help

This appendix contains dialog box descriptions that are not associated with a menu command.

Edit Via List

Use this command to create a working list of vias for your design. You can specify your own vias or you can choose from those defined in the library or database.

The list represents a selectable order of vias used by certain manual and auto routing commands, such as *Add Connect*. The via order allows an application to automatically select the first via in the list that meets the criteria. For example, if add connection needs to add a via from INT1 to INT2, the first via in the list that meets this criteria is presented as the default via.

Edit Via List Dialog Box

Use this field . . .	To . . .
Select a via from the library or the database	Display all padstacks in the library or database
Filter: Show vias from the library	Display all padstacks in the library
Filter: Show vias from the database	Display all padstacks in the database
Filter via names:	Narrow your view of the list of vias in the library or database. This field accepts regular expression syntax.
Or enter a via name	Add a via that is not in the library or database to the *Via list*
Via List	Create a working list of vias for the design
Remove	Remove the via from the working via list (left pane)
Up	Reorder the vias in the Via List
Down	Reorder the vias in the Via List
Purge	Remove vias from the Via list that are not defined in the library or database. *Yes* removes all undefined vias, in all via lists, in the design. *No* removes all undefined vias, in the via list associated with the currently selected net, in the design

Edit Via List Icons

The Edit Via List dialog box displays icons to represent a via's type. The icon associated with a via is determined by the geometry of the via and its stack up. The exception is the Microvia type, which you must explicitly specify (checkbox) in the *Usage options* section of the Pad Designer. Additionally, Start and End layer names accompany the via's name and icon.

Blind or Buried Via

Through Hole Via

Microvia

Die Pad

Surface Mount Pad

Procedures

Adding a via to the via list from the library or database

1. In one of the *Physical Domain* worksheets, click in the *Vias* cell.

   The *Edit Via List* dialog box appears.

2. Click on a via in the *Select a via from the library* field.

Tip

You can narrow the listing of vias in the library or database by entering the via's name in the *Filter via names* field. This field requires regular expression syntax. As you type, qualifying names appear. You can also SHIFT-click to select a contiguous range of vias or CTRL-click to select non contiguous vias.

The via populates the *Via list* field.

3. If desired, continue to add additional vias to the via list.
4. Click *OK* to save your edits and dismiss the *Edit Via List* dialog box.

Adding a via to the via list that is not in the library or database

1. In one of the *Physical Domain* worksheets, click in the *Vias* cell.

   The *Edit Via List* dialog box appears.
2. In the *Or enter a via name field*, enter the name of the via, preceded by the path name.

   Once you press *Enter*, the via is added to the *Via list* field along with a Warning icon.
3. If desired, continue to add additional vias to the via list.
4. Click *OK* to save your edits and dismiss the *Edit Via List* dialog box.

Reordering the via list

1. In one of the *Physical Domain* worksheets, click in the *Vias* cell.

   The *Edit Via List* dialog box appears.
2. In the *Via list* field, click on a via and click *Up* or *Down* to relocate the via in the list.

Tip

Click *Remove* to remove the selected via from the list. This does not remove the via from the library or database.

To purge non-library vias from the via list

1. In one of the *Physical Domain* worksheets, click in the *Vias* cell.

   The *Edit Via List* dialog box appears.

2. Click *Purge*.

In the confirmation window, choose one of the following options:

- *Yes*

  Purges all vias that you may have added to the via list, using the *Or enter a via name* field, for all via lists in the design.

- *No*

  Purges all vias that you may have added to the via list, using the *Or enter a via name* field, for the via list associated with the currently selected net.

- *Cancel*

  Dismisses the confirmation message (without changes to the via list).

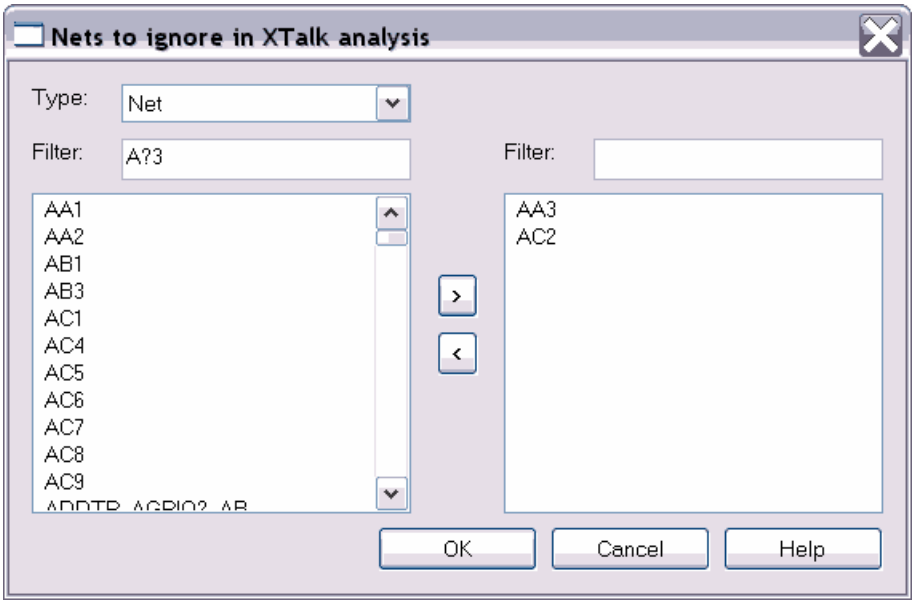
Ignore Nets

Use this command to specify nets to exclude when performing crosstalk analysis.

Procedure

1. In the *Est Xtalk* or *Sim Xtalk* worksheets of the *Signal Integrity* workbook, click in the *Ignore Nets* cell.

   The *Nets to ignore in XTalk analysis* dialog box appears.



2. Choose *Net* from the *Type* drop-down menu.

   Note: You can also choose *Electrical CSet* to select nets to which the CSet is assigned.

3. Click a net in the Left column.

   Note: Use `SHIFT+Click` to select a contiguous range of nets; use `Control+Click` to select non-contiguous nets. Use wildcards to modify the list of nets.

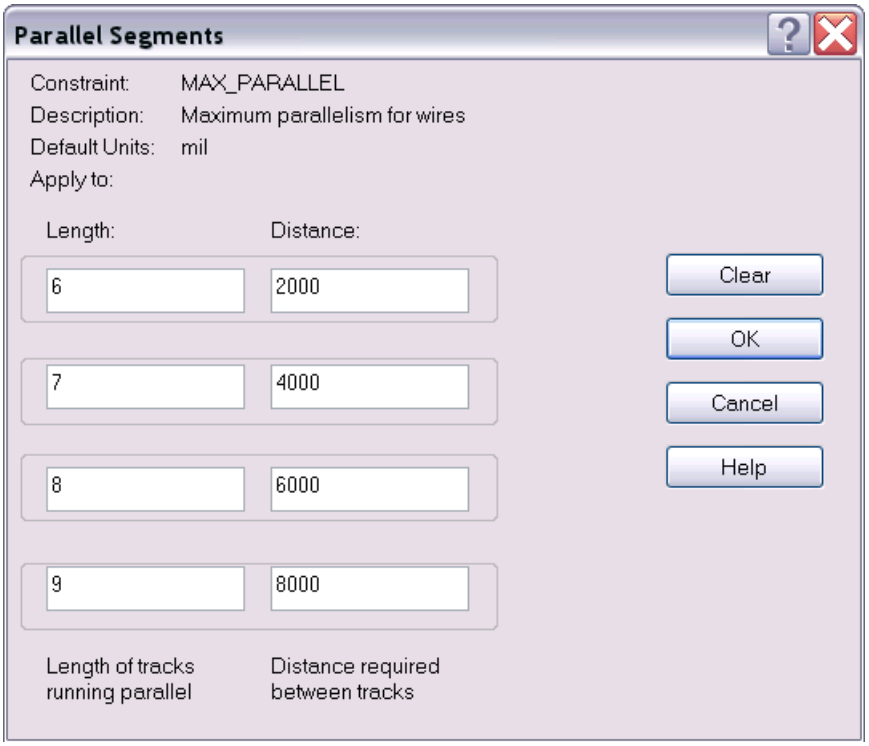
4. Click the right arrow.

5. Click *OK*.

Max Parallel

Use this command to specify up to four different parallelism rules for an object.

Example



The fields above should be interpreted as:

- Etch must be 6 mils apart from other etch running parallel for 2000 mils.
- Etch must be 7 mils apart from other etch running parallel for 4000 mils.
- Etch must be 8 mils apart from other etch running parallel for 6000 mils.
- Etch must be 9 mils apart from other etch running parallel for 8000 mils.

Constraint Manager stores these values as `2000,6,4000,7,6000,8,8000,9` in the *Parallel Max* column cells of the *Wiring* worksheet in the *Routing* workbook.