

Allegro® X Constraint Manager Reference

Product Version 23.1
September 2023

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

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

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

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

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

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

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

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

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

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

Contents

<u>File Menu Commands</u>	9
<u>File – Import – Constraints</u>	9
<u>File – Import – Technology File</u>	11
<u>File – Import – Electrical CSets</u>	14
<u>File – Import – Analysis Results</u>	16
<u>File – Import – Worksheet File</u>	17
<u>File – Import – Worksheet Customization</u>	18
<u>File – Import – Ravel File</u>	19
<u>File – Export – Text File</u>	23
<u>File – Export – Worksheet File</u>	24
<u>File – Export – Constraints</u>	26
<u>File – Export – Technology File</u>	28
<u>File – Export – Constraint Sets</u>	31
<u>File – Export – Analysis Results</u>	33
<u>File – Export – Worksheet Customization</u>	34
<u>File – Export – HTML File</u>	35
<u>File – File Viewer</u>	36
<u>File – Record Script</u>	37
<u>File – Stop Recording</u>	38
<u>File – Playback Script</u>	39
<u>File – Close</u>	40
 <u>Edit Menu Commands</u>	41
<u>Edit – Undo</u>	41
<u>Edit – Redo</u>	42
<u>Edit – Cut</u>	43
<u>Edit – Copy</u>	44
<u>Edit – Paste</u>	45
<u>Edit – Paste Special</u>	46
<u>Edit – Find</u>	47
<u>Edit – Find Next</u>	50

Allegro X Constraint Manager Reference

<u>Edit – Find Previous</u>	51
<u>Edit – Toggle Bookmark</u>	52
<u>Edit – Next Bookmark</u>	53
<u>Edit – Previous Bookmark</u>	54
<u>Edit – Go to Source</u>	55
<u>Edit – Change</u>	57
<u>Edit – Clear</u>	60
<u>Edit – Formula</u>	61
<u>Edit – Dependencies</u>	76
<u>Edit – Calculate</u>	77
<u>Edit – Calculate All</u>	78

<u>Objects Menu Commands</u>	79
<u>Objects – Filter</u>	79
<u>Objects – Filters re-apply</u>	86
<u>Objects – Select</u>	87
<u>Objects – Select and Show Element</u>	88
<u>Objects – Deselect</u>	89
<u>Objects – Expand</u>	90
<u>Objects – Expand All</u>	91
<u>Objects – Collapse</u>	92
<u>Objects – Waive</u>	93
<u>Objects – Restore</u>	95
<u>Objects – Create – Class</u>	96
<u>Objects – Create – Class-Class</u>	97
<u>Objects – Create – Region</u>	98
<u>Objects – Create – Region Class</u>	100
<u>Objects – Create – Region Class-Class</u>	101
<u>Objects – Create – Net Group</u>	102
<u>Objects – Create – RKOGroup</u>	105
<u>Objects – Create – Match Group</u>	107
<u>Objects – Create – Ratsnest Bundle</u>	111
<u>Objects – Create – Pin Pair</u>	114
<u>Objects – Create – Differential Pair</u>	116
<u>Objects – Create – Electrical CSet</u>	122

Allegro X Constraint Manager Reference

<u>Objects – Create – Physical CSet</u>	125
<u>Objects – Create – Spacing CSet</u>	126
<u>Objects – Create – Same Net Spacing CSet</u>	127
<u>Objects – Create – Assembly CSet</u>	129
<u>Objects – Create – High Voltage CSet</u>	130
<u>Objects – Add to – Class</u>	131
<u>Objects – Add to – Bus</u>	132
<u>Objects – Add to – Net Group</u>	133
<u>Objects – Add to – Match Group</u>	134
<u>Objects – Add to – Differential Pair</u>	135
<u>Objects – Add to – Ratsnest Bundle</u>	140
<u>Objects – Group members</u>	141
<u>Objects – Remove</u>	152
<u>Objects – Rename</u>	153
<u>Objects – Clear all Xnet renames</u>	155
<u>Objects – Rename Xnets to highest level in hierarchy</u>	156
<u>Objects – Delete</u>	158
<u>Objects – Constraint Set References</u>	160
<u>Objects – Change all design unit attributes</u>	166

Column Menu Commands 167

<u>Column – Analyze</u>	167
<u>Column – Analysis Modes</u>	169
<u>Column – Sort</u>	170

View Menu Commands 171

<u>View – View Options</u>	171
<u>View – Hide Column</u>	179
<u>View – Show All Columns</u>	180
<u>View – Expand All Rows</u>	181
<u>View – Collapse All Rows</u>	182
<u>View – Display Priority</u>	183
<u>View – Refresh</u>	185
<u>View – Always on Top</u>	186

Allegro X Constraint Manager Reference

<u>Audit Menu Commands</u>	187
<u>Audit – Constraints</u>	187
<u>Audit – SI Setup</u>	189
<u>Audit – Obsolete Objects</u>	190
<u>Audit – Invalid Objects</u>	192
<u>Audit – Electrical CSets</u>	194
<u>Window Menu Commands</u>	197
<u>Window – New Window</u>	197
<u>Window – Cascade</u>	198
<u>Window – Tile</u>	199
<u>Window – Arrange Icons</u>	200
<u>Window – Close All</u>	201
<u>Window – Previous Recent View</u>	202
<u>Window – Next Recent View</u>	203
<u>Window – Next Worksheet Tab</u>	204
<u>Window – Previous Worksheet Tab</u>	205
<u>Analyze Menu Commands</u>	207
<u>Analyze – Initialize</u>	207
<u>Analyze – Settings</u>	208
<u>Analyze – Analysis Modes</u>	212
<u>Analyze – Analyze</u>	234
<u>Analyze – Show Worst Case</u>	235
<u>Tools Menu Commands</u>	237
<u>Tools – SigXplorer</u>	237
<u>Tools – Explore Topology</u>	239
<u>Tools – SigWave</u>	242
<u>Tools – Constraint Compiler</u>	243
<u>Tools – Ravel – Delete All Markers</u>	249
<u>Tools – Setup Property Definitions</u>	250
<u>Tools – Precision</u>	257
<u>Tools – Options</u>	258

Allegro X Constraint Manager Reference

<u>Tools – Update Topology</u>	262
<u>Tools – Uprev Topology</u>	263
<u>Tools – Update DRC</u>	264
<u>Tools – Customize Worksheet</u>	265
<u>Tools – Report</u>	276
<u>Help Menu Commands</u>	279
<u>Help – Tip of the Day</u>	279
<u>Appendix A: Dialog Box Help</u>	281
<u>Edit Via List</u>	282
<u>Edit Via Structure</u>	290
<u>Ignore Nets</u>	294
<u>Max Parallel</u>	295

Allegro X Constraint Manager Reference

File Menu Commands

File – Import – Constraints

Use this command to import the dictionary and constraints file (`.dcfx`), 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.



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.

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 `.dcfx` file. With *Merge* enabled, Constraint Manager adds new constraints, new objects (including ECSets), and new constraints on objects imported from the `.dcfx` file to the design, as well as cross-section information.

Note: If the cross-section includes information on placed embedded components, then the import throws error and merge is aborted.

Objects and constraints in the design that are not in the `.dcfx` file remain in the design, unchanged. If constraint values differ between the design and the `.dcfx` file, the constraint value in the `.dcfx` file prevails over the constraint value in the design.

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

If an object in the design is constrained but the object in `.dcfx` file is not, Constraint Manager 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 `.dcfx` 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 `.dcfx` file. With *Overwrite All constraints* enabled, Constraint Manager adds new constraints, new objects (including ECSets), and new constraints on objects imported from the `.dcfx` file to the design. Constraint Manager clears objects in the design that are not in the `.dcfx` 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 a either an XML based Technology Constraint File .tcfx file ior 16.x technology file .tcf 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(.tcfx) Dialog Box

File name Specifies the name of the tech file, either the XML .tcfx file or the .tcf file.

Browse to find the tech file or click *Library* to display the Procedure.

Files of type Select either the new .tcfx file or the legacy .tcf file.

Import Mode You can import the tech file in one of three modes: *Merge*, *Replace*, or *Overwrite all*. *Overwrite all* mode is enabled by default.

Note: The Overwrite All mode ensures that the board stackup matches the technology file stackup. It does not actually delete all existing layers and then add them all back again. It finds matches and then inserts, adds, and/or deletes to perform the operation.

Merge constraints (Do not delete any objects, attributes or relationships) 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.

Allegro X Constraint Manager Reference

File Menu Commands

*Replace constraints
(Only update those
objects, which exist in
the input file)*

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
(Update all information)*

Click this button to enable *Overwrite all* 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.

Note: When you import a technology file, the locked/unlocked status of the CSets is ignored and the constraints from the technology file are processed. After processing, the CSets are locked or unlocked depending upon which flag, fObjectReadOnly or fObjectNOTReadOnly, is set in the technology file.

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.

tech

Available in the Look in field. Click to choose a file from those that exist in the directories defined by the TECHPATH environment variable. TECHPATH is an existing environment variable (envved command) which is supported by Constraint Manager.

Allegro X Constraint Manager Reference

File Menu Commands

Procedure

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

The Import a technology file (.tcfx) dialog box appears.

3. Enter the file name for a .tcf or .tcfx file. Either browse to find the file

-or-

click *tech* 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 (.tcfx) 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](#).

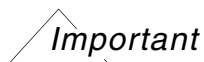
Dialog Boxes

Import an electrical ECSet file (.top) Dialog Box

<i>File name</i>	Specifies the name of the topology file .top file.
<i>Files of type</i>	Select the .top file.
<i>Update existing or create new Custom Measurement worksheet</i>	Choose to allow import of Custom Measurement Worksheet in Constraint Manager if a topology template (.top) file for an existing Electrical CSet has Custom Measurements defined in it.
<i>Help</i>	Displays help for this command.

Procedures

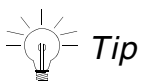
Importing a topology file



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.



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 – Import – Ravel File

Ravel provides a way to perform advanced level constraints checks in a design. You can use Ravel rules for checking various areas in a design, for example, manufacturing, electrical, and signal integrity.

RAVEL (Relational Algebra Verification Expression Language) is a language for writing DRC rules. It is also a DRC engine that checks design rules written in Ravel language in PCB Editor and SiP Layout.

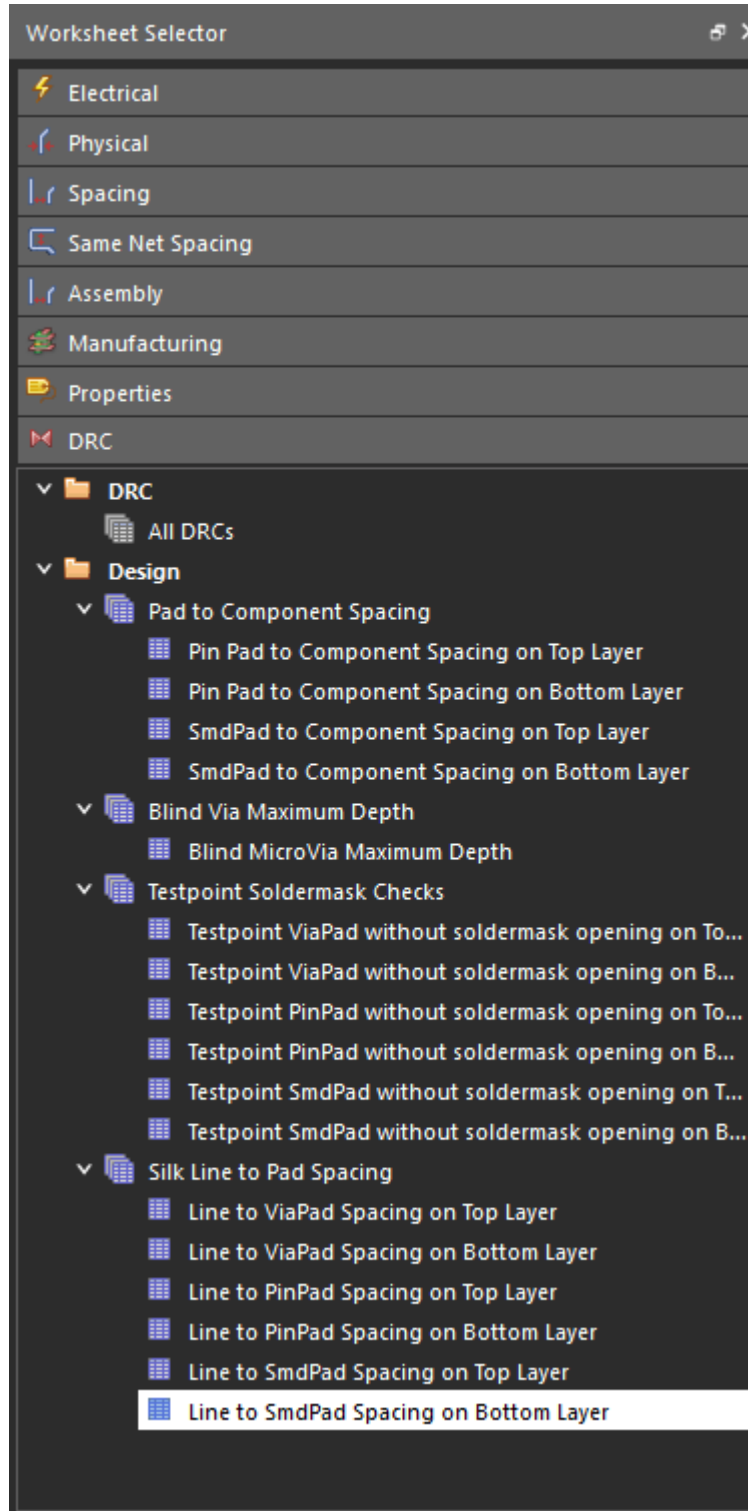
In Constraint Manager, you can use *File – Import – Ravel* command to import Ravel files (.rav and .ravc). The Ravel files are ASCII text files. The .rav files contains source code files and .ravc files are control files that refer to compiled rules file known as generic rule deck.

Note: Ensure that `RAVPATH` points to the top level directory where you have installed the Ravel Standard Library (RSL).

Allegro X Constraint Manager Reference

File Menu Commands

On importing a Ravel file, Ravel rules are added to the Constraint Manager. They are displayed as worksheets grouped as workbooks under Design folder in DRC domain.



Allegro X Constraint Manager Reference

File Menu Commands

Ravel checks are run always on a complete design. You can run either a single check or multiple checks on a design as follows:

- Select a single check and perform *Analyze*. DRC markers are created in the design where violations are found.
- Select multiple checks using *Objects – Reports* to perform analysis. A report file is created that includes all violations for each check.

When a Ravel check is re-run, the markers for that check are first deleted and then recreated. To remove Ravel DRC markers generated by a single check, fix violations in the design and re-run the check. To delete markers for all the checks use *Tools – Ravel – Delete All Markers* command in the Constraint Manager.

Use following license to run this functionality:

Allegro_Rel_Rules_Checker

Note: To write rules in the Ravel DRC language a developer license is required, which can be obtained through a services contract.

Procedure

1. Choose *File – Import – Ravel File*.

The *Import Ravel* dialog box appears.

2. Navigate to the directory where Ravel file resides.
3. Select a Ravel file.
4. Click *Open*.

Constraint Manager imports the Ravel file. A Ravel file may contain multiple rules. Each rule is added as a worksheet. These worksheets are organized in workbooks under Design folder in the DRC domain.

5. Choose a Ravel check from the Design folder.

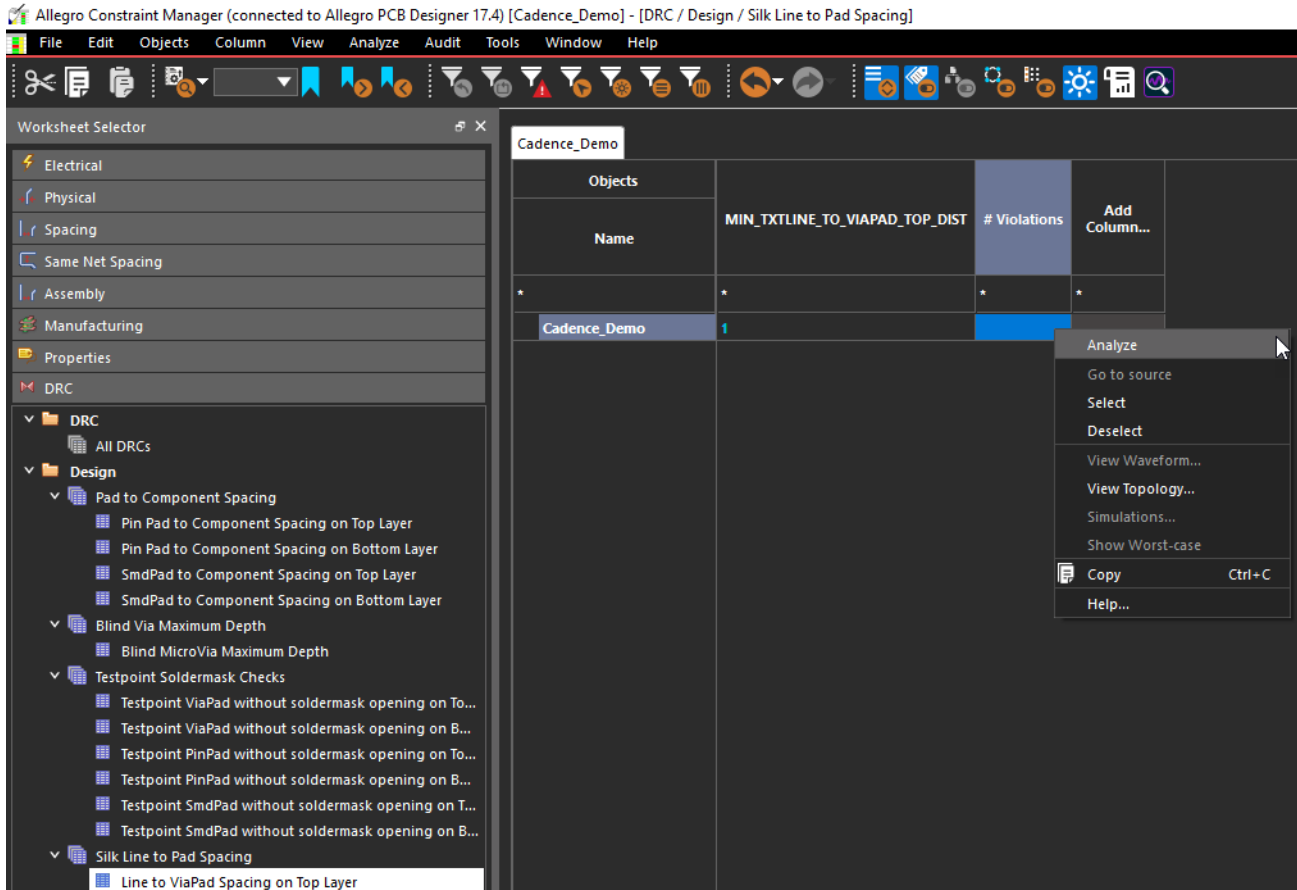
The selected Ravel check is displayed in the worksheet viewer.

6. In the worksheet, right-click and choose *Analyze*.

Allegro X Constraint Manager Reference

File Menu Commands

The selected Ravel check is performed on whole design and DRC markers are created in the design.



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.

Use this field . . .

To . . .

File name

Specify the name of the file.

Files of type

Select the type of file.

Expand All Rows

Select this check box to export all the data in the collapsed rows.

Note: If this check box is not selected, Constraint Manager exports only data that is visible on the worksheet.

*Only visible rows/
columns*

Exports only visible columns on the active worksheet.

*Only selected cells from
View*

Exports only selected cells on the active worksheet.

All rows/columns

Exports all the contents of the active worksheet.

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*.

Allegro X Constraint Manager Reference

File Menu Commands

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

File – Export – Constraints

Use this command to export a design constraints (.dcfx) . The design 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.

This command also lets you generate the generic cross-section information in the design constraint file. You can specify constraints for template cross-section in this file, which can later be imported to a design with any number of layers.

The generic cross-section information includes constraint values from all the generic layers. Generic layer types are combination of Layer Types (Conductor and Plane) and Constraint Types (user and system-defined). You can create new names in the Constraint Type column in the Cross-Section editor or can use system-defined Constraint Type names as generic layers.

In release 17.0, the Physical and Spacing Constraint Sets supports Hierarchical Layer Types: Conductor and Plane. The constraints applied at this level are inherited by the child layers of each group type.

When you import a .dcfx file with cross-section information, only the constraint information is updated based upon the current import modes. The cross-section of the design is not updated.



This command is not implemented when launching Constraint Manager from Allegro® Design Entry HDL or Allegro® System Architect.

Dialog Boxes

Export a dictionary and constraints file (.dcfx) Dialog Box

<i>File name</i>	Specifies the name of the .dcfx file.
<i>Contents</i>	
<i>User property definitions</i>	Check this box to export all user-defined property definitions to the .dcfx file.

Allegro X Constraint Manager Reference

File Menu Commands

<i>Electrical constraints</i>	Check this box to export all electrical constraint sets to the .dcfx file.
<i>Physical & spacing constraints</i>	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 .dcfx file.
<i>Export cross-section</i>	Choose the sub options to export the cross-section information to the .dcfx file. The cross-section information includes all the conductor, surface, dielectric, and bonding wire layers and their characteristics.
<i>None</i>	Check this option if you do not want to export cross-section information. This option is enabled only when <i>Physical & spacing constraints</i> is unchecked.
<i>Design-specific</i>	Choose to export design specific cross-section information.
<i>Generic</i>	Choose to export layer specific cross-section information.
<i>Properties</i>	Check this box to display a text description of the import results without performing the import task. The report is similar to the report generated by the <i>File – Import – Constraints</i> command.
<i>Assembly Constraints</i>	Check this box to export Packaging Assembly Constraints.
<i>Save</i>	Click this button to create the .dcfx file using the parameters that you applied.
<i>Cancel</i>	Click this button to cancel the command.
<i>Help</i>	Displays help for this command.

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 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 (`.tcfx`). 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 constraints, user property definitions, and design units. You can use the *Contents* section of this command to control the information in the file. The `.tcfx` file supports all the information which is currently available in the `.tcf` file.

Example

If you create a tech file using only the physical and spacing constraints, the following information is stored 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

This command also lets you generate the generic cross-section information in the design constraint file. You can specify constraints for template cross-section in this file, which can later be imported to a design with any number of layers.

The generic cross-section information includes constraint values from all the generic layers. Generic layer types are combination of Layer Types (Conductor and Plane) and Constraint Types (user and system- defined). You can create new names in the Constraint Type column in the Cross-Section editor or can use system-defined Constraint Type names as generic layers.

In release 17.0, the Physical and Spacing Constraint Sets supports Hierarchical Layer Types: Conductor and Plane. The constraints applied at this level are inherited by the child layers of each group type.

When you import a `.tcfx` file with cross-section information, only the constraint information, layer types, and cross-section are updated in the design in Overwrite mode only. Importing a `.tcfx` file in Replace mode does not change the cross-section of the design. It is not recommended to import a `.tcfx` file which contains cross-section information using Merge mode.

Export a technology file (.tcfx) Dialog Box

<i>File name</i>	Specifies the name of the tech file that you are creating.
<i>Contents</i>	
<i>Power Integrity constraints</i>	Check this box to export all power integrity constraints to the tech file.
<i>User property definitions</i>	Check this box to export all user-defined property definitions to the tech file.
<i>Electrical constraints</i>	Check this box to export all electrical constraint sets to the tech file.
<i>Physical & spacing constraints</i>	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.
<i>Export cross-section</i>	Choose the sub-options to export the cross-section information to the tech file. The cross-section information includes all the conductor, surface, dielectric, and bonding _wire layers and their characteristics.
<i>None</i>	Check this option if you do not want to export cross-section information. This option is enabled only when <i>Physical & spacing constraints</i> is unchecked.
<i>Design-specific</i>	Choose to export design specific cross-section information. If this option is selected, the stackup information is also exported to the tech file.
<i>Generic</i>	Choose to export layer specific cross-section information.
<i>Properties</i>	Check this box to display a text description of the export results without performing the export task. The report is similar to the report generated by the <i>File – Export – Constraints</i> command.

Allegro X Constraint Manager Reference

File Menu Commands

<i>Manufacturing Constraints</i>	Check this box to export the manufacturing constraints. Note: If this option is checked only inter-layer spacing constraints are exported as manufacturing constraints.
<i>Assembly Constraints</i>	Check this box to export the assembly constraints.
<i>Save</i>	Click this button to create the tech file using the parameters that you applied.
<i>Cancel</i>	Click this button to cancel the command.
<i>Help</i>	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 (.tcfx) dialog box appears.
3. Enter a file name for the tech file.
4. Complete the parameters to define the contents of the tech file.
5. Click *OK* to start the tech file creation process and dismiss the dialog box.

File – Export – Constraint Sets

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 – Constraint Sets*.

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 14), Constraint Manager compares the revision number when you update the topology (see [Tools – Update Topology](#) on page 262).

Exporting all ECSets

1. Choose *File – Export – Constraint Sets*.

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.

Allegro X Constraint Manager Reference

File Menu Commands

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 – Export – HTML File

Use this command to export an HTML file of the active worksheet.

Note: The command is not available for exporting HTML version of the following worksheets:

- *Manufacturing – Design for Fabrication*
- *Manufacturing – Design for Assembly*
- *Spacing – CSet assignment matrix*
- *Spacing – Inter Layer spacing*

Procedure

1. Choose *File – Export – HTML File*.

The *Export HTML File* dialog box appears.

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

Constraint Manager exports the active worksheet in an HTML format. The exported HTML file display all columns and rows in expanded form using the same color for background and texts.

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 – Record Script

Use this command records a series of actions. It creates a text file containing the commands that you execute and adds a `.scr` extension to the file name.

You can use scripts to perform various tasks in the Constraint Manager.

This option is available when Constrain Manager is launched from Allegro Design Entry HDL.

Record Dialog Box

<i>File Name</i>	Specifies the name of the file in which you record your actions.
<i>File Type</i>	Specifies the <code>.scr</code> extension to the file name.
<i>Open</i>	Starts recording your actions.
<i>Stop</i>	Stops recording your actions or replaying a script.
<i>Cancel</i>	Closes the dialog box.

Procedures

Creating a Script

1. Run the *File – Record Script* command.

The *Record* dialog box appears.

2. In the *File Name* text box, enter a name for the script.
3. Click *Open*.

The Record dialog box disappears.

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

File – Stop Recording

Use this command to stop the recording of a script.

This option is available when Constrain Manager is launched from Allegro Design Entry HDL.

File – Playback Script

Use this command to playback a series of actions recorded by a script.

This option is available when Constrain Manager is launched from Allegro Design Entry HDL.

Playback Dialog Box

<i>File Name</i>	Specifies the name of the file in which you want to replay.
<i>File Type</i>	Specifies the <code>.scr</code> extension to the file name.
<i>Open</i>	Starts replaying your actions.
<i>Cancel</i>	Closes the dialog box.

Procedures

Creating a Script

1. Run the *File – Playback Script* command.

The *Playback* dialog box appears.

2. In the *File Name* text box, enter a name for the script.
3. Click *Open*.

The *Playback* dialog box disappears and the script replays.

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 Allegro Design Authoring.

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. You can also copy multiple values from different columns and rows in Constraint Manager and paste the values into other multiple columns and rows in Constraint Manager.

Edit – Paste

Use this command to paste the value, from a cell that was copied or cut, into the selected cell. You can also copy cells from an Excel sheet and paste the values in a Constraint Manager worksheet. You can also use the Ctrl + C and Ctrl + V key combination to perform the copy/paste operation.

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

Choose one of the following options from the pull-down menu:

(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 . . .

Replace attribute value

To . . .

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).

Allegro X Constraint Manager Reference

Edit Menu Commands

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

Worst-case violation bubbles up to parent object

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.

Allegro X Constraint Manager Reference

Edit Menu Commands

Figure 2-1 CSet variances by layer

Constraint Set Folder View

Objects			Referenced Physical CSet	Line Width	
Type	S	Name		Min	Max
				mm	mm
*	*	*	*	*	*
Dsn		▼ Cadence_Demo	DEFAULT	0.200	0.000
PCS		▼ DEFAULT		0.200	0.000
LTyp		▼ Conductor		0.000	0.000
Lyr	1	TOP		0.200	0.000
Lyr	3	INNER1		0.200	0.000
Lyr	4	INNER2		0.200	0.000
Lyr	6	BOTTOM		0.200	0.000
LTyp		▼ Plane		0.000	0.000
Lyr	2	GND		0.200	0.000
Lyr	5	POWER		0.200	0.000

Net Folder View

Objects			Line Width
Type	S	Name	Min
			mm
*	*	*	*
NCIs		▼ POWER_GROUP(10)	
Net		GND EARTH	0.600
Net		V+ 12	0.600
Net		VCC	0.600
Net		V12N	0.600
Net		0	0.600
Net		1V2	0.600
Net		1V8	0.600
Net		2V5	0.600
Net		3V3	0.600
Net		5V	0.600

Calculate Ok Cancel Help

Objects			Line Width
Type	S	Name	Min
			mm
*	*	*	*
NCIs		▼ POWER_GROUP(10)	0.500
Net		GND EARTH	0.500
Net		V+ 12	0.500
Net		VCC	0.500
Net		V12N	0.500
Net		0	0.500
Net		1V2	0.500
Net		1V8	0.500
Net		2V5	0.500
Net		3V3	0.500
Net		5V	0.500

Calculate 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 77 or Edit – Calculate All on page 78). Constraint Manager performs the calculation and returns the calculated value to the cell that contains the formula.

Formulas are portable among designs. See

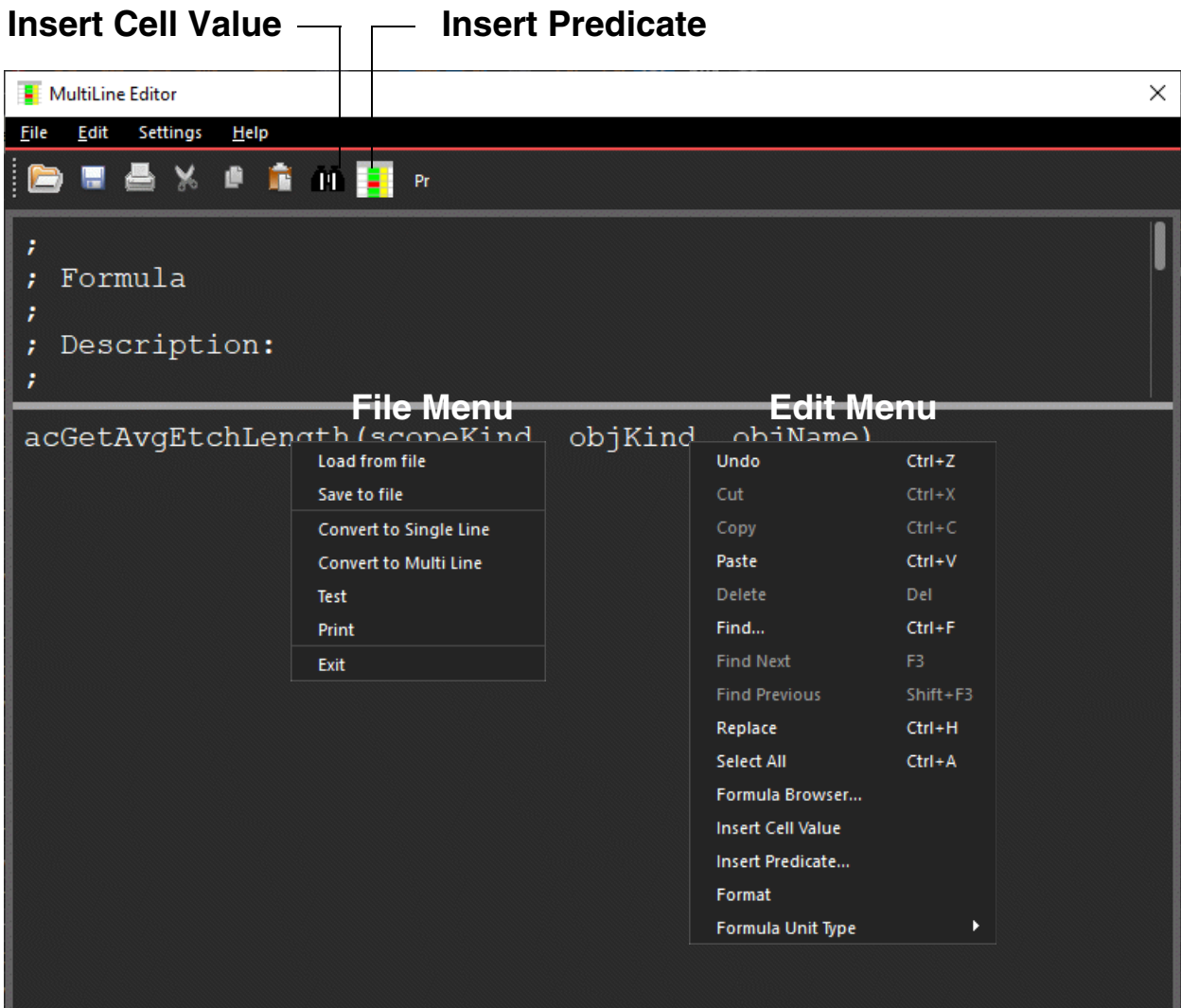
- File – Import – Constraints on page 9
- File – Import – Worksheet Customization on page 18
- File – Export – Constraints on page 26
- File – Export – Worksheet Customization on page 34

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



Allegro X Constraint Manager Reference

Edit Menu Commands

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.

Allegro X Constraint Manager Reference

Edit Menu Commands

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 66 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 66 for more detailed descriptions).

Allegro X Constraint Manager Reference

Edit Menu Commands

Use this command . . .

Formula Unit Type

To . . .

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

Allegro X Constraint Manager Reference

Edit Menu Commands

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

Allegro X Constraint Manager Reference

Edit Menu Commands

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, Net Classes, Differential Pairs, and Match Groups	Returns the average length of a group
acGetTotalLength	Buses, Net Classes, Differential Pairs, and Match Groups	Returns the total length of a group, including manhattan ratsnest length of unrouted segments
acGetAvgEtchLength	Buses, Net Classes, Differential Pairs, and Match Groups	Returns the average etch length of a group
acGetTotalEtchLength	Buses, Net Classes, Differential Pairs, and Match Groups	Returns the total etch length of a group
acGetAvgManhattanLength	Buses, Net Classes, Differential Pairs, and Match Groups	Returns the average manhattan length of a group
acGetTotalManhattanLength	Buses, Net 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

Allegro X Constraint Manager Reference

Edit Menu Commands

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, Net Classes, Differential Pairs, and Match Groups	Returns the minimum length of a group
acGetMinEtchLength	Buses, Net Classes, Differential Pairs, and Match Groups	Returns the minimum etch length of a group
acGetMinManhattanLength	Buses, Net 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, Net Classes, Differential Pairs, and Match Groups	Returns the maximum length of a group
acGetMaxEtchLength	Buses, Net Classes, Differential Pairs, and Match Groups	Returns the maximum etch length of a group
acGetMaxManhattanLength	Buses, Net 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

Allegro X Constraint Manager Reference

Edit Menu Commands

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 <i>Actual</i> with the strings <i>Routed</i> or <i>Unrouted</i> .
acIsPlaced		Returns TRUE if the components related to an object are all placed; nil otherwise. The corresponding measurement will populate the associated <i>Actual</i> with the strings <i>Placed</i> or <i>Unplaced</i> .
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.

Allegro X Constraint Manager Reference

Edit Menu Commands

Predicate	Scope	Function
acGetValue		<p>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.</p> <ul style="list-style-type: none">■ 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

Allegro X Constraint Manager Reference

Edit Menu Commands

Predicate	Scope	Function
<code>acGetValueUnitString</code>		<p>Returns a string containing the units used in the value specified by the parameters.</p> <p>For example, if the parameters are set up to access a Min Prop Delay constraint value of "10 ns," this function returns "ns."</p> <p>Most predefined properties in Constraint Manager do not contain the unit string; therefore, the function just returns an empty string.</p>
<code>acDebug_print*</code>		<p>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.</p> <ul style="list-style-type: none">■ <code>acDebug_print</code> corresponds to SKILL print■ <code>acDebug_println</code> corresponds to SKILL println■ <code>acDebug_printf</code> corresponds to SKILL printf■ <code>acDebug_pprint</code> corresponds to SKILL pprint
<code>acAddValueDependency</code>		<p>Specify an attribute value for the calling function. Used internally to check if the value of the function is outdated or not.</p>

Allegro X Constraint Manager Reference

Edit Menu Commands

Predicate	Scope	Function
<code>acAddObjectDependency</code>		Specify an object dependency for the calling function. Used internally to check if the value of the function is outdated or not.
<code>acCreateExternalDRC</code>		Creates external DRCs to indicate a violation of the rule using user-defined measurement for an object. The external DRCs are recorded in the Design Rules Check (DRC) Report and can be viewed as bowties in the design canvas.
<code>acDebug_checkObject</code>	Design object	Checks validity of an object. This predicate is active only in a test environment. An error message is displayed if the specified object is not found.
<code>acDeleteExternalDRC</code>		Deletes previously created external DRCs from an object. Use to remove user-defined DRCs prior to DRC check.
<code>acGetBranchCount</code>	Xnets and Nets	Returns the number of branches that are routed for an Xnet or net.
<code>acGetEtchLengthByLayer</code>	Xnets and Nets	Returns the etch length of a Xnet or net on a specified layer.
<code>acGetEtchLengthPinPair</code>	Pin Pairs	Returns the total etch length of a pin pair.
<code>acGetLengthPinPair</code>	Pin Pairs	Returns the total length of a pin pair. Uses both etch and manhattan length of connections which are not routed.
<code>acGetManhattanLengthPinPair</code>	Pin Pairs	Returns the manhattan ratsnest length of a pin pair.
<code>acGetPercentManhattanPinPair</code>	Pin Pairs	Returns the ratio of the actual length of a pin pair divided by its manhattan length.

Allegro X Constraint Manager Reference

Edit Menu Commands

Predicate	Scope	Function
acGetViaCount	Xnets, Nets, and Pin Pairs	Returns the number of vias on a net, Xnet, or pin pair.
acGetLayerThickness	Layer objects	Returns the thickness of the specified cross-section layer.

Note: All advanced constraints delay measurements include the Z-axis delay if calculations are performed on Pin Pairs.

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 77 or Edit – Calculate All on page 78.

Edit – Dependencies

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

- [Edit – Formula](#) on page 61
- [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.

Edit – Calculate

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

- [Edit – Formula](#) on page 61
- [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.

Edit – Calculate All

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

- [Edit – Formula](#) on page 61
- [Customizing Design Rule Checks](#) in the *Constraint Manager User Guide*

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 . . .
<i>Object types filter</i>	Control which objects to display. The unchecked objects in this field restrict the objects available for display.
<i>Restore default Object filter</i>	Restore the object type filter settings to the default settings for the active worksheet.
Advanced filters	
<i>Selected nets/xnets only</i>	Show only those nets that are selected in PCB Editor's canvas. Note: <i>Application Select</i> must be enabled (choose <i>View – Options</i>).
<i>Highlighted nets/xnets only</i>	Highlight only those nets that are selected in PCB Editor's canvas.

Allegro X Constraint Manager Reference

Objects Menu Commands

Use this field . . .	To . . .
<i>Failed only</i>	Show only those objects with constraint violations (<i>Actual</i> and <i>Margin</i> cells rendered red).
<i>Objects with Directly Set values</i>	Show only those objects that have directly set constraints.
<i>Ignore Referenced CSets</i>	Ignore objects that have referenced CSets.
<i>Columns with Directly Set values</i>	Show only those columns that have directly set constraints
<i>Active partition only</i>	Show only those objects in the active partition (when running with a partitioned design).
<i>Active DRCs</i>	Show the design rule checks that are preserved. See Objects – Waive on page 93 for more information.
<i>Waived DRCs</i>	Show suppressed design rule checks. See Objects – Waive on page 93 for more information.
<i>Global Objects (G)</i>	<p>Shows global nets, Xnets, and buses. The global objects are displayed as <i>G</i> in <i>Sub-type</i> column.</p> <p>This option is available when launch Constraint Manager from Allegro® Design Entry HDL or Allegro® System Architect.</p>
<i>Interface Objects (I)</i>	<p>Shows Interface nets, Xnets, and buses. The interface objects are displayed as <i>I</i> in <i>Sub-type</i> column.</p> <p>This option is available when launch Constraint Manager from Allegro® Design Entry HDL or Allegro® System Architect.</p>
<i>Local Objects</i>	<p>Shows Local nets, Xnets, and buses. Shows Interface signals. The local objects are displayed as blank in <i>Sub-type</i> column.</p> <p>This option is available when launch Constraint Manager from Allegro® Design Entry HDL or Allegro® System Architect.</p>
<i>ICs</i>	<p>Suppress display of discrete components (Part Instance Object Types) from the Component Properties worksheets.</p> <p>This option is available when launch Constraint Manager from layout editors.</p>

Allegro X Constraint Manager Reference

Objects Menu Commands

Use this field . . .

Always show group members

To . . .

Disables any XNets/Nets object suppression behavior when they are members of a Net Group or Net Class.

For example, if this option is enabled, Power/Ground nets which have Voltage properties are suppressed from the Electrical domain worksheets but visible when they are members of a Net Group or Net Class.

To work with legacy mode uncheck this option.

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.

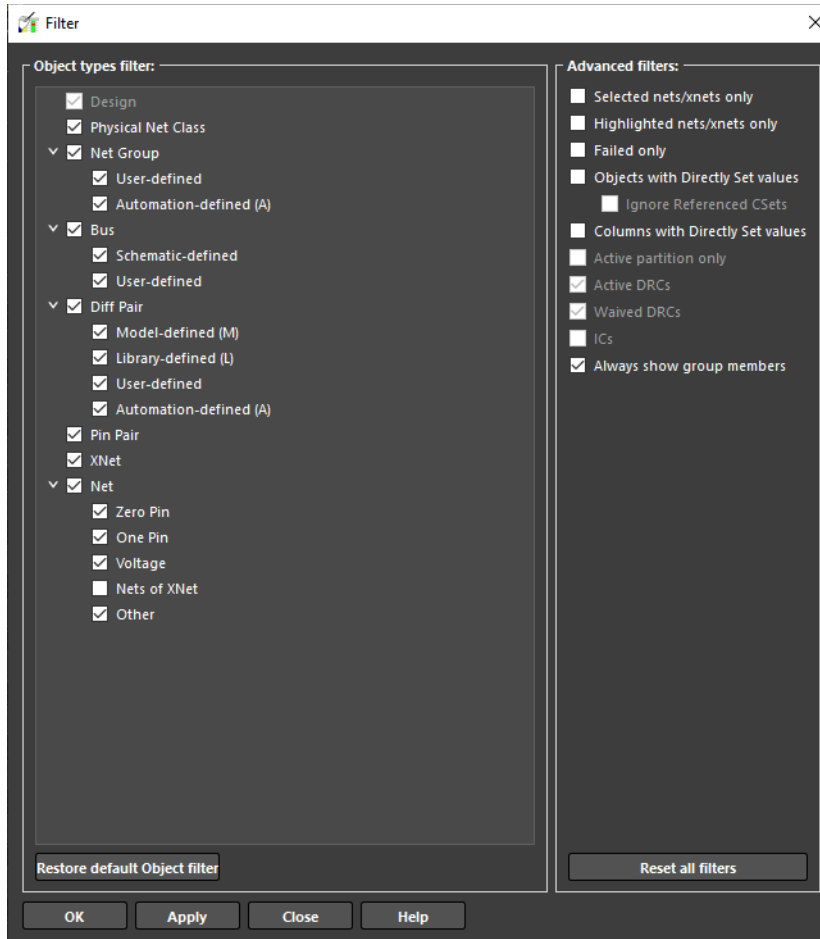
Filtering with the Filter dialog box

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

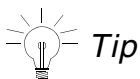
Allegro X Constraint Manager Reference

Objects Menu Commands

The *Filter* dialog box appears with default settings for the active worksheet. See [Table 3-3](#) on page 79 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.

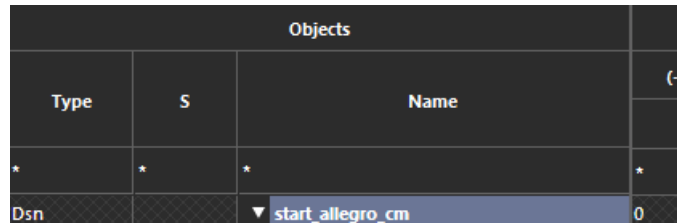
1. From an active worksheet, click the top row of a non-numeric column, such as *Vias* in the *Physical* domain.

Allegro X Constraint Manager Reference

Objects Menu Commands

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.



Objects			
Type	S	Name	(-)
*	*	*	*
Dsn		▼ start_allegro_cm	0

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)

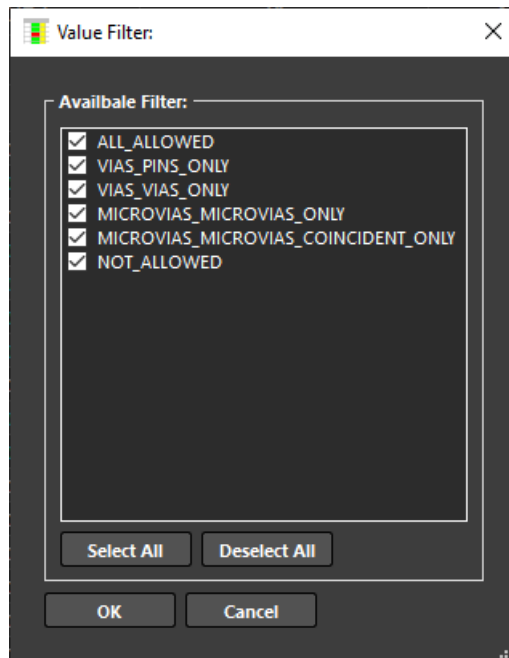
The *Filter* takes advantage of defined lists, such as Object Types, CSets, and other columns with user-selectable values by presenting you with a list of check boxes to control what appears in the selected column.

Allegro X Constraint Manager Reference

Objects Menu Commands

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 – Expand All

Use this command to view all the child tree nodes 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 All*.
 - or -
 - ☐ Right-click and choose *Expand All* 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.

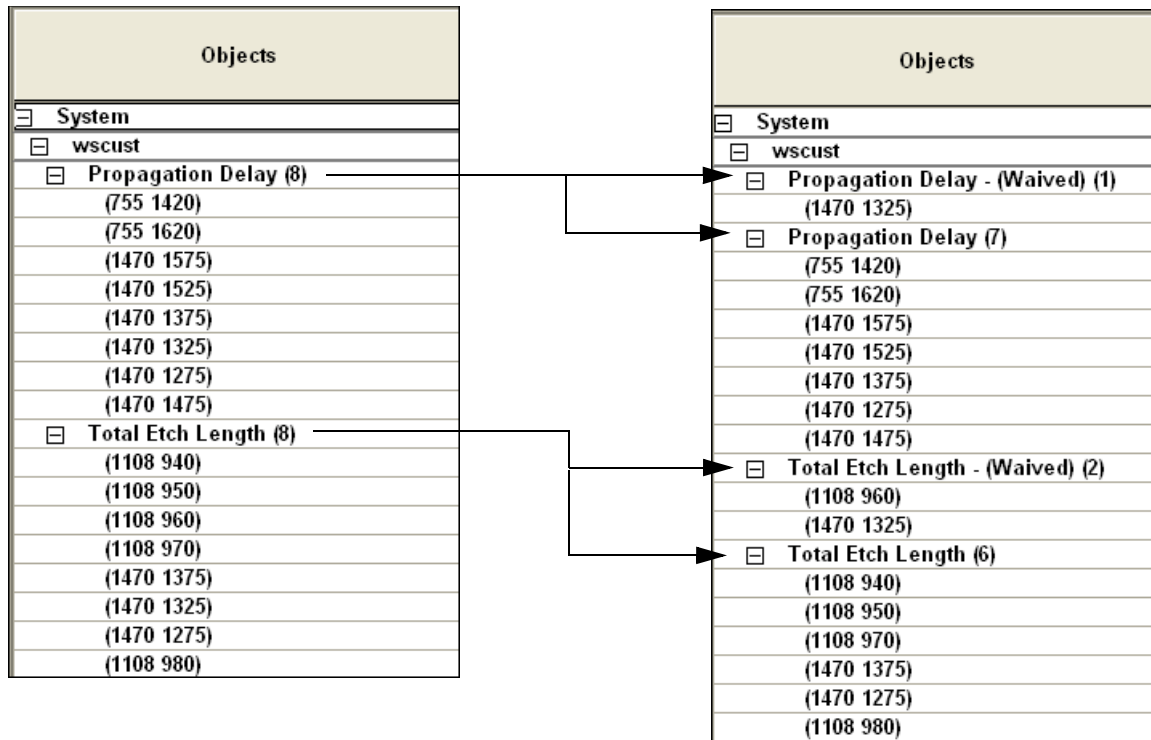
Procedure

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 95 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 79.
5. Click to select a DRC instance to suppress.
6. Optionally, enter a user-friendly name in the *Comment* column.

Allegro X Constraint Manager Reference

Objects Menu Commands

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 93 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 79.

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 – 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 – Class*.

-or-

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

3. Check *Create for both 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 *NCIs* in the *Type* column.

See also: [Objects – Group members](#) on page 141.

Objects – Create – 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.

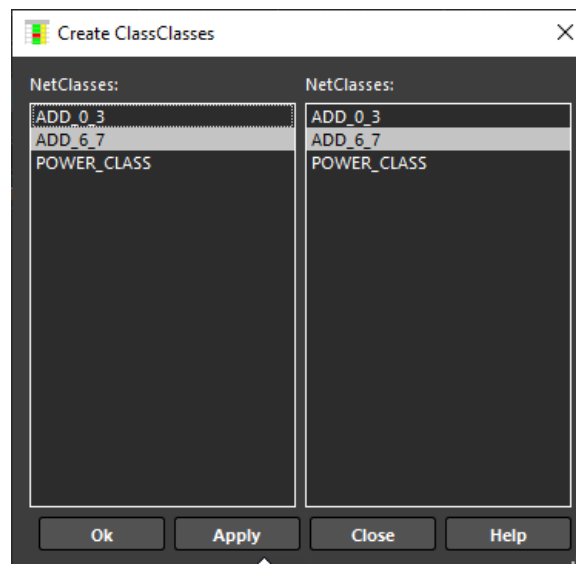
2. Do one of the following:

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

-or-

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

3. Specify the Class-to-Class relationship in the *Create ClassClasses* dialog box.



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,

Assigning Region to Single Shape

- 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.

Allegro X Constraint Manager Reference

Objects Menu Commands

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).

Note: You can however, select multiple shapes and assign a single constraint region to them. For instructions, see [assign region](#) command in the *Allegro PCB and Package Physical Layout Command Reference*.

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 *Rcls* 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-Class 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 – Net Group

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



Tip

For more information, see [Net Groups](#) in the *Allegro Constraint Manager User Guide*.

Procedures

Creating a net group

1. Select a group of nets, Xnets, or differential pairs.
2. Do one of the following:
 - ☐ Choose *Objects – Create – Net Group*
– or –
 - ☐ Right-click and choose *Create – Net Group* from the pop-up menu.
3. Enter a name for the Net Group.
4. Click *OK*.

Deleting a Net Group

1. Select a net group.
2. Do one of the following:
 - ☐ Choose *Objects – Delete*
– or –
 - ☐ Press *Delete*.

Constraint Manager destroys the net group object and preserves the individual members.

Renaming a Net Group

1. Select a net group.
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 net group

1. Select a net group.
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 net group in the PCB Editor

1. In Constraint Manager, select a net 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 select object in Constraint Manager is highlighted (or de highlighted) in PCB Editor or in APD.

Examining net group membership

1. Select a net group.
2. Do one of the following:
 - ☐ Choose *Objects – Group members*.
– or –

Allegro X Constraint Manager Reference

Objects Menu Commands

- ❑ Right-click and choose *NetGroup members* from the pop-up menu.
– or –
- ❑ Right-click on the net group name and choose *Expand* from the pop-up menu.

Redefining net group membership

1. Select a net group.
2. Do one of the following:
 - ❑ Choose *Objects – Group members*.
– or –
 - ❑ Right-click and choose *NetGroup members* from the pop-up menu.
3. Add or remove nets, Xnets, or pin pairs as appropriate.
4. Click *OK*.

Objects – Create – RKOGroup

Use this command to create a keepout exception by net group where physical elements of member nets are allowed and do not report keepout DRCs. You can add nets, Xnets, net groups, and differential pairs as a member of an RKO group.



Tip

For more information, see [RKO Groups](#) in the *Allegro® Constraint Manager User Guide*.

Procedures

Creating a RKO group

1. In the *Properties* domain, expand *Net* folder.
2. Select a group of nets, Xnets, net groups, or differential pairs in the *Route/Vias Keepout Exception* worksheet.
3. Do one of the following:
 - ☐ Choose *Objects – Create – RKO Group*
– or –
 - ☐ Right-click and choose *Create – RKO Group* from the pop-up menu.
4. Check *Preserve Existing Membership*, if applicable.
The same member can exist in both the new and existing RKO groups.
5. Enter a name for the RKO group, or accept the default.
6. Click *OK*.

Deleting a RKO Group

1. Select a RKO group.
2. Do one of the following:
 - ☐ Choose *Objects – Delete*
– or –

- ☐ Press *Delete*.

Constraint Manager destroys the RKO group object and preserves the individual members.

Renaming a RKO Group

1. Select a RKO group.
2. Do one of the following:
 - ☐ Choose *Objects – Rename*
– or –
 - ☐ Right-click and choose *Rename* from the pop-up menu.

Examining RKO group membership

1. Select a RKO group.
2. Do one of the following:
 - ☐ Choose *Objects – Group members*.
– or –
 - ☐ Right-click and choose *RKOGroup members* from the pop-up menu.

Redefining RKO group membership

1. Select a RKO group.
2. Do one of the following:
 - ☐ Choose *Objects – Group members*.
– or –
 - ☐ Right-click and choose *RKOGroup members* from the pop-up menu.
3. Check *Preserve Existing Membership*, if applicable.

The same member can exist in both the new and existing RKO groups.
4. Add or remove nets, Xnets, diff pairs, buses, or net groups as appropriate.
5. 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 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. 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 – Group members*.

– or –

- ☐ Right-click and choose *Match Group members* 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 – Group members*.

– or –

- ☐ Right-click and choose *Match Group members* from the pop-up menu.

3. Add or remove nets, Xnets, or pin pairs as appropriate.

Allegro X Constraint Manager Reference

Objects Menu Commands

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 – Add to – 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 [Ratsnest Bundle Membership](#) on page 150.

Redefining ratsnest bundle membership

See [Ratsnest Bundle Membership](#) on page 150.

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.

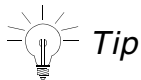
The pin pair is selected (or deselected) in PCB Editor or in APD.

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.



To learn more about working with Differential Pair constraints, see

- ❑ [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.

Allegro X Constraint Manager Reference

Objects Menu Commands

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.

Allegro X Constraint Manager Reference

Objects Menu Commands

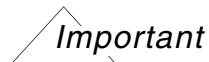
Use this field . . .

Create

To . . .

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.



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.



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.

Allegro X Constraint Manager Reference

Objects Menu Commands

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. Select *Electrical Modes* in the left pane.
3. Enable All differential pair checks.
4. In the *Objects* column, click on a differential pair object.

5. 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 160).

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 160).

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.

Allegro X Constraint Manager Reference

Objects Menu Commands

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 160).

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 160).

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 160).

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 160).

Objects – Create – Assembly CSet

Use this command to create an *Assembly CSet*. An assembly constraint set is a collection of rules that allow you to gauge whether the package, as designed, will meet the physical and spacing requirements necessary for the part to be successfully manufactured and assembled. You can apply an assembly constraint to a wirebond, die and design.

You define assembly rules under the *Assembly Constraint Set* object folder. These rules can subsequently be applied to wirebond and die objects.

Creating an Assembly CSet

1. Do one of the following:

- ☐ Choose *Objects – Create – Assembly CSet*.

- or -

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

The *Create Assembly CSet* dialog box appears and the seed constraint is identified by *copy constraints from*.

2. Enter a name for the *Assembly CSet*.

3. Click *OK*.

Constraint Manager creates a new *Assembly CSet* with the blank values.

4. Change constraint values as desired.

For information about assigning the Assembly CSet to objects, see [“Objects – Constraint Set References”](#) on page 160.

Objects – Create – High Voltage CSet

Use this command to create a *High Voltage CSet*. A high voltage constraint set is a collection of rules that help ensure that high voltage objects in the design meet the creepage and clearance spacing requirements.

You define high voltage rules under the *Electrical Constraint Set* and *Net* object folders in the *Electrical* domain to specify electrical clearances between copper objects with high voltage differences. These rules can subsequently be applied to high voltage objects which include nets, xnets, and net classes.

Creating a High Voltage CSet

1. Do one of the following:

- ☐ Choose *Objects – Create – High Voltage CSet*.

- or -

- ☐ Right-click and choose *Create High Voltage CSet* from the pop-up menu.

The *Create High Voltage CSet* dialog box appears and the seed constraint is identified by *copy constraints from*.

2. Enter a name for the *High Voltage CSet*.

3. Click *OK*.

Constraint Manager creates a new *High Voltage CSet* with the blank values.

4. Change constraint values as desired.

Assign the High Voltage CSet to net-based objects (see [“Objects – Constraint Set References”](#) on page 160).

Objects – Add to – Class

Use this command to add the selected net object to an existing *Net Class*.

Procedure

1. In the *Objects* column, select the net object to be added to the Net Class.

2. Perform one of the following tasks:

☐ Choose *Objects – Add to – Class*.

- or -

☐ Right-click and choose *Add to – Class*.

The *Add to NetClass* dialog box appears.

3. From the drop-down list, select the Net Class to which selected object is to be added.

Optionally, use the *Current Members* button to view the existing members of the net class.

4. Click *OK*.

Objects – Add to – Bus

Use this command to add the selected net object to an existing *Bus*.



This option is enabled only if you have an existing *Bus*. Starting 16.6 release, new bus creation is not supported in PCB Editor. Instead of buses use Net Groups.

Procedure

1. In the *Objects* column, select the net object to be added to the *Bus*.
2. Choose *Objects – Add to – Bus*.

Alternatively, right-click on the net object and choose *Add to – Bus*.

The *Add to Bus* dialog box appears.

3. From the drop-down list, select the *Bus* to which selected object is to be added.
Optionally, use the *Current Members* button to view the existing members of the bus.
4. Click *OK*.

Objects – Add to – Net Group

Use this command to add the selected net object to an existing *Net Group*.

A net group is a collection of different type of net objects. Net objects, such as, nets, buses, differential pairs, and XNets, can be the members of a Net Group.

Procedure

1. In the *Objects* column, select the net object to be added to the Net Group.
2. Choose *Objects – Add to – Net Group*.

Alternatively, right-click on the net object and choose *Add to – Net Group*.

The *Add to Net Group* dialog box appears.

3. From the drop-down list, select the Net Group to which selected object is to be added.

Optionally, use the *Current Members* button to view the existing members of the Net Group.

4. Click *OK*.

Objects – Add to – Match Group

Use this command to add the selected net object as a member of an existing *Match Group*.

Note: This command is enabled only if you have existing Match Groups in the design.

Procedure

1. In the *Objects* column, select the net object to be included in the Match Group.
2. Choose *Objects – Add to – Match Group*.

Alternatively, right-click on the net object and choose *Add to – Match Group*.

The *Add to Match Group* dialog box appears. The drop-down lists displays the list of Match Groups already created.

3. From the drop-down list, select the Match Group to which selected object is to be added.

Optionally, use the *Current Members* button to view the existing members of the Match Group.

In the *Add to Match Group* dialog box, the *Preserve existing membership* check box is selected by default. This ensures that on adding new members to the match group, member nets or Xnets in the original Match Group are preserved along with the new additions.

4. (Optional) To remove the member net or Xnet in the original Match Group on addition of new members, clear the *Preserve existing membership* check box.
5. Click *OK*.

Objects – Add to – 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. For *model-defined* differential pairs, you can only examine the members 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.

Allegro X Constraint Manager Reference

Objects Menu Commands

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.

Create

Create the differential pair object with the two nets or Xnets in the selections column as its members.

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).

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.

Allegro X Constraint Manager Reference

Objects Menu Commands

Use this field . . .

+ *Filter*

- *Filter*

Create

To . . .

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.

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.

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 – Group Members*.

- or -

- ☐ In the Objects column, right-click on a differential pair and choose *Diff Pair members*.

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 116.

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. Select *Electrical Modes* on the left pane.
3. Enable All differential pair checks.
4. In the *Objects* column, click on a differential pair object.
5. 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 – Add to – Ratsnest Bundle

Use this command to add a pin pair as a member of a ratsnest bundle. A pin pair cannot be a member of more than one ratsnest bundle.

Procedure

1. In the *Objects* column, select the pin pair to be added to the ratsnest bundle.
2. Choose *Objects – Add to – Ratsnest Bundle*.

Alternatively, right-click on the pin pair and choose *Add to – Ratsnest Bundle*.

The *Add to Ratsnest Bundle* dialog box appears.

3. From the drop-down list, select the bundle to which selected pin pair object is to be added.
4. If required, use the *Current Members* button to view the existing members of the ratsnest bundle.
5. Click *OK*.

Objects – Group members

Use this command to view and redefine the membership of the selected net object. Net objects supported by this command is enabled for following net objects.

- Net Class
- Net Group
- Bus
- Match Group
- Differential Pair
- Ratsnest Bundle

Net Class Membership

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 – Group members*.

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*.

Allegro X Constraint Manager Reference

Objects Menu Commands

Net Class Membership Dialog Box

This dialog box lets you view and modify the members of an existing net class. You can use the fields described in the following table, to add or remove members from the net class.

Use this field . . .	To . . .
<i>Drop-down list</i>	Select an appropriate option to display all nets, Xnets, Nets, Buses, Differential Pairs, or NetGroups in the design.
<i>Left pane</i>	Displays a list of all available objects of the type selected from the drop-down list.
<i>Current Members</i>	Displays a list of all the objects that are member of the Net Class. <i>Name:</i> Displays the member name. <i>Type:</i> Displays the type of the object. <i>Net Class:</i> Displays the name of the net class of which the net is a member.
<i>Right (>)and left (<)arrow buttons</i>	Use these buttons to add or remove a net object to the net class.
<i>Filter</i>	Use this field to limit the net objects displayed in the left pane. Enter a string (and wildcard characters) to further focus on the object types chosen with the filter drop-down list.

Net Group Membership

Procedure

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

To aid you in locating a *Net Group*, use *Objects – Filter* commands.

2. Choose *Objects – Group members*.

The NetGroup membership Dialog Box appears.

3. Optionally, use the *Object Type* drop-down menu in the in the *NetGroup 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*.

NetGroup membership Dialog Box

This dialog box lets you view and modify the members of an existing net group. You can use the fields described in the following table, to add or remove members from the net group.

Use this field . . .	To . . .
<i>Drop-down list</i>	Select an appropriate option to display all nets, Xnets, Nets, Buses, Differential Pairs, or NetGroups in the design.
<i>Left pane</i>	Displays a list of all available objects of the type selected from the drop-down list.
<i>Current Members pane</i>	Displays a list of all the objects that are member of the Net Group. <i>Name</i> : Displays the member name. <i>Type</i> : Displays the type of the corresponding object. <i>Net Group</i> : Displays the name of the net group of which the net is a member.
<i>Right (>) and left (<) arrow buttons</i>	Use these buttons to add or remove a net object to the net class.

Allegro X Constraint Manager Reference

Objects Menu Commands

Use this field . . .

Filter

To . . .

Use this field to limit the net objects displayed in the left pane.

Enter a string (and wildcard characters) to further focus on the object types chosen with the filter drop-down list.

RKO Group Membership

Procedure

1. In the *Objects* column, select a *RKO Group*.
To aid you in locating a *RKO Group*, use *Objects – Filter* commands.
2. Choose *Objects – Group members*.
The RKOGROUP membership Dialog Box appears.
3. Optionally, use the *Object Type* drop-down menu in the in the *RKOGROUP Membership* dialog box to filter your selection.
4. Use the left and right arrows to populate the *Current Members* column to suit your needs.
5. Check *Preserve Existing Membership*, if applicable.
6. Click *OK*.

RKOGROUP membership Dialog Box

This dialog box lets you view and modify the members of an already existing RKO group. You can use the fields described in the following table, to add or remove members from the RKO group.

Use this field . . .	To . . .
<i>Filter drop-down menu</i>	Select appropriate option to display all nets, Xnets, buses, net groups, or differential pairs in the design.
<i>Left column</i>	List all available net object of the type selected from the Filter drop-down list.
<i>Filter type in field</i>	Use this field to limit the net objects displayed in the Left column. Enter a string (and wildcard characters) to further focus on the object types chosen with the filter drop-down menu.
<i>Members (right column)</i>	List the current members of the selected RKO group.

Note: The RKO group name is displayed in the title bar of the dialog box.

Allegro X Constraint Manager Reference

Objects Menu Commands

Use this field . . .

Right (>) and left (<) arrow buttons

Preserve existing membership

To . . .

Use these buttons to add or remove a net object to the RKO group.

Use this button to add a net object to the multiple RKO groups.

If unchecked, the net object will be removed from its previous RKO group.

Bus Membership

Procedures

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

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 . . .

Filter drop-down menu

Filter type in field

Left column

Members (right column)

To . . .

Focus a selection to nets, Xnets, or Ratsnest Bundles.

Enter a string (and wildcard characters) to further focus on the object types chosen with the filter drop-down menu.

List all available nets or Xnets in the design.

Use the filter drop-down menu and type-in field to limit the list.

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 – Group members*.

- or -

- ☐ Right-click and choose *Bus Members* 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 – Group members*.

- or -

- ☐ Right-click and choose *Bus Members* 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.

Match Group Membership

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

Allegro X Constraint Manager Reference

Objects Menu Commands

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

Procedures

Examining a Match Group

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

- ☐ Choose *Objects – Group members*.
- or -
- ☐ Right-click and choose *Match Group Members* 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 – Group members*.
- or -
- ☐ Right-click and choose *Match Group Members* 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*.

Differential Pair Membership

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

Note: Using Constraint Manager, you can examine or change the members of a *user-defined* differential pair objects only. In case of *model-defined* differential pair, you can

only examine the differential pair members in Constraint Manager.

Procedure

- In the Object column, select the differential pair and choose *Objects – Group Members*.

- or -

- Right-click and choose *Diff Pair members* from the pop-up menu.

The Differential Pair Membership Dialog Box displays. Use this dialog box to view and modify the membership of the selected differential pair.

Ratsnest Bundle Membership

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.

Procedures

Examining membership of a ratsnest bundle

1. In the Objects column of the Ratsnest Bundle Properties worksheet (in the Properties domain), select one or more member pin pairs of an expanded ratsnest bundle.
2. Right-click on the selection and choose *Ratsnest Bundle members* from the pop-up menu.

The *Ratsnest Bundle* dialog box appears.

3. Click *Current Members*.

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

4. Click *Close*.

Changing membership of a ratsnest bundle

1. In the Objects column of the Ratsnest Bundle Properties worksheet (in the Properties domain), select one or more member pin pairs of an expanded ratsnest bundle.

Allegro X Constraint Manager Reference

Objects Menu Commands

2. Right-click on the selection and choose *Ratsnest Bundle members* from the pop-up menu.

The *Ratsnest Bundle* dialog box appears.

3. From the drop-down menu, click to select from the available bundles in the design.
4. 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.

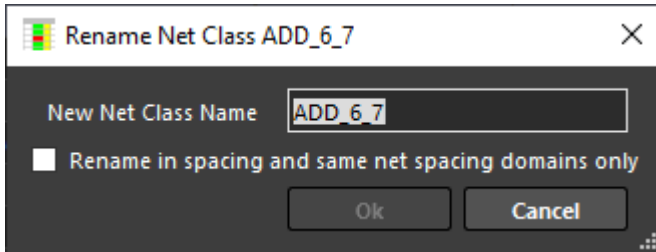
Renaming a Net Class

1. In the *Objects* column in the *Net* folder in the Spacing or Physical domain, select the Net Class to be renamed.
 2. Do one of the following:
- ☐ Choose *Objects – Rename*.
- or -
- ☐ Right-click and choose *Rename* from the pop-up menu.

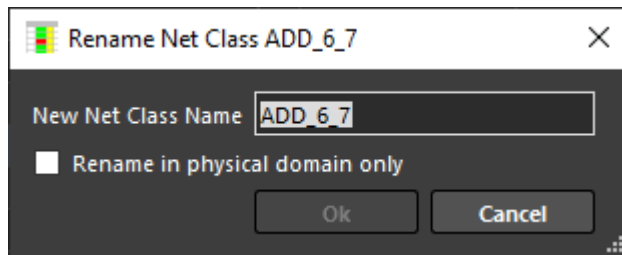
Allegro X Constraint Manager Reference

Objects Menu Commands

The Rename Net Class dialog appears:



Net Class being renamed
in the Spacing domain.



Net Class being renamed
in the Physical domain.

3. Specify a new name for the Net Class.
4. Select the *Rename in spacing and same net spacing domains only* option if you want to rename the Net Class only in the Spacing and Same Net Spacing domains, such that the Net Class name is retained in the Physical domain.

OR

Select the *Rename in physical domain only* option if you want to rename the Net Class only in the Physical domain, such that the Net Class name is retained in the Spacing and Same Net Spacing domains.

5. Click Yes.

Constraint Manager deletes the Net Class.

Objects – Clear all Xnet renames

The *Clear all Xnet renames* command clears all the renames for the Xnets in the design and regenerate the Xnet name that follows new Xnet naming algorithm.

In pre-16.6 releases, the Xnets were named in lexicographical order. In 16.6, the new logic for calculating Xnet names is introduced that uses:

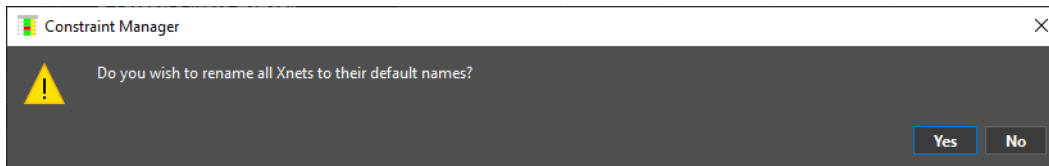
- Top-level nets for naming Xnet.
- In case of driver/receiver, the net connected to 0 driver pin and least number (≥ 1) of receiver pins is used to name the Xnet.

Note: This command is available only when the Constraint Manager is launched from logic design tools.

Procedure

1. Choose *Objects – Clear all Xnet renames*.

The confirmation dialog box is displayed.



2. Click *Yes*.

The command clears all the Xnet renames and regenerate the new Xnet name. You can see all the details in a report log that is displayed.

```
XNet "XNET_BUS1A<0>" has been renamed to "XNET_BUS1_TOP<0>".  
XNet "XNET_BUS1A<2>" has been renamed to "XNET_BUS1_TOP<2>".  
  
2 XNets were cleared.
```

Objects – Rename Xnets to highest level in hierarchy

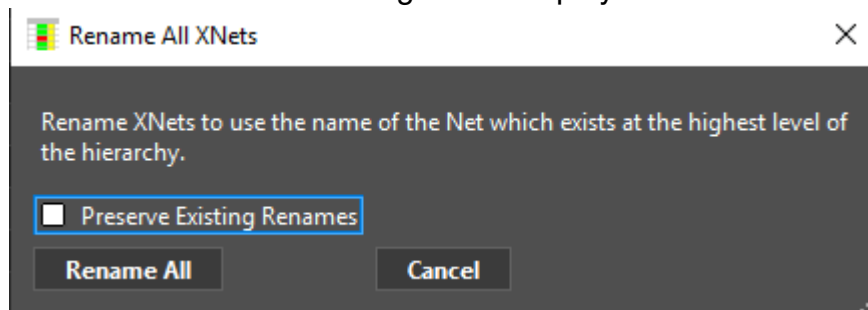
This command renames the net names using the current level of hierarchy, and not from net names from the lower blocks in the hierarchy. Use this command to uprev old designs to the new Xnet naming algorithm that uses highest-level net names.

Note: This command is available only when the Constraint Manager is launched from front-end tools.

Procedure

1. Choose *Objects – Rename Xnets to highest level in hierarchy*

.The confirmation dialog box is displayed.



2. Choose *Preserve Existing Renames* option to restore Xnet renames in the design.
3. Click *Yes* to rename all the Xnets according to new Xnet naming algorithm.

Allegro X Constraint Manager Reference

Objects Menu Commands

The command force rename of all the xnets in the design. A report log is displayed when the command is over.

```
XNet "TOP_XNET_BUS1<0>" has been renamed to "TOP_XNET_BUS1A<0>".
XNet "TOP_XNET_BUS1<1>" has been renamed to "TOP_XNET_BUS1A<1>".
XNet "TOP_XNET_BUS1<2>" has been renamed to "TOP_XNET_BUS1A<2>".
XNet "TOP_XNET_BUS1<3>" has been renamed to "TOP_XNET_BUS1A<3>".
XNet "TOP_XNET_L1" has been renamed to "TOP_XNET_L1A".
XNet "TOP_XNET_L2" has been renamed to "TOP_XNET_L2A".
XNet "MID_XNET_BUS1<0>" has been renamed to "MID_XNET_BUS1A<0>".
XNet "MID_XNET_BUS1<1>" has been renamed to "MID_XNET_BUS1A<1>".
XNet "MID_XNET_BUS1<2>" has been renamed to "MID_XNET_BUS1A<2>".
XNet "MID_XNET_BUS1<3>" has been renamed to "MID_XNET_BUS1A<3>".
XNet "MID_XNET_L1" has been renamed to "MID_XNET_L1A".
XNet "MID_XNET_L2" has been renamed to "MID_XNET_L2A".
XNet "XNET_BUS3<0>" has been renamed to "XNET_BUS3A<0>".
XNet "XNET_BUS3<1>" has been renamed to "XNET_BUS3A<1>".
XNet "XNET_BUS3<2>" has been renamed to "XNET_BUS3A<2>".
XNet "XNET_BUS3<3>" has been renamed to "XNET_BUS3A<3>".
XNet "XNET_L5" has been renamed to "XNET_L5A".
XNet "XNET_L6" has been renamed to "XNET_L6A".
XNet "LOW_LOCAL_XNET_BUS3<0>" has been renamed to
"LOW_LOCAL_XNET_BUS3A<0>".
```

Objects – Delete

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

Procedures

Deleting a bus, net group, match group, differential pair, or pin pair

1. In the *Objects* column, select a bus, net group, 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.

Deleting a Net Class

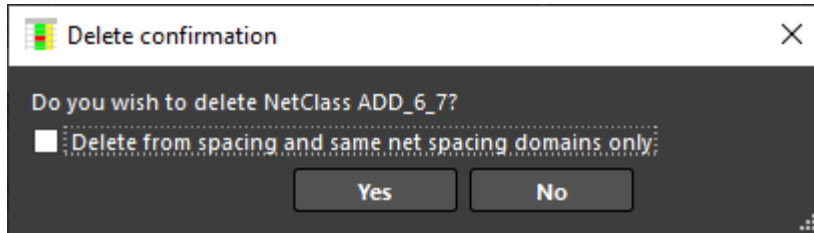
1. In the *Objects* column in the *Net Class-Class* folder in the Spacing or Physical domain, select a Net Class.
2. Do one of the following:

Allegro X Constraint Manager Reference

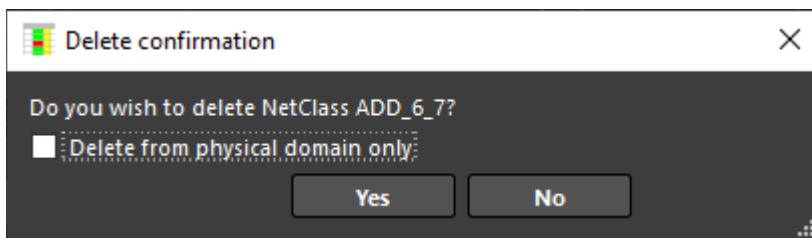
Objects Menu Commands

- ❑ Choose *Objects – Delete*.
- or -
- ❑ Right-click and choose *Delete* from the pop-up menu.

The Delete confirmation dialog appears



Net Class being deleted
in the Spacing domain.



Net Class being deleted
in the Physical domain.

3. Select the *Delete from spacing and same net spacing domains only* option if you want to delete the Net Class only from the Spacing and Same Net Spacing domains, such that the Net Class is retained in the Physical domain.

OR

Select the *Delete from physical domain only* option if you want to delete the Net Class only from the Physical domain, such that the Net Class is retained in the Spacing and Same Net Spacing domains.

4. Click *Yes*.

Constraint Manager deletes the Net Class.

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.

Important

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.

Allegro X Constraint Manager Reference

Objects Menu Commands



Tip

You can influence each pass by specifying the following. See

- ❑ [Mapping Modes](#) on page 162
- ❑ [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			

Allegro X Constraint Manager Reference

Objects Menu Commands

Pass 5	Pass 6	Pass 7
1. Refdes	1. Refdes	1. Pinuse (less restrictive)
2. Pin Number	2. Diff Pair Type	2. In the less-restrictive pinuse pass, any I/O- or connector-connector pin maps to any driver or receiver pin.
3. Diff Pair Type		3. 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 toConstraint Manager maps pins by this precedence and makes these passes See Pin Mapping Passes on page 161.
■ <i>Pinuse</i>	1. Pinuse and Buffer Model	3, 4, 7
■ <i>Refdes</i>	1. Refdes and Pin Number 2. Pinuse and Refdes	1, 2, 5, 6
■ <i>Pinuse and Refdes</i>	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 *Setup – Constraints* and click the *Wiring* tab) or in Constraint Manager (in the *Wiring* worksheet of the *Routing* workbook at the *Electrical CSet* domain). The mapping mode is stored with the topology file.

You can select one of the pre-defined mapping modes for the ECSet from the drop-down list. Following is the list of available mapping modes:

- *Pinuse*: Maps the pins of an ECSet to the XNet using the PINUSE setting.
- *Refdes*: Maps the pins of an ECSet to the XNet using the RefDes setting.
- *Pinuse and Refdes*: Employs both mapping techniques described above.
- *(Clear)*: No specified mapping mode.

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 Groups* 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.

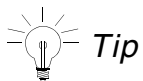
Procedures

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 net 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.
- or -
 - ☐ In the *Referenced Constraint Set* column, right-click and choose *Change*.
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*.

You can also go to a parent CSet of an object directly. For example, to go to a parent Physical CSet of an object, right-click the assigned Physical CSet in the *Referenced Physical CSet* column and choose *GoTo Physical CSet*.



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 net 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.

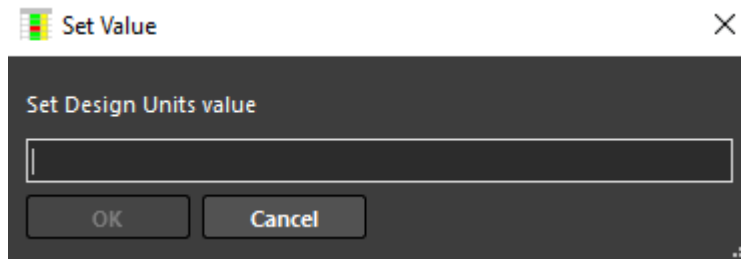
Objects – Change all design unit attributes

You can use this command to change the design units of all the constraint objects in the Physical, Spacing and Same Net Spacing domain.

Procedure

1. Choose the object to change the design unit.
2. Choose *Objects – Change all design unit attributes*.

The *Set Value* dialog box appears.



3. Specify the new design unit.
4. Press OK.

Note: You can enter column delimited values in the Set value dialog box for the CSets that are applied to the corresponding layers.

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.

Allegro X Constraint Manager Reference

Column Menu Commands

- 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*.

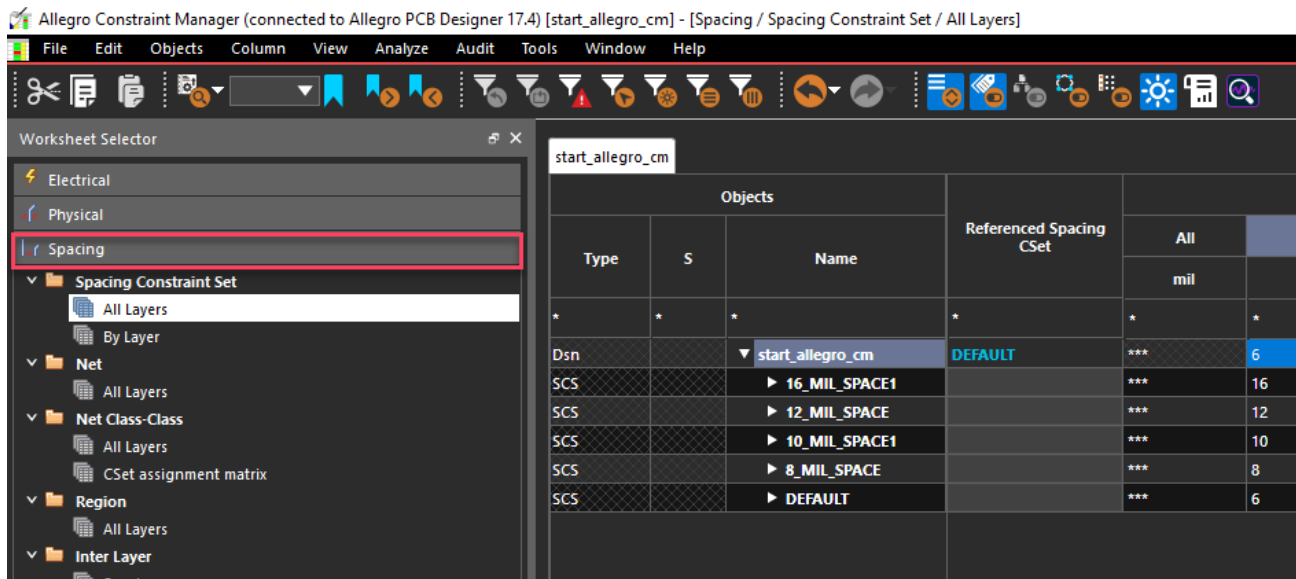
Allegro X Constraint Manager Reference

Column Menu Commands

Column – Analysis Modes

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

You can turn-on the analysis directly for any mode from the column header using pop-up menu. If the related analysis is off, column header is displayed in Yellow.



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 – View Options

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

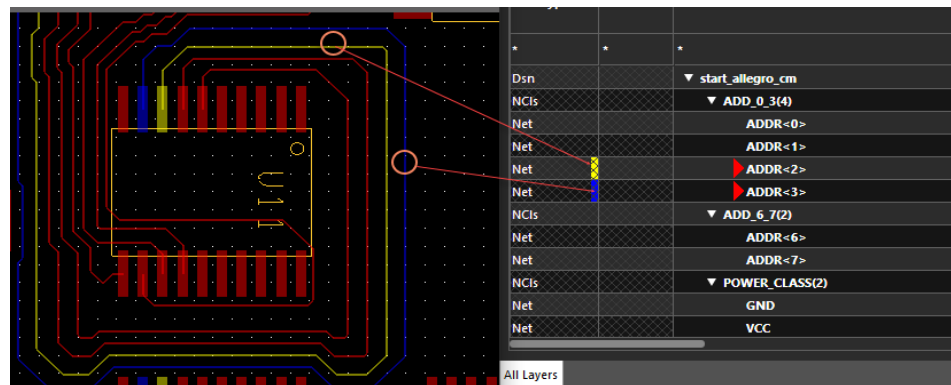
View Options Dialog Box

Use this field	To
<hr/>	
UI Options – Main Window	<p>Increase the available screen area for viewing the worksheets. You can hide the <i>tool bar</i>, <i>status bar</i>, <i>worksheet selector</i>, and <i>color legend</i>. Alternatively, you can un-dock, and reposition, the color legend.</p> <p>You can also filters Constraint Manager worksheet view by Design Instance or Block Levels on a schematic.</p> <p>Note: This option is available only when Constraint Manager is invoked from schematic.</p> <div><p>The status bar provides key feedback on many operations. We recommend that you do not hide it.</p></div>
UI Options – Workbooks	
<i>Row Numbers</i>	Show row numbers.
<i>Cell Selection Highlights</i>	Show a background tint applied to the row and column header of the selected cell.
<i>Alternative rows highlight</i>	Show a background tint applied to the alternative rows.

Allegro X Constraint Manager Reference

View Menu Commands

Use this field	To
<i>Application Select</i>	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.
<i>Single-Click Editing</i>	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.
<i>Show Values Inherited from Design</i>	Clear this check box to stop showing values inherited from the <i>Design</i> level but not from other structural parents (Net Classes, Buses, etc.).
<i>Show Application Color</i>	Constraint Manager shows color assigned to the object in the application next to the type of the object. For example, in Allegro right-click on a net and select <i>Net – Assign</i> color from the pop-up menu and then select a color from the Assign Color dialog box. When a color is assigned, it is displayed in Constraint Manager.



Allegro X Constraint Manager Reference


View Menu Commands

Use this field	To
<i>Show number of Group members</i>	If checked (the default mode), displays the number of group members in parentheses after each group name in the Objects column.
<i>Show dialog for Numerical Filters</i>	If checked (the default mode), enables the Numerical Filter dialog when clicking on the top filter row and allows to use relational operators to the filter string. For example, equal to, less than, less than or equal to, greater than, greater than or equal to, and in range.
<i>Use pretty print</i>	When selected, displays the cell content in a format which is easy-to-read and understand.
<i>Font</i>	Specify a font family for text in workbooks and worksheets.
<i>Names</i>	Specify net display in physical or logical (canonical) format.
<i>Restore default settings</i>	<p>Use Constraint Manager's default settings on start up.</p> <p>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.</p> <p>Enabling this option also restores information messages disabled in the earlier sessions.</p>

UI Options –Views Synchronization

Allegro X Constraint Manager Reference

View Menu Commands

Use this field	To
<i>View Synchronization</i>	<p>Synchronize the object display across all rows in all worksheets.</p> <p>When enabled, <i>Synchronize Rows</i> recognizes the row modifications that you make in the <i>active</i> worksheet — including filters, scroll position, and object expansion — and synchronizes all worksheets that you access with these same display settings.</p> <p>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.</p> <p>Note: Depending on the size of the design, Constraint Manager may defer row synchronization until you click a new worksheet to make it active.</p> <p>Constraint Manager preserves the state of <i>Synchronize Rows</i> (ON by default) when you exit the tool.</p> <p> <i>Tip</i></p> <p>When <i>Synchronize Rows</i> is ON globally, you can disable row synchronization on an individual worksheet by clicking in the <i>Objects</i> column label, then right-clicking and choosing <i>Stay Synchronized</i> (uncheck the option) from the pop-up menu.</p>
<i>Top Row</i>	Synchronize the object display across top rows in all worksheets.
<i>Objects Expand/Collapse</i>	Choose to expand or collapse the objects across all rows in all worksheets.
<i>Columns sorting order</i>	Choose to enable sorting across all columns in all worksheets.
UI Options – Colors	
<i>Use colors</i>	<p>Turn off cell colors.</p> <p>At times, it may be easier on your eyes to examine cells without colors. Simply uncheck the <i>use colors</i> checkbox to turn off cell colors.</p>


Allegro X Constraint Manager Reference

View Menu Commands

Use this field	To
<i>Color Palette</i>	Customize colors or use Constraint Manager's default colors. <ul style="list-style-type: none">■ Default■ Custom
<i>Pass</i>	Specify the cell color for resulting analysis values (actuals) that meet the specified constraint setting.
<i>Fail</i>	Specify the cell color for resulting analysis values (actuals) that do <i>not</i> meet the specified constraint setting.
<i>Analysis/Formulas error</i>	<p>Specify the cell color when analysis cannot be performed or if there is an error in a calculated formula. The <i>status</i> window displays the cause of the error.</p> <p>Possible failures may result when trying to analyze with:</p> <ul style="list-style-type: none">■ <i>only</i> 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
<i>Directly set</i>	Specify the cell color of an Electrical CSet assignment or a net-related constraint override set on an object
<i>Out of Date/Warning</i>	Specify the color of an out of date cell.
<i>Cell Selections</i>	Specify the color of cell selections. Use this with the Cell Selection Highlights check box, located in the Workbooks section of this dialog box.
<i>Read-Only</i>	Specify the cross hatch color for read only cells.
<i>Formula indicator</i>	Specify the color for a column containing a formula.

Allegro X Constraint Manager Reference

View Menu Commands

Use this field	To
<i>Conflict</i>	<p>Specify the color to mark constraint values with a conflict, that is if there are different values for the same constraint coming from different levels of hierarchy.</p> <p>When nets are aliased in a schematic tool, their constraint values are merged. In some cases, both nets will have different values for the same constraint, which is considered a conflict. The conflict resolution feature provides a method for reporting and resolving these conflicting values. One of the indicators that a conflict exists is that the constraint value is shown in the <i>Conflict</i> color in the Constraint Manager worksheet.</p>
<i>Alt rows background</i>	<p>Specify the color of the alternative rows for improved readability</p>
<i>Reset to default colors</i>	<p>Restore the default cell color scheme:</p> <ul style="list-style-type: none">■ Pass = green■ Fail = red■ Analysis error = yellow■ Directly set = blue <p>If you have modified color options (pass, fail, analysis error, directly set), these settings will be recalled when you uncheck <i>use defaults</i>.</p>
UI Options – Tabbed View	
<i>Use Tabbed View (require Views re-open)</i>	<p>Enable tabbed selection of participating designs in a system.</p> <div> <i>Tip</i><p>Once visible, you can right-click on a tab to change its orientation or to synchronize tabs across worksheets.</p></div>
<i>Synchronize Tabs</i>	<p>Synchronize all open workbooks associated with the design represented by the active tab.</p>
<i>Advanced Tabbed View</i>	<p>Provides better control over the Design tabs. This is useful in the front-end designs with hierarchical designs containing many unique designs, such that the tabs shrink so much that they are not readable.</p>

Allegro X Constraint Manager Reference

View Menu Commands

Use this field	To
<i>Tabs on . . .</i>	<p>Specify where (top, left, bottom, right) to display tabs in Constraint Manager by selecting one of the following options:</p> <ul style="list-style-type: none">■ Top■ Bottom■ Left■ Right <p>These options are also available form the right-click pop-up menu on all the tabs.</p>

UI Options – Tooltips

<i>Show Tool Tips</i>	Show tool tips.
<i>Tooltip display time (sec)</i>	Show tool tips display time in seconds.

UI Options – Unsupported

<i>Dynamic Group re-positioning in hierarchy based on Group's members</i>	<p>Choose to allow Group objects to bubble to the highest level of all group members.</p> <p>If all members of a group are from a higher level, the group displays at that higher level even if the Group Object was originally created at a lower level.</p> <p>For example, if a Net Class is created on a lower level which has all of its net members from a higher level then the Net Class will bubble up to the higher level.</p> <p>Note: This option is available when Constraint Manager is invoked from schematic. This option is disabled by default.</p>
---	--

Allegro X Constraint Manager Reference

View Menu Commands

Use this field	To
<i>Show all Pin Pairs in all worksheets. Pin Pairs which have constraints in a specific worksheet(e.g., Impedance) will be displayed in other worksheets(e.g., Propagation delay). This improves performance of worksheet open.</i>	Choose to display Pin Pairs in all worksheets.
<i>Object Type Dividers</i>	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.

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 – Display Priority

Use this command to control the display of individual columns by setting the display priority. You can display only those columns that are of interest and hide the irrelevant columns.

The symbols >>(show more) or <<(show less) are added to the column headers for the columns that have different priorities. You can double-click the column header to display all or few columns.

To set the priority select the column header, right-click and choose Display Priority. The priority can be set as high, medium, or low. By default priority for all the columns is set to medium.

- High = display column always irrespective of column header is expanded or collapsed
- Low = hide column always when column header is expanded
- Medium = display column when column header is expanded

The display settings are saved in the registry for the current user.

Procedure

Setting Column Priority to High

1. Open *Spacing Constraint Set – All Layer Worksheet*.
2. Choose *Line to >>* column header.
3. Double-click to expand the column.

Or

Hover over the column header. Right-click and chose *Show More*.

The complete set of all *Line To* constraints are displayed.

4. Click to choose column for *Line* constraint.
5. Right-click and choose *Display Priority – High*.

The high priority ensures that the *Line* column is displayed whether the *Line to* column is expanded or collapsed.

6. Double-click *Line to<<* column header to collapse all columns.

All the constraints are hidden except *Line*.

Setting Column Priority to Low

1. Double-click *Line to>>* column header to expand all columns.
2. Click to choose column for *Bond Finger* constraint.
3. Right-click and choose *Display Priority – Low*.

The low priority ensures that the *Bond Finger* column is hidden whether the *Line to* column is expanded or collapsed.

4. Double-click *Line to <<* column header to collapse all columns and again double-click to expand all columns.

All the constraints are displayed except *Bond Finger*.

5. Double-click *Line to>>* column header again to see the low priority *Bond Finger* column.

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*.

Allegro X Constraint Manager Reference

Audit Menu Commands

3. Click *Save*.

Example

```
CONSTRAINT AUDIT REPORT
*****

DESIGN: start_allegro_cm
-----

NET VIOLATIONS

DOUT2:
  Overrides:
    PROPAGATION_DELAY_MIN
    PROPAGATION_DELAY_MAX
    PROPAGATION_DELAY_PATH_TYPE

DOUT1:
  Overrides:
    PROPAGATION_DELAY_MIN
    PROPAGATION_DELAY_MAX
    PROPAGATION_DELAY_PATH_TYPE

CIN:
  Overrides:
    NO_IO_CHECK

D<4>:
  Overrides:
    NO_IO_CHECK
```

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

Allegro X Constraint Manager Reference

Audit Menu Commands

Procedure

1. Choose *Audit – Obsolete Objects*.

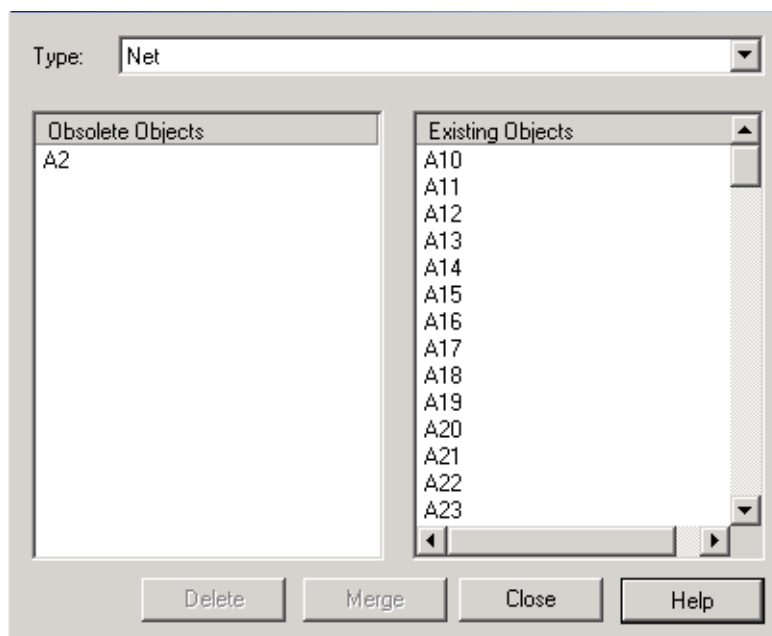
If there are not any obsolete objects, a confirming message appears.

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 – Invalid Objects

Use this command to generate a report of objects that must be reconciled between Constraint Manager and the PCB-, package-, or schematic-databases.

Invalid objects appear due to design corruption. There are multiple type of invalid objects in a design. This command however, identifies only following object types:

- diff pairs with number of members other than 2
- empty buses
- empty match groups

Constraint Manager displays a *No Invalid Objects* message as appropriate.

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

Audit Invalid Objects Dialog Box

Use this field	To
<i>Type</i>	Filter on an object type <ul style="list-style-type: none">■ diff pairs with number of members other than 2■ empty buses■ empty match groups
<i>Invalid Objects</i>	List all the invalid objects in the Allegro or Design Entry HDL database, yet exist in Constraint Manager.
<i>Existing Objects</i>	List all objects that exist in the PCB-, package-, or schematic-database, and in Constraint Manager.
<i>Delete</i>	Remove objects, listed in the <i>invalid objects</i> list, from the PCB-, package-, or schematic-database.
<i>Merge</i>	Assign all properties and constraints from the object selected in the <i>invalid object</i> list to the object selected in the <i>existing object</i> list (properties and constraints on the existing object will not be overwritten).

Procedure

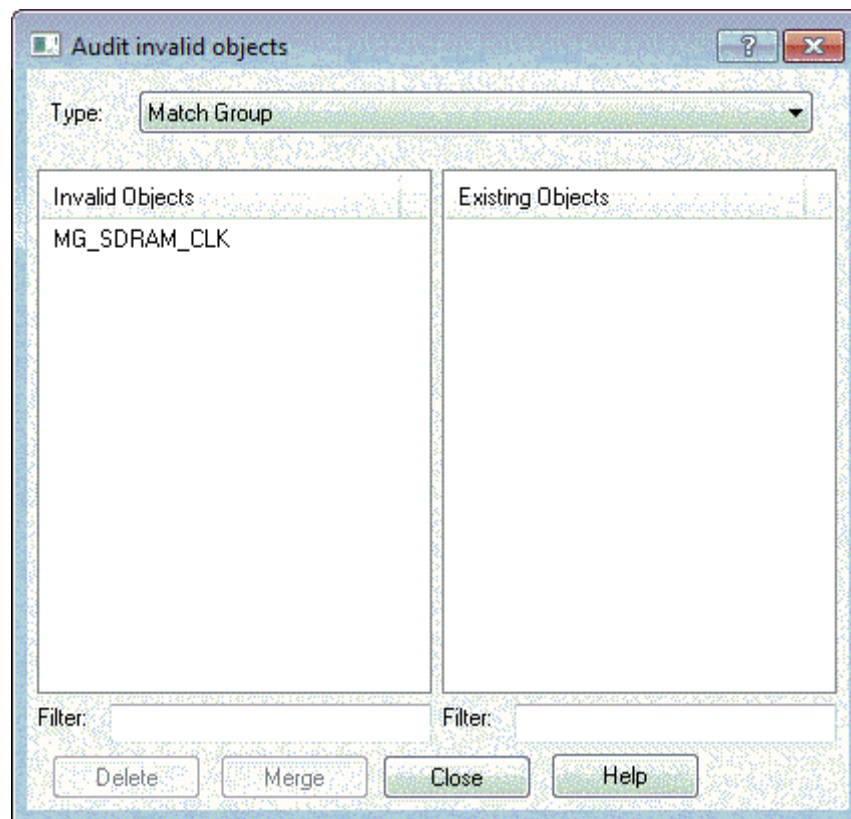
1. Choose *Audit – Invalid Objects*.

If there are no invalid objects, a confirming message appears.

If there are invalid objects, the *Audit invalid objects* dialog box appears.

2. Click the drop-down menu to filter on an object type (Bus, Match Group, Diff pair).
3. Delete or Merge the invalid objects.
4. Click *Close*.

Example



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 160 for more information about referencing ECSets and topology mapping criteria.

Apply Status

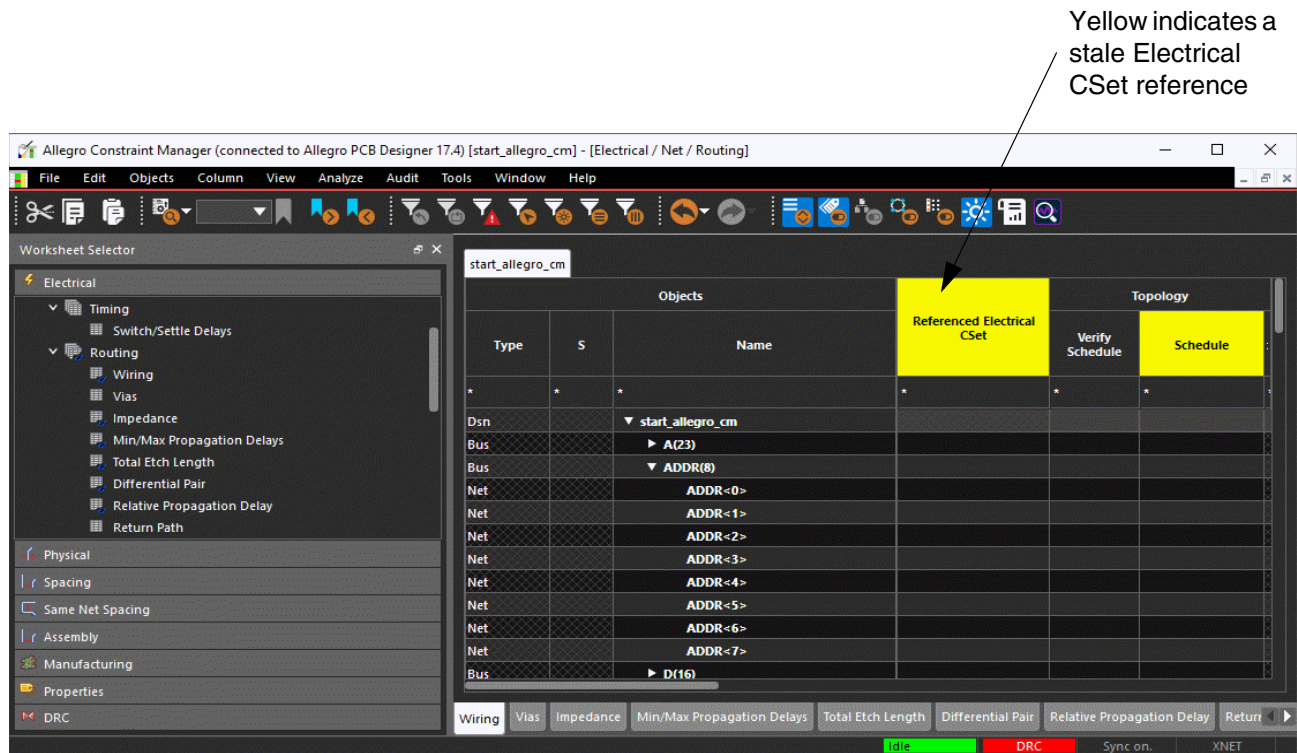
The *Referenced Electrical CSet* column of any net-related worksheet is colored yellow when:

- Constraint Manager is running with Design Entry HDL

Allegro X Constraint Manager Reference

Audit Menu Commands

- 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*.

Allegro X Constraint Manager Reference

Audit Menu Commands

3. Click *Save*.
4. Examine the audit report.
5. Resolve topology mismatches.

See [Objects – Constraint Set References](#) on page 160 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.

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.

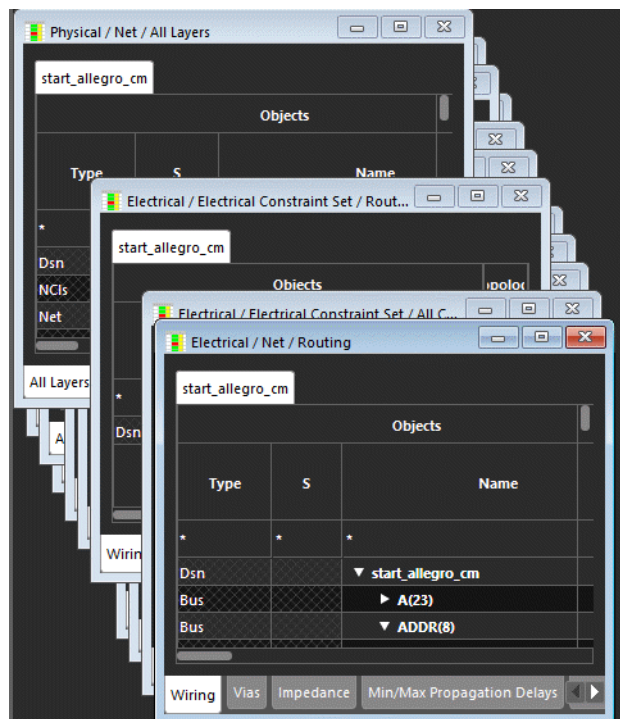
Window – Cascade

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.



Allegro X Constraint Manager Reference

Window Menu Commands

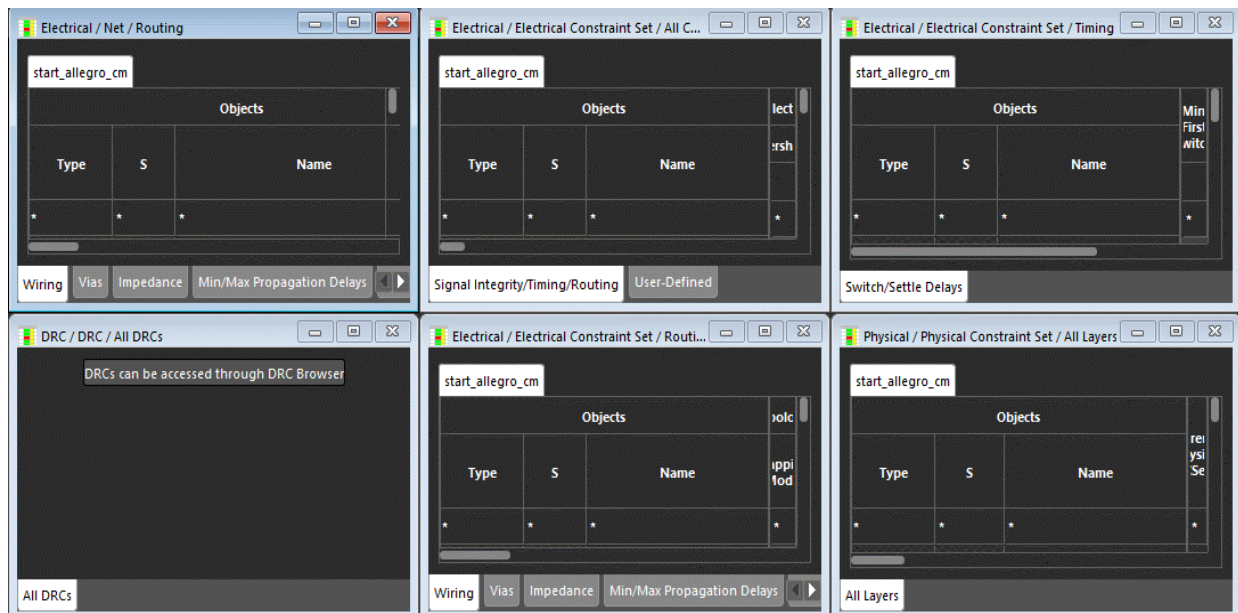
Window – Tile

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 – Previous Recent View

Use this command to open the previous view in all worksheets.

Window – Next Recent View

Use this command to open the next recent view 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.

Allegro X Constraint Manager Reference

Window Menu Commands

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.



When you change values in a worksheet, you may have to re-analyze (choose *Analyze – Analyze*) 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.

Allegro X Constraint Manager Reference

Analyze Menu Commands

Reflection

Type

Specifies the type of simulation to use for getting reflection results. The default is *Reflection*.

Measurement

Specifies the measurements of simulation to use for getting reflection results. The default is *Pulse*.

Choosing *Custom* option enables *Assign* button, which opens Electrical Properties worksheet to define parameter values for nets and Xnets at the board level. If not specified, default values are used for simulation, which are as follows:

- ClockFrequency: 50MHz
- ClockDutyCycle: 0.5
- CycleCount: 1
- Offset: 0ns
- Jitter: 0
- BitPattern: 01

Allegro X Constraint Manager Reference

Analyze Menu Commands

Crosstalk

Aggressor Switch mode

Specifies the aggressor switching mode (Odd or Even). The default is *Odd*.

Aggressor Driver

Specifies the number of crosstalk simulations required for generating actuals for the `MAX_PEAK_XTALK` constraint. When you choose *All*, 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.

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 type and measurement
 - ☐ Crosstalk simulation parameters
3. Optionally, click *Save Waveforms*.
4. Optionally, click preferences to set default simulation parameters.
5. Click *OK*.

Allegro X Constraint Manager Reference

Analyze Menu Commands

Analyze – Analysis Modes

Procedures

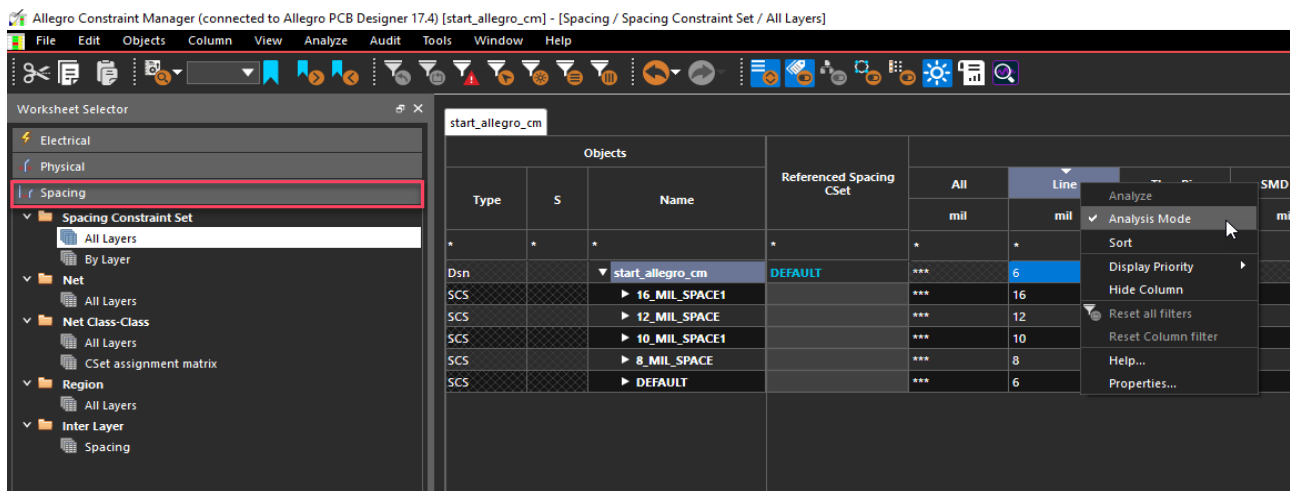
Use this command to enable individual design rule checks (DRCs), associated options, and custom measurements.

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 [Design for Fabrication in Analysis Modes Constraints Reference](#) guide for detailed information on individual constraints.

You can turn-on the analysis directly for any mode from the column header using pop-up menu. If the related analysis is off, column header is displayed in Yellow.



Analysis Modes Dialog Box

The *Analysis Modes* dialog box contains the modes and options for the *Design*, *Electrical*, *Physical*, *Spacing*, *Same Net Spacing*, *Assembly*, and *Manufacturing* checks.

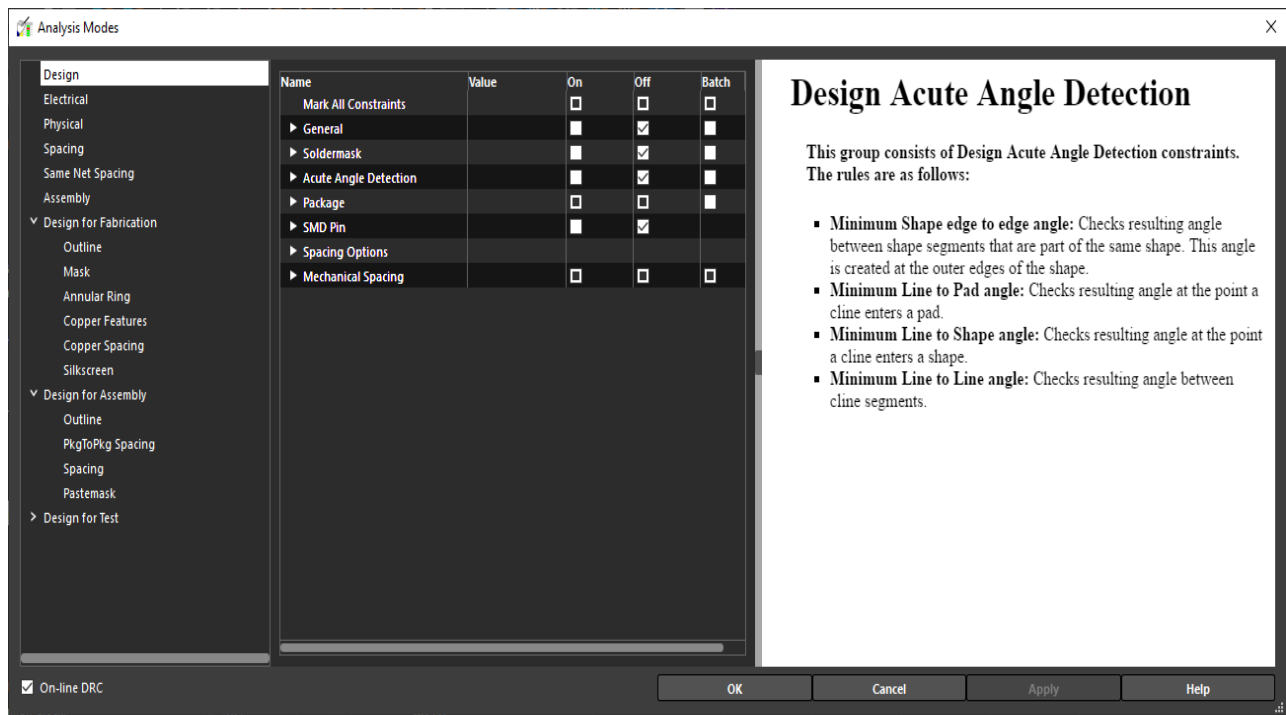


Figure 8-3 Analysis Modes Dialog Box

Common Controls

<i>Value</i>	Displays the value of the constraint.
<i>On</i>	Interactively checks for design rule violations. <ul style="list-style-type: none">■ 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 <i>Tools –Report</i>)■ A DRC bowtie marker appears in the board layout canvas
<i>Off</i>	Disables design rule checking to improve system performance.

Allegro X Constraint Manager Reference

Analyze Menu Commands

Batch

Checks for design rule violations only when a batch command (electrical-only) is performed in the layout tool.

On-line DRC

When unchecked, 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, and APD performs the equivalent function.

- When you turn DRC back on (check the *Online 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.

Allegro X Constraint Manager Reference

Analyze Menu Commands

Design Modes and Options

Specifies and controls design rule checks for plane, mechanical hole, testpoint, soldermask, acute angle detection, package, SMD pin, and spacing parameters.

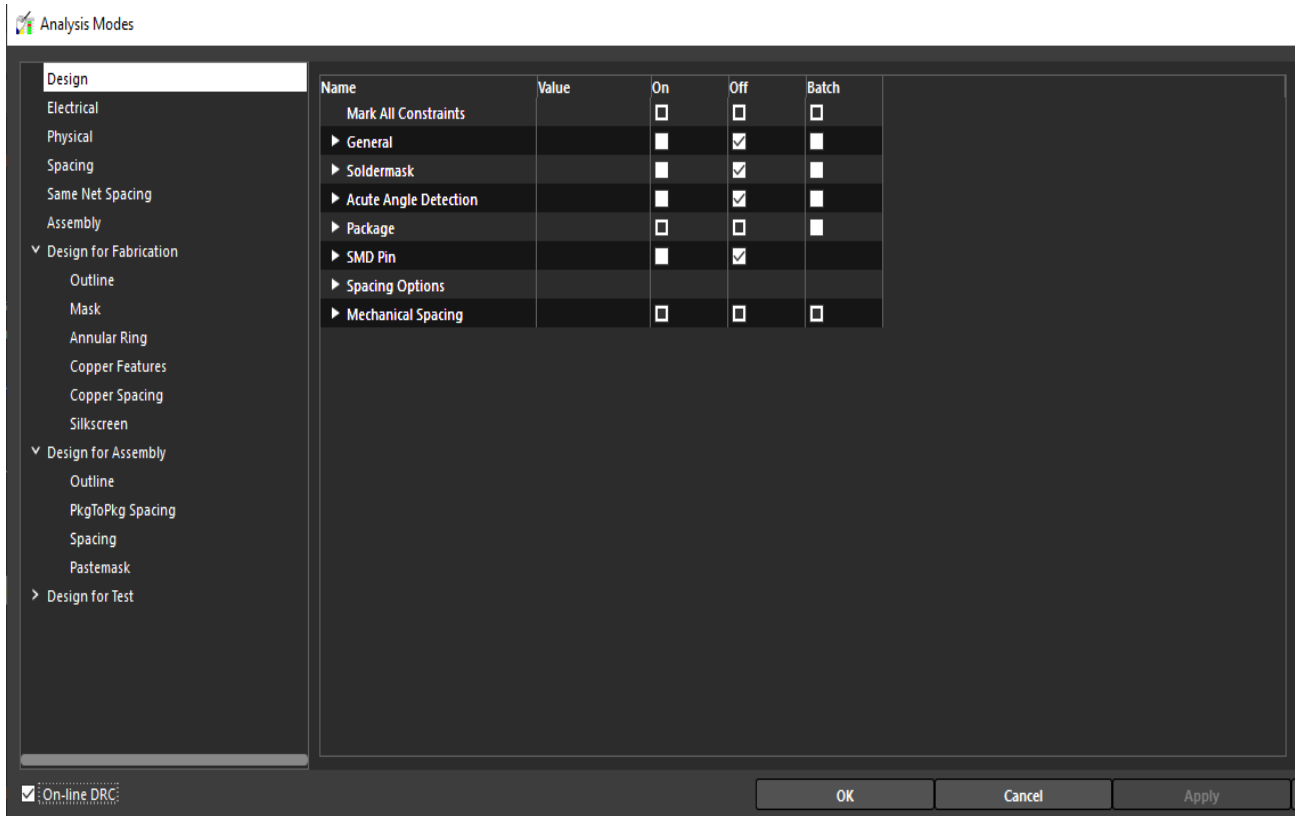


Figure 8-4 Design Modes and Options

Allegro X Constraint Manager Reference

Analyze Menu Commands

General

Define design rule checks for plane, mechanical hole, and testpoint and trigger them based on changes to the layout.

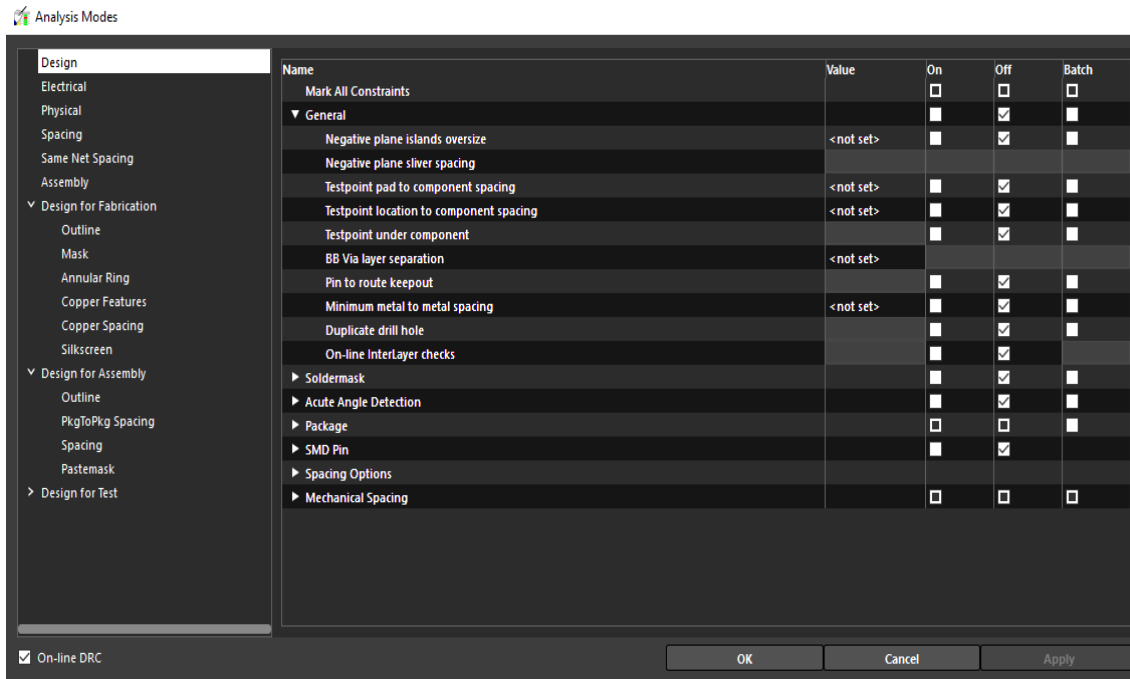


Figure 8-5 General Design Modes

Allegro X Constraint Manager Reference

Analyze Menu Commands

Soldermask

Controls whether specific design rule checks for soldermask are triggered based on changes to the layout.

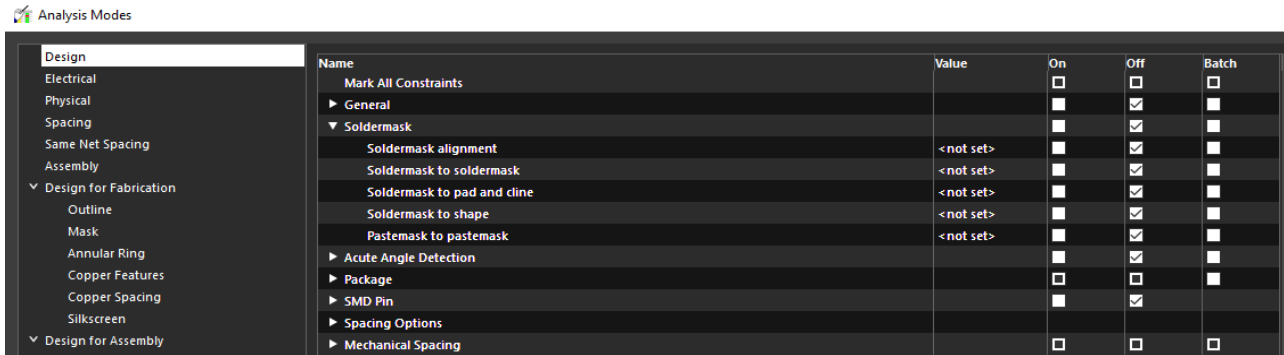


Figure 8-6 Design Modes for Soldermask

Acute Angle Detection

Controls angle based checks at the junction of two elements (pad edge, shape edge, and line) that form an angle.

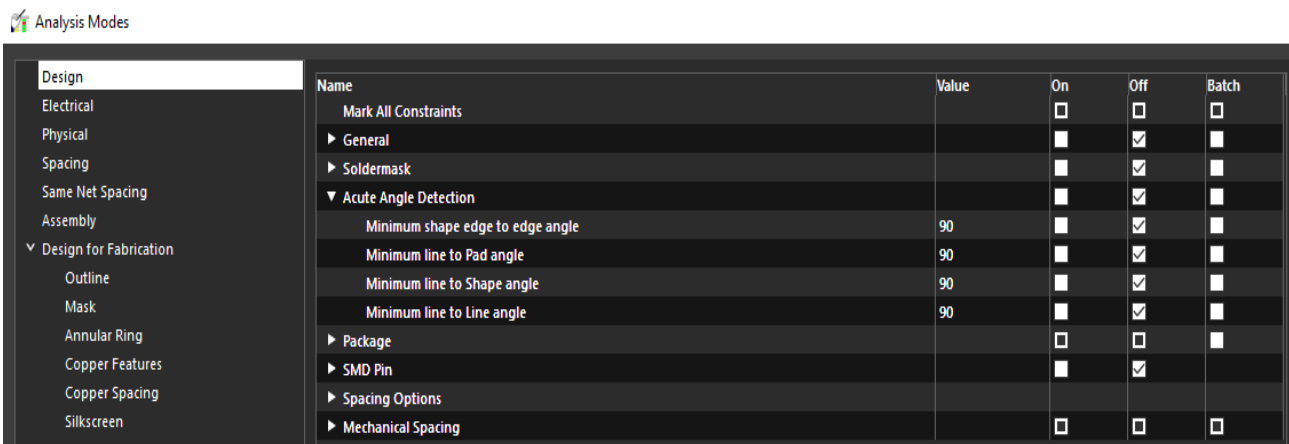


Figure 8-7 Design Modes for Acute Angle Detection

Refer to the [Design for Fabrication](#) in *Analysis Modes Constraints Reference* guide for detailed information on individual constraints.

Allegro X Constraint Manager Reference

Analyze Menu Commands

Package

Controls whether specific design rule checks for package are triggered based on changes to the layout.

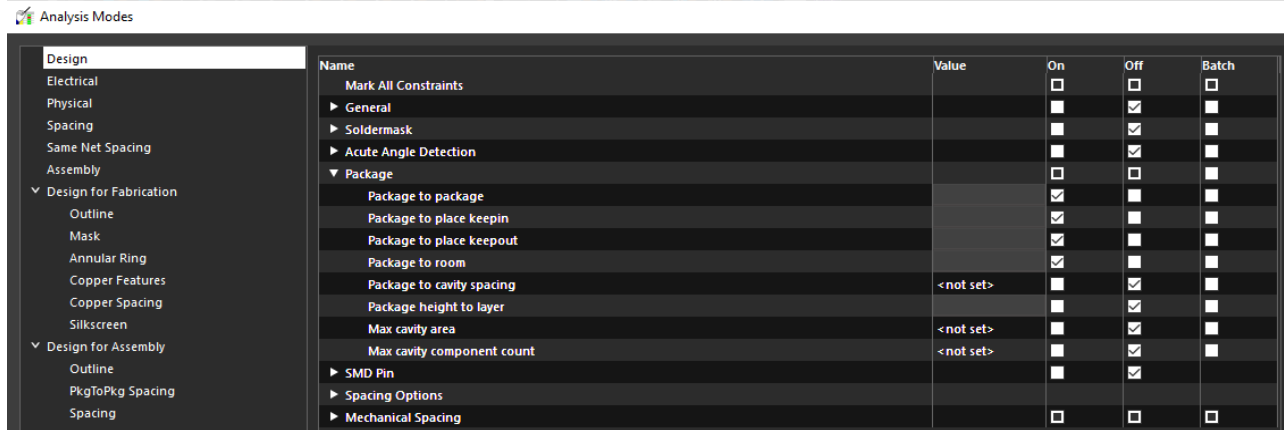


Figure 8-8 Design Modes for Package

SMD Pin

Controls whether specific design rule checks for SMD pins are triggered based on changes to the layout.

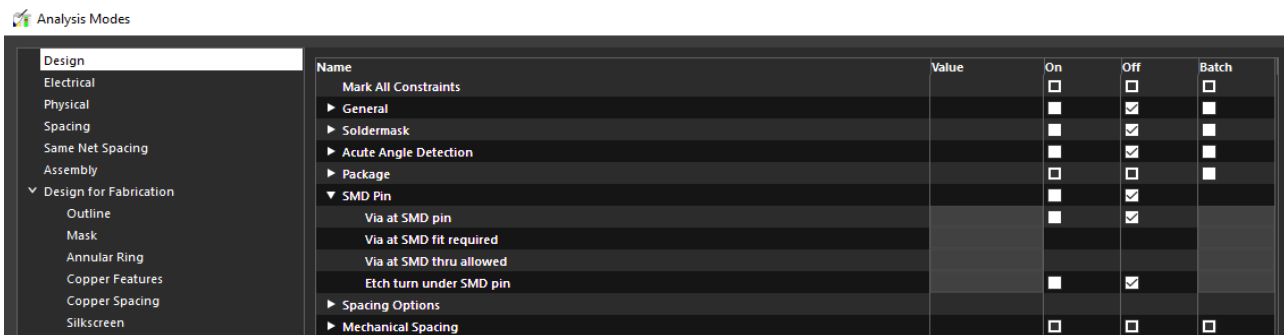


Figure 8-9 Design Modes for SMD Pins

Allegro X Constraint Manager Reference

Analyze Menu Commands

Spacing Options

Controls whether specific design rule checks for spacing options are triggered based on changes to the layout.

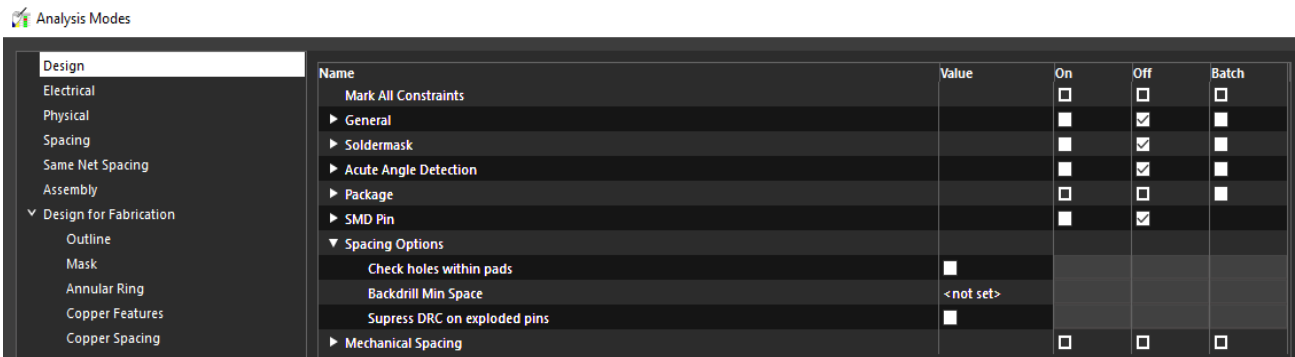


Figure 8-10 Design Modes for Spacing Options

Refer to the [Design for Fabrication](#) in *Analysis Modes Constraints Reference* guide for detailed information on individual constraints.

Electrical Modes and Options

Use the *Electrical* page to selectively enable the inclusion of etch, pin delay, or via delay in length calculations.

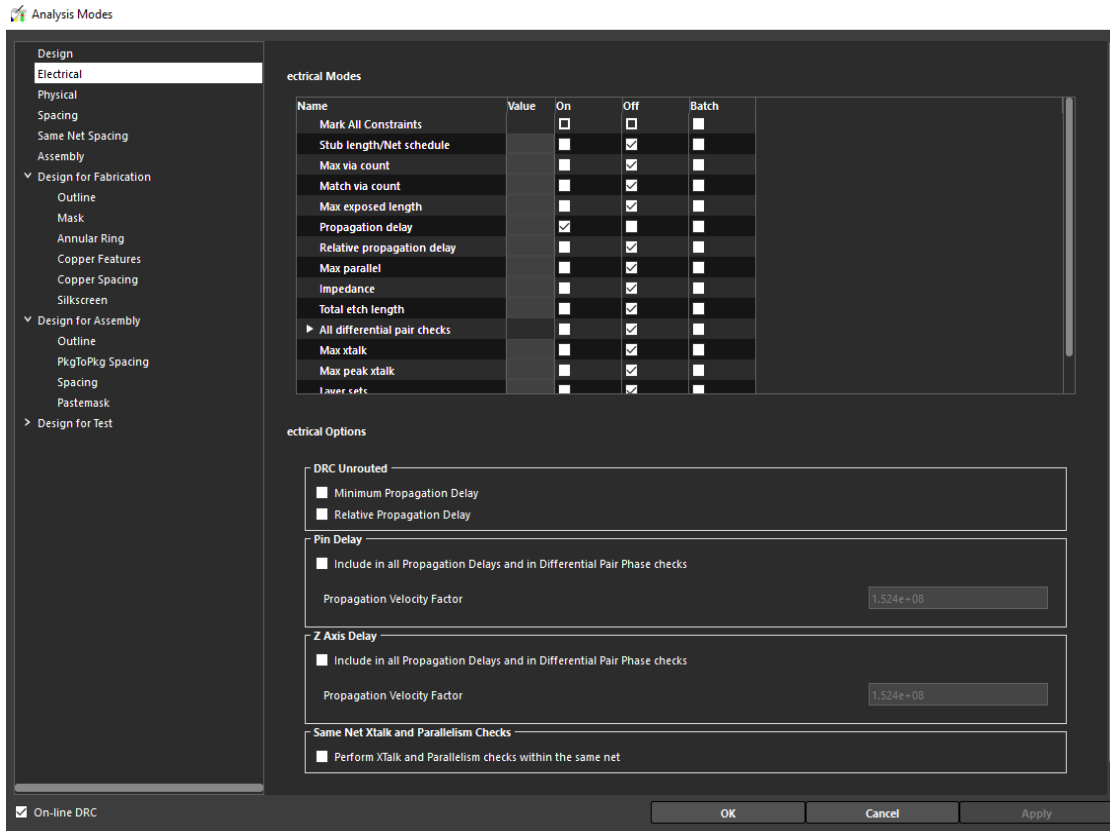


Figure 8-11 Electrical Modes and Options

Electrical Options

■ DRC Unrouted

When *DRC unrouted* is checked—for either *Minimum 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.

- Pin delay options is visible when High-Speed option is enabled in Allegro PCB Editor. *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.

Z Axis options is visible when High-Speed option is enabled in Allegro PCB Editor.

- *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.

Same Net Xtalk and Parallelism Checks options are visible when High-Speed option is enabled in Allegro PCB Editor.

- *Differential Pair Constraints*

Preserves the constraint resolution (precedence) of differential pair overrides. When enabled, differential pair overrides have a higher precedence than constraint regions.

Allegro X Constraint Manager Reference

Analyze Menu Commands

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 recommended for new designs. You should review your constraints and eliminate the need for this option.

Differential Pair Constraints options are visible when High-Speed option is enabled in Allegro PCB Editor.

Refer to the [Design for Fabrication in Analysis Modes Constraints Reference](#) guide for detailed information on individual constraints.

The *All Differential pair checks* option is not a constraint and is discussed below.

All differential pair checks

The *All differential pair checks* is an electrical DRC mode which controls all differential pair constraints. Setting a DRC mode for this option turns design rule checking on or off for the differential pair rules, listed on the DiffPair Values tab. Specifically, the differential pair checks are for phase control, uncoupled length, and minimum line spacing between the two nets.

Allegro X Constraint Manager Reference

Analyze Menu Commands

Tips for Differential Pairs Checks

- Displaying constraints (In the Show Constraints window launched from *Display – Constraint* in the layout tool) between the two objects in question can be helpful in telling you the constraint resolution. See example below:

Constraint Hierarchy		
	Element 1	Element 2
Description	Pin "U11.14"	Net "Addr"
Location	(450 975)	(750 1175)
NetClass	ADD 6 7	ADD 0 3
NetGroup		
Bus	ADDR	ADDR
DiffPair		
XNet		
Net	ADDR<7>	ADDR<3>
PinPair		
NetClass-NetClass	ADD 6 7:ADD 0 3	ADD 6 7:ADD 0 3
Region		
Routing Section		

Resolved Spacing Constraints			
Resolved Level	Source Name	Constraint	Value
Unresolved	None	Package to Package Spacing	0 MIL
NetClass-NetClass	16 MIL SPACE1	Thru Pin to Thru Pin Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Thru Pin to SMD Pin Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Thru Pin to Thru Via Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Line to Thru Pin Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Thru Pin to Shape Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	SMD Pin to SMD Pin Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	SMD Pin to Thru Via Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Line to SMD Pin Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Shape to SMD Pin Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Thru Via to Thru Via Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Line to Thru Via Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Shape to Thru Via Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Line to Line Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Line to Shape Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Shape to Shape Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Blind/Buried Via to Blind/Buried Via Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Blind/Buried Via to Thru Pin Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Blind/Buried Via to SMD Pin Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Blind/Buried Via to Thru Via Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Blind/Buried Via to Line Spacing	16 MIL
NetClass-NetClass	16 MIL SPACE1	Blind/Buried Via to Shape Spacing	16 MIL

Figure 8-12 Show Constraints Window

- An ECSet reference takes precedence over all physical constraints, regardless of where the PCSet is referenced. This includes the PCSet being directly referenced by the differential pair, inherited from a Net Class, or from the differential pair being routed through a region that references the PCSet. The rule does not change whether you are in the primary width mode or the neck mode.

Allegro X Constraint Manager Reference

Analyze Menu Commands

- In 16.2 and later, if you see `Line To Line DRCs` between differential pair objects, you may need to specify a non-zero value for `Min Line Spacing` in the `PCSet`. A zero value invokes the `Line To Line` check. To verify this condition, display the `Show Constraint` window to view constraints between the two clines with the DRC and check that the `Min Line Spacing` value is non-zero.
- When the `Diff Pair Gap` constraint is less than the `Line To Line` spacing constraint, the `Min Line Spacing` constraint must be specified. The `Min Line Spacing` constraint is for differential pairs only and takes precedence over the `Line To Line` spacing constraint. If this constraint is not specified, the `Line To Line` spacing constraint is enforced. The `Min Line Spacing` value should be less than the `Neck Gap` minus the `Tolerance`.
- In releases prior to 16.2, the `Min Line Spacing` constraint can only be specified in the Electrical worksheets of Constraint Manager (`Min Line Spacing` column in the *Routing – Differential Pair* worksheet). In 16.2, the constraint was also added to the Physical Constraint worksheets. As with all physical constraints, the constraint value can vary by layer.

Custom Measurements

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

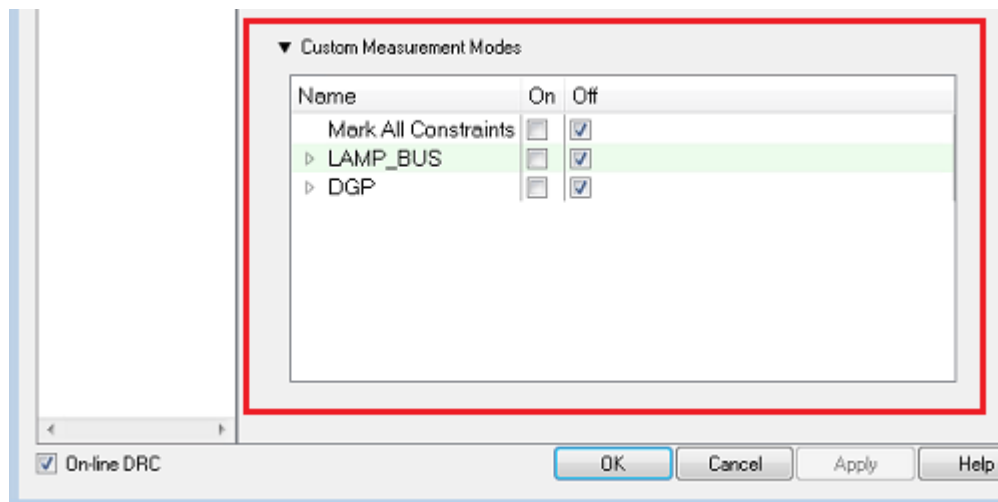


Figure 8-13 Custom Measurement Modes



Tip

You define custom measurements in SigXplorer Expert.

Allegro X Constraint Manager Reference

Analyze Menu Commands

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).

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.

Physical Modes

Controls whether specific design rule checks for physical modes are triggered based on changes to the layout.

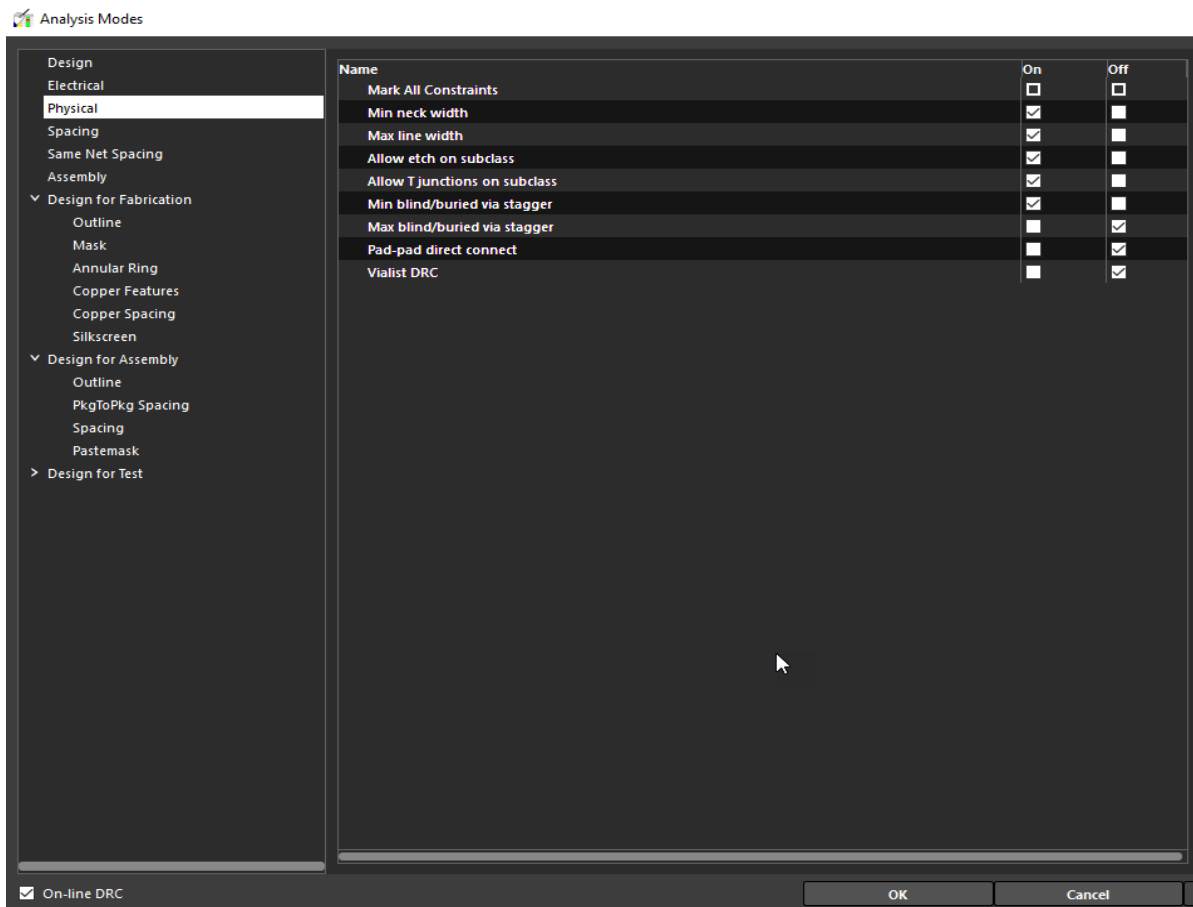


Figure 8-14 Physical Modes

Allegro X Constraint Manager Reference

Analyze Menu Commands

Refer to the [Design for Fabrication](#) in *Analysis Modes Constraints Reference* guide for detailed information on individual constraints.

Spacing Modes

Controls hole based checks between drill hole and conductor element. Controls whether specific design rule checks for spacing modes are triggered based on changes to the layout.

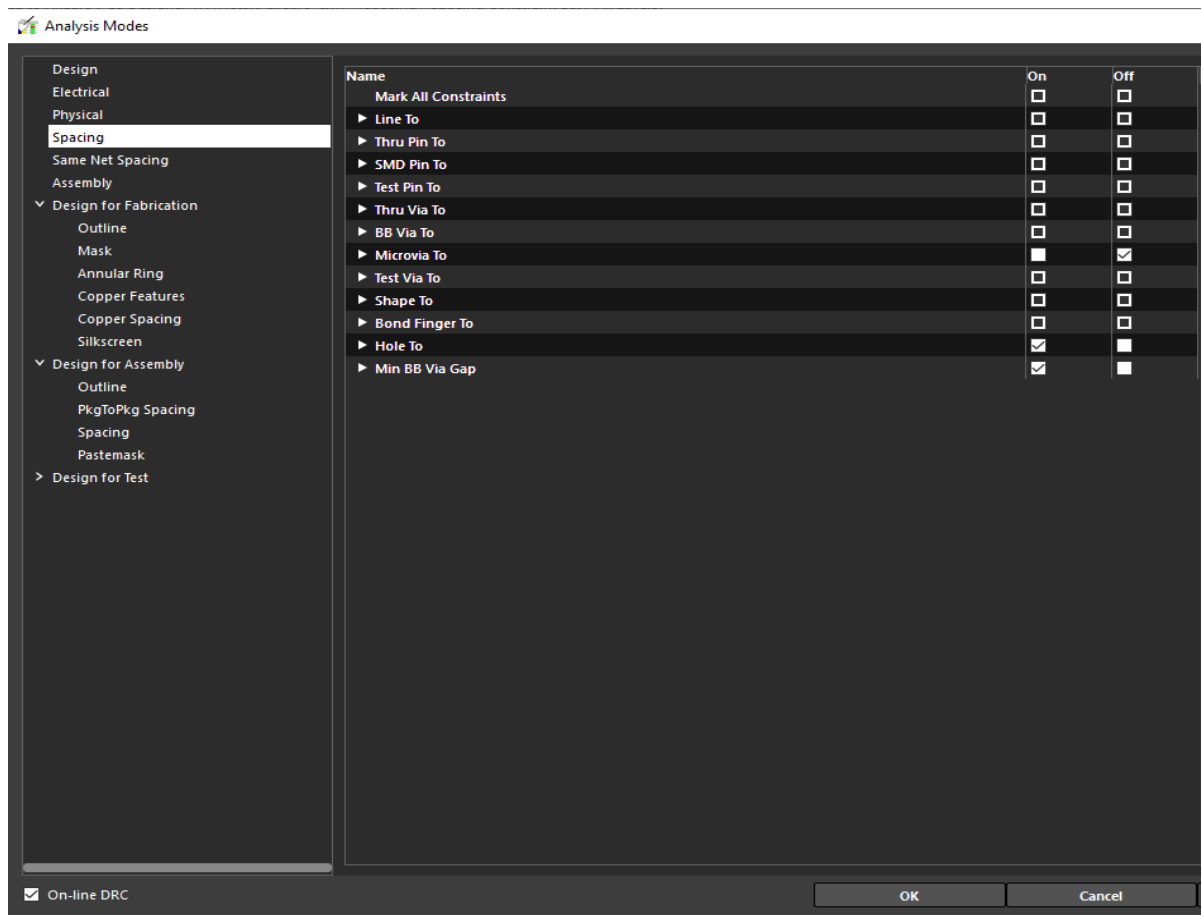


Figure 8-15 Spacing Modes and Options

Refer to the [Design for Fabrication](#) in *Analysis Modes Constraints Reference* guide for detailed information on individual constraints.

Same Net Spacing Modes

Controls whether specific design rule checks for same net spacing are triggered based on changes to the layout.

Allegro X Constraint Manager Reference

Analyze Menu Commands



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 *Allegro Constraint Manager User Guide*.

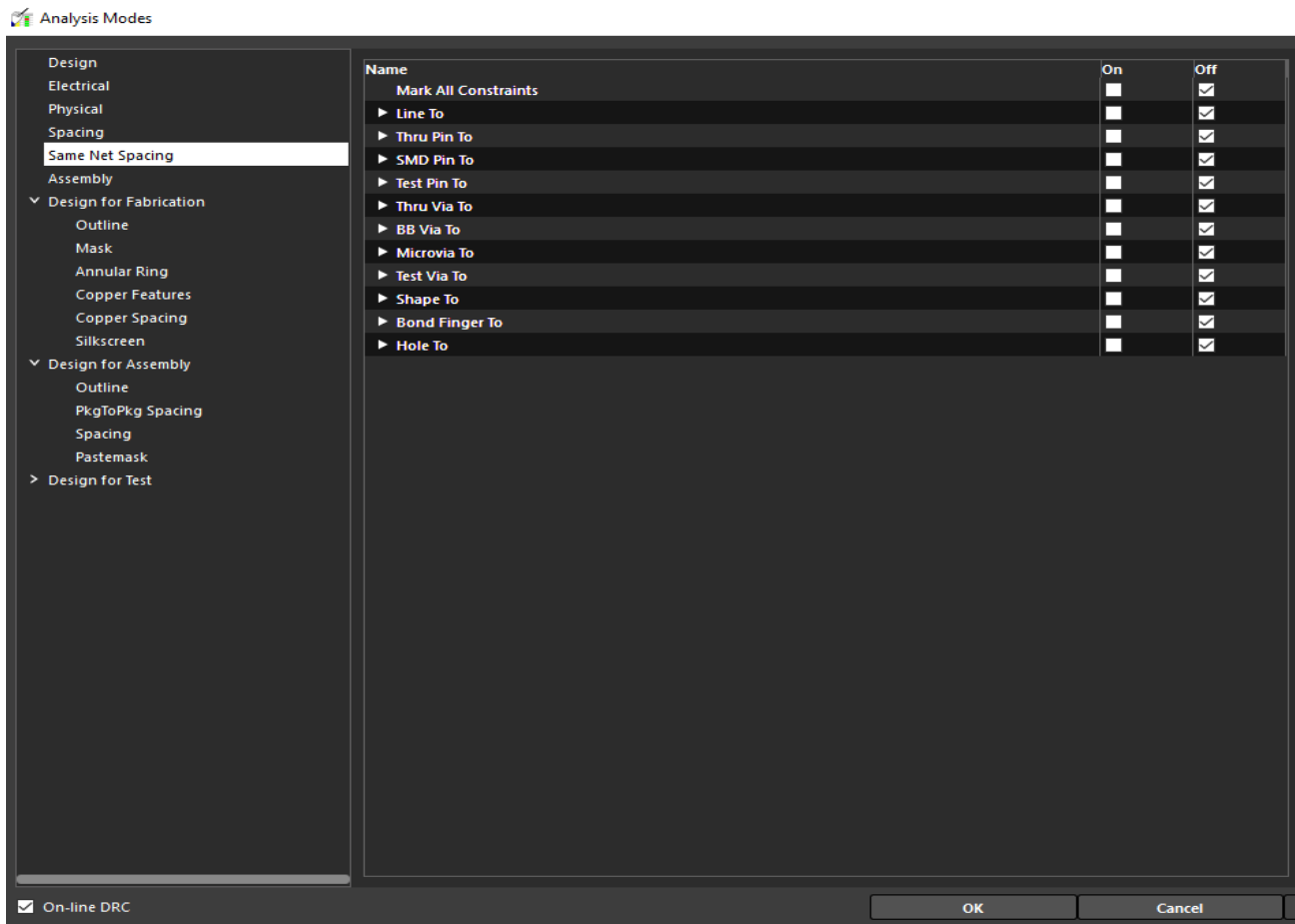


Figure 8-16 Same Net Spacing Modes

Refer to the [Design for Fabrication](#) in *Analysis Modes Constraints Reference* guide for detailed information on individual constraints.

Allegro X Constraint Manager Reference

Analyze Menu Commands

Assembly Modes

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

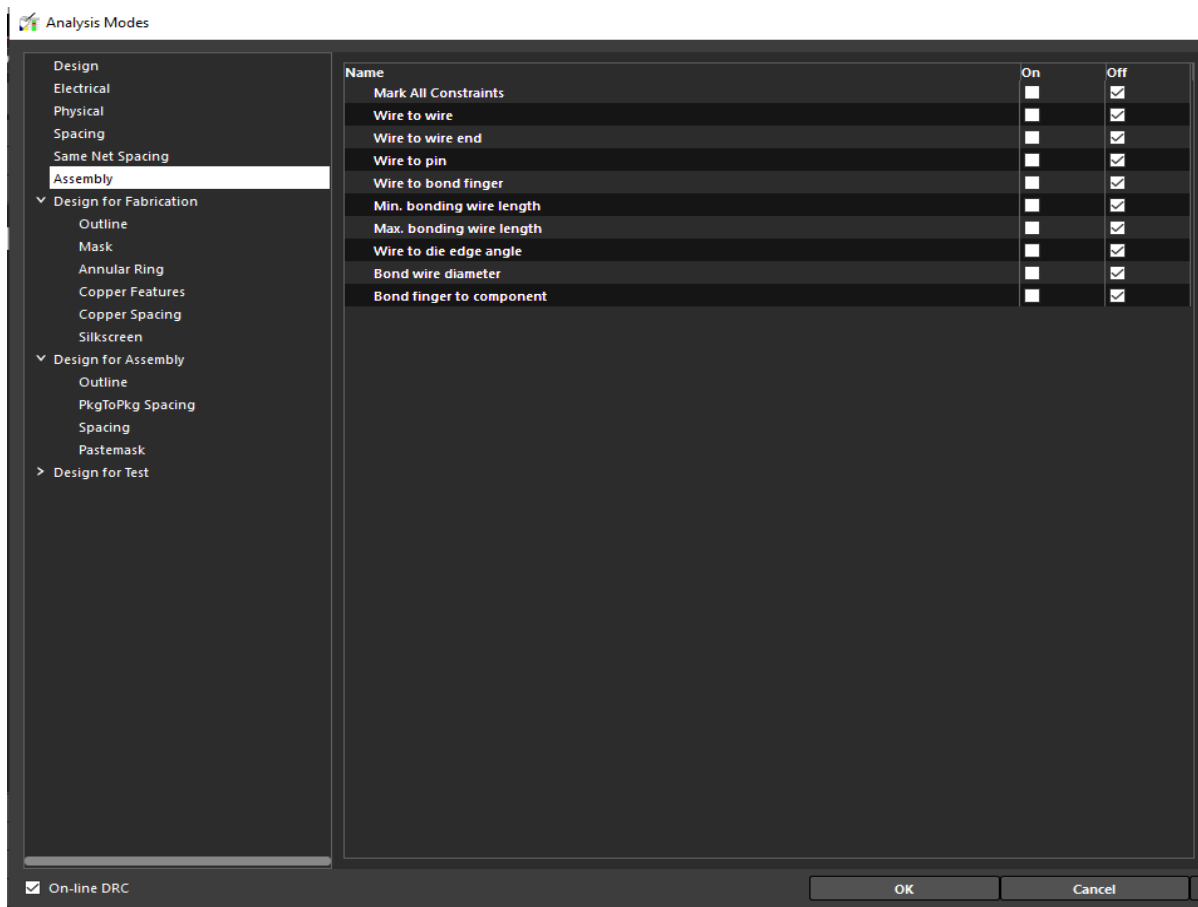


Figure 8-17 Assembly Modes

Refer to the Design for Fabrication in *Analysis Modes Constraints Reference* guide for detailed information on individual constraints.

Allegro X Constraint Manager Reference

Analyze Menu Commands

Design for Fabrication (DFF) Modes

Species spacing parameters for design for fabrication mode checks.

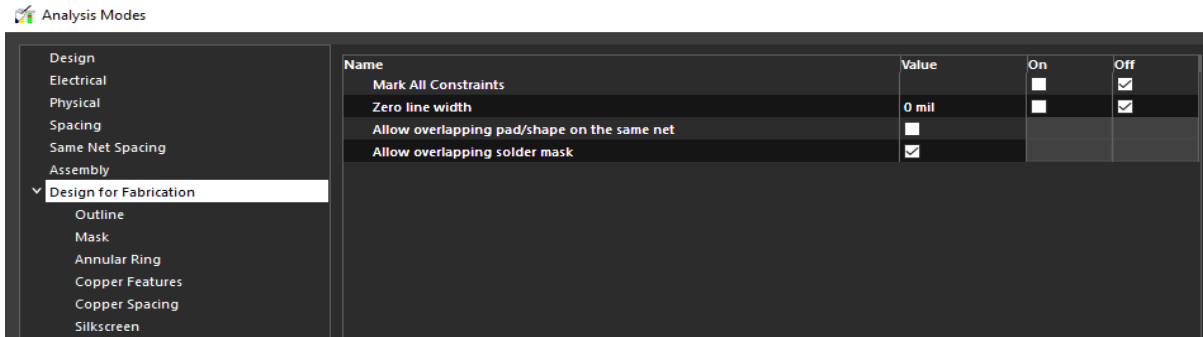


Figure 8-18 Design for Fabrication Modes

Outline

Defines the rules for the spacing between traces, pins, vias, and other non-signal geometry to the board outline.

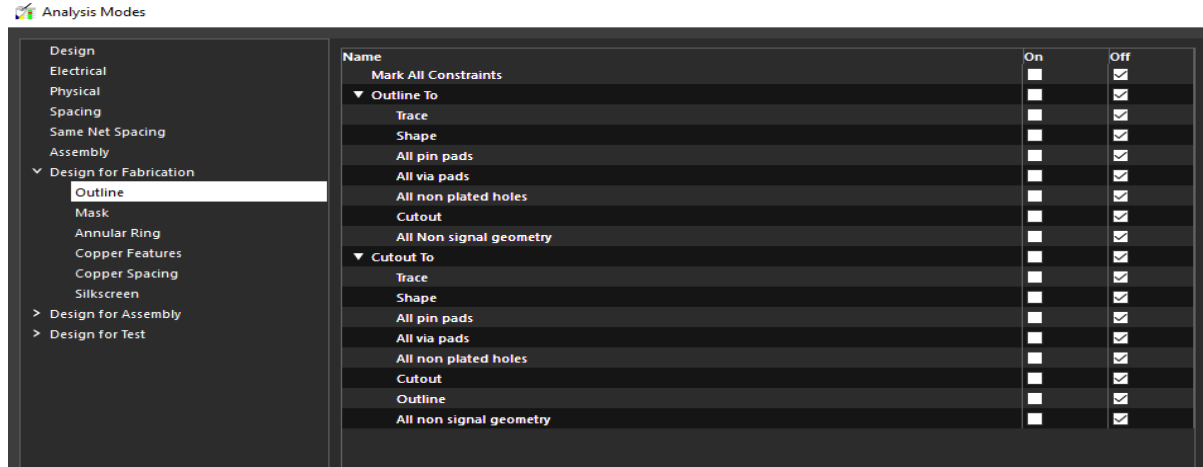


Figure 8-19 Outline Checks

Allegro X Constraint Manager Reference

Analyze Menu Commands

Mask

Defines the rules for minimum mask slivers width and square areas of mask islands.

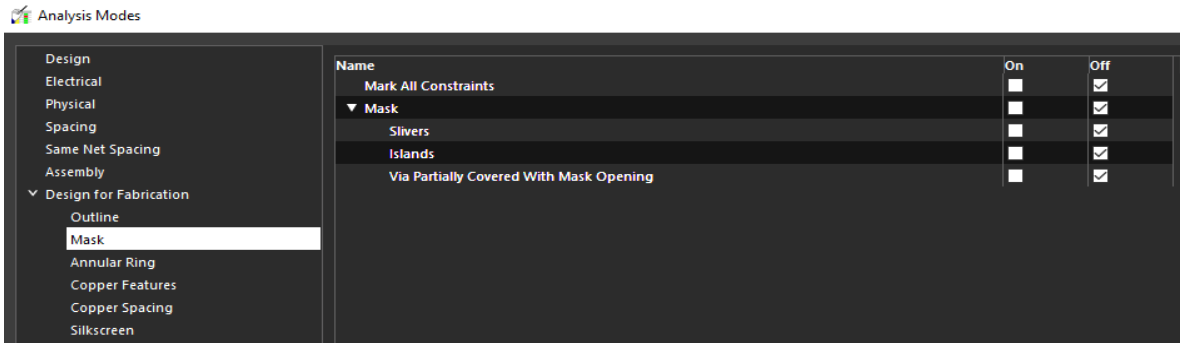


Figure 8-20 Mask Checks

Annular Ring

Defines the size requirements for padstack definitions for hole to pad and pad to mask size relationships for pins and vias.

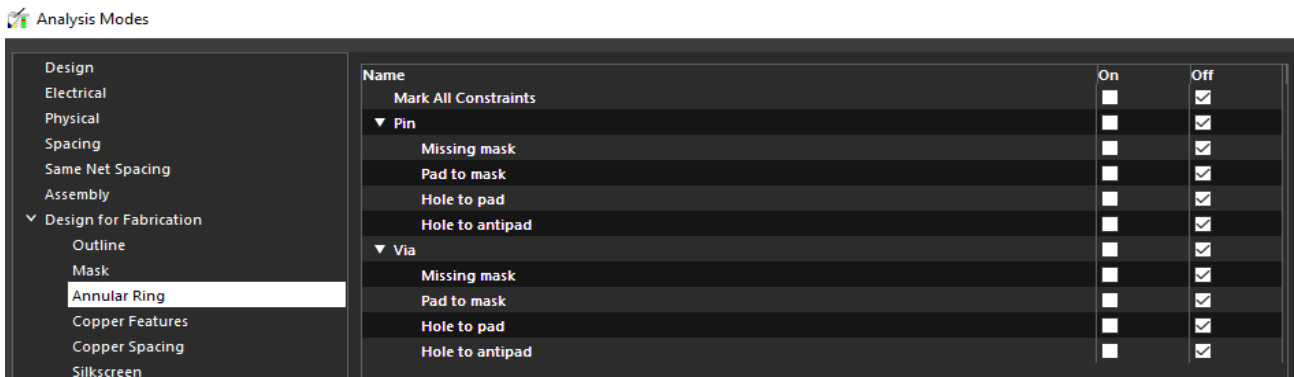


Figure 8-21 Annular Ring Checks

Allegro X Constraint Manager Reference

Analyze Menu Commands

Copper Spacing

Defines the rules for the minimum manufacturing spacing allowed between trace, shape, pin pad, via pad, non-plated hole, and non-signal geometry objects.

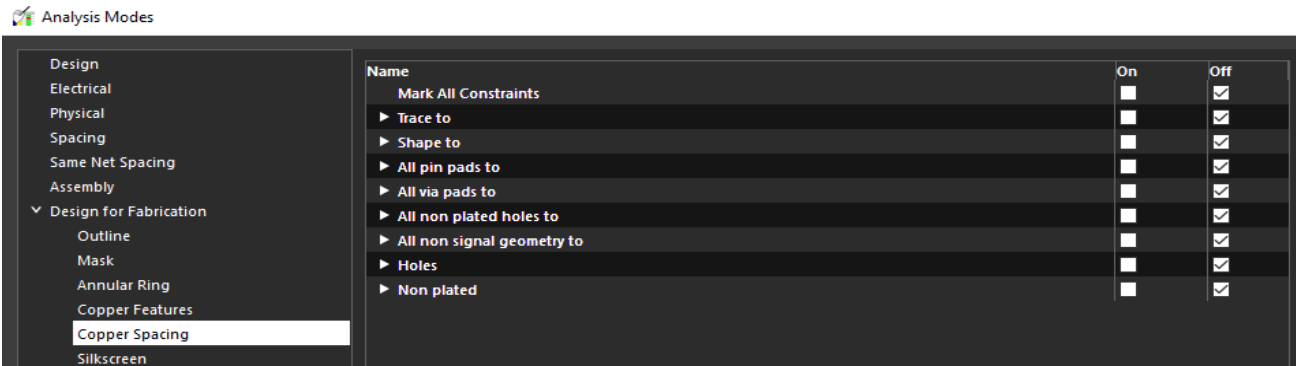


Figure 8-22 Copper Spacing Checks

Silkscreen

Defines the rules for the spacing between pin pads, via pads and non-plated holes to silkscreen geometry.

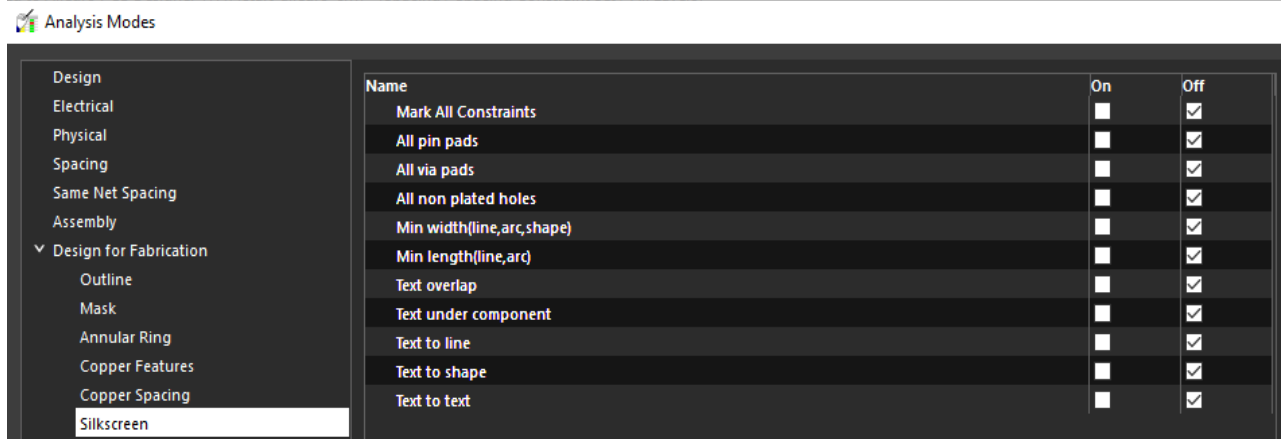


Figure 8-23 Silkscreen Checks

Refer to the [Design for Fabrication in Analysis Modes Constraints Reference](#) guide for detailed information on individual constraints.

Procedures

DRC modes procedures

1. Choose *Analyze – Analysis Modes*.

The *Analysis Modes* dialog box appears.

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



Tip

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

3. Check *On-line DRC*.
4. Click the *Electrical Options* if you want to include the distance between unrouted connections, pin delay, or via structures in delay calculations.
5. Click the *Design* if you want to include plane, soldermask, mechanical hole, and testpoint options.
6. Click the *Manufacturing* if you want to include outline, silkscreen, copper spacing, and annular ring 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*.

Allegro X Constraint Manager Reference

Analyze Menu Commands

The *Analysis Modes* dialog box appears.

2. Click the *Electrical* 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*, 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*, click on a Net, Xnet, Bus, or Diff Pair.
2. Optionally, choose *Reports- Settings* to configure the simulator. Do one of the following:
 - ☐ Choose *Analyze – Analyze*.
 - or -
 - ☐ Right-click and choose *Analyze* from the pop-up menu.



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*.

Allegro X Constraint Manager Reference

Analyze Menu Commands

Tools Menu Commands

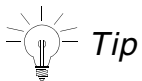
Tools – SigXplorer

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



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:

Allegro X Constraint Manager Reference

Tools Menu Commands

- ❑ 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 14 for topology mapping criteria.

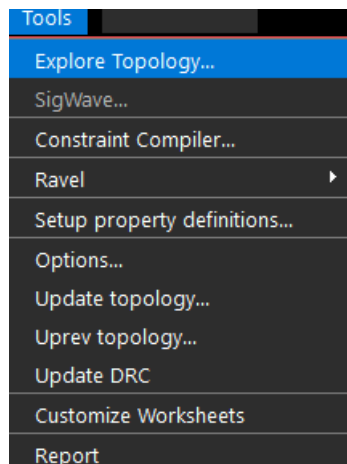
Tools – Explore Topology

Use this command to launch Topology Workbench to explore circuit topologies.

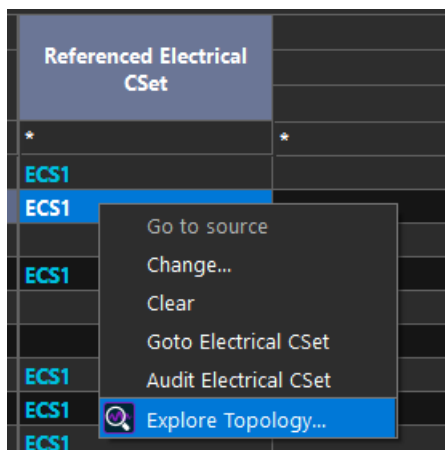
Note: This command is available only for designs (.brd) created in Allegro System Capture or in Allegro X Advanced Package Designer.

Extracting a Topology From an Existing Design

1. Click an object whose topology you want to extract in the *Object* column,
2. Do one of the following:
 - ☐ Choose *Tools – Explore Topology*.



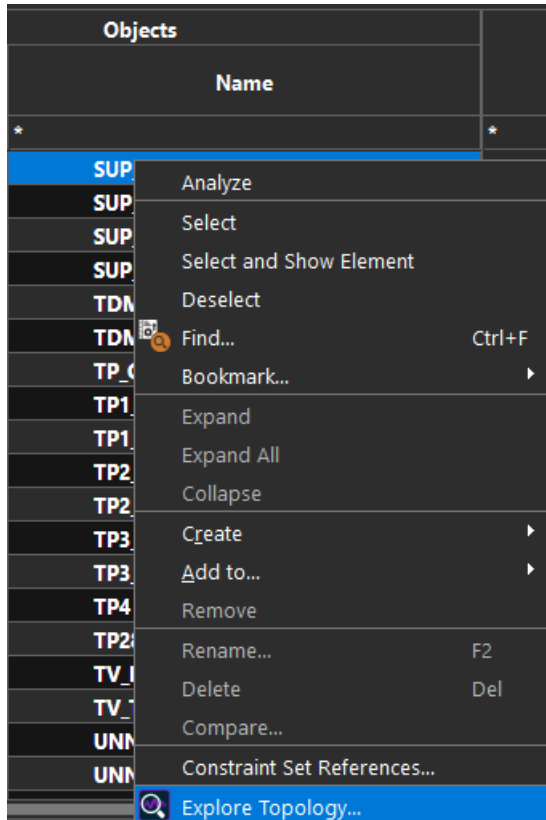
- ☐ Right-click an Electrical CSet.



Allegro X Constraint Manager Reference

Tools Menu Commands

- ❑ Right-click and choose *Explore Topology*.



The *Cadence Product Choices* dialog box opens.

3. Choose *Sigrity Aurora* and click *OK*.

The Topology Workbench launches and displays the circuit topology.

4. Double-click the required connection on the canvas.

The *Properties* panel opens.

5. Define or modify constraint values.
6. Click *Update Constraint Manager* in the *Workflow* panel to update the modified constraint values back to Constraint Manager.
7. Click *Yes* in the confirmation message that appears.
8. Click *Save as ECSet File* in the *Workflow* panel and close Topology Workbench.

Allegro X Constraint Manager Reference

Tools Menu Commands

Related Topics

- [Topology Workbench User Guide](#)
- [Exporting Constraints from a Topology](#)

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 simulate again. Refer to *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 simulate again, follow the procedures for the Analyze – Analyze command.

Tools – Constraint Compiler

This command starts Constraint Compiler, that provides an infrastructure for automatic translation of design constraints from an external library of constraints into Constraint Manager. Using Constraint compiler you can add initial constraint information in the design or update a design that has existing constraints. Constraint Compiler uses design's connectivity information (buses, differential pairs, nets, and so on) in conjunction with data agnostic constraint information to create specific rules for various interfaces in a design. The constraint compiler lets you add constraints at the interface level based on manufacturer guidelines.

Constraint Compiler is available with the following product licenses:

- Allegro PCB Editor with High Speed Option
- Allegro Venture PCB Designer Suite
- Allegro Enterprise PCB Designer Suite
- Allegro X Advanced Package Designer (APD)
- Allegro Design Authoring (Schematic)

For more information, see *[Allegro® Constraint Manager User Guide](#)*.

Dialog Boxes

- [Constraint Compiler Dialog Box](#)
- [More Options Dialog Box](#)

Constraint Compiler Dialog Box

Library Path

Displays the path of the current working directory. Click the browse button to view the list of available library paths:

- The path of default library from the installation hierarchy
- The path of user-defined libraries set using *acopath* environment variable

Selecting a path opens the file browser to navigate to a different folder and sets it as library path.

Allegro X Constraint Manager Reference

Tools Menu Commands

<i>Select Files</i>	Lists all the rule/table files (.csv, .xml, .zip) available in the <i>Library Path</i> . Select any .csv file to preview the file content in a window in the right-hand pane.												
<i>Load Selected Files</i>	Click to load all the selected library files to run customized search based on keys. The keys are the table headers defined in the rule/table files.												
<i>Filter Template Keys (Optional)</i>	Use the query functionality to filter a library file using keys. <table border="0" style="margin-left: 20px;"> <tr> <td><i>Keys</i></td><td>Displays table keys to provide easier selection</td></tr> <tr> <td><i>Query Filter</i></td><td>Specify filter definitions to queries to select only those files that meet the defined criteria</td></tr> <tr> <td><i>Define Query Filter</i></td><td>Displays filter settings</td></tr> <tr> <td><i>Filter</i></td><td>Displays name of the key</td></tr> <tr> <td><i>Operation</i></td><td>Choose to select an operation that you want to set for the selected key</td></tr> <tr> <td><i>Value(s)</i></td><td>Choose to select a value for the selected key from the pull-down menu</td></tr> </table>	<i>Keys</i>	Displays table keys to provide easier selection	<i>Query Filter</i>	Specify filter definitions to queries to select only those files that meet the defined criteria	<i>Define Query Filter</i>	Displays filter settings	<i>Filter</i>	Displays name of the key	<i>Operation</i>	Choose to select an operation that you want to set for the selected key	<i>Value(s)</i>	Choose to select a value for the selected key from the pull-down menu
<i>Keys</i>	Displays table keys to provide easier selection												
<i>Query Filter</i>	Specify filter definitions to queries to select only those files that meet the defined criteria												
<i>Define Query Filter</i>	Displays filter settings												
<i>Filter</i>	Displays name of the key												
<i>Operation</i>	Choose to select an operation that you want to set for the selected key												
<i>Value(s)</i>	Choose to select a value for the selected key from the pull-down menu												
<i>Save</i>	Click to save query filter settings in a query (.qry) file												
<i>Run Saved Query</i>	Choose to load an existing query in the compiler												
<i>Compiler Options</i>	Click to open compiler settings <table border="0" style="margin-left: 20px;"> <tr> <td><i>Update only auto-generated objects</i></td><td>Choose to update only those objects that were auto-generated in previous run using Constraint Compiler. Enabling this option protects existing object groupings (for example, Diff Pairs) from being regenerated. An error is reported if the condition exists.</td></tr> <tr> <td><i>Update only auto-generated constraints</i></td><td>Choose to update only those constraints that were auto-generated in previous run using Constraint Compiler. Enabling this option protects existing constraint values (for example, Total Etch Length Max) from being updated. A warning is reported if the condition exists.</td></tr> </table>	<i>Update only auto-generated objects</i>	Choose to update only those objects that were auto-generated in previous run using Constraint Compiler. Enabling this option protects existing object groupings (for example, Diff Pairs) from being regenerated. An error is reported if the condition exists.	<i>Update only auto-generated constraints</i>	Choose to update only those constraints that were auto-generated in previous run using Constraint Compiler. Enabling this option protects existing constraint values (for example, Total Etch Length Max) from being updated. A warning is reported if the condition exists.								
<i>Update only auto-generated objects</i>	Choose to update only those objects that were auto-generated in previous run using Constraint Compiler. Enabling this option protects existing object groupings (for example, Diff Pairs) from being regenerated. An error is reported if the condition exists.												
<i>Update only auto-generated constraints</i>	Choose to update only those constraints that were auto-generated in previous run using Constraint Compiler. Enabling this option protects existing constraint values (for example, Total Etch Length Max) from being updated. A warning is reported if the condition exists.												

Allegro X Constraint Manager Reference

Tools Menu Commands

<i>Use Differential Pairs when updating groups</i>	<p>Choose to select both members of a differential pair for addition into a group object, such as Net Group and Net Class.</p> <p>This option is enabled by default and includes the differential pair object instead of the individual differential pair members.</p> <p>Note: Differential pair objects must already exist in design or created prior to processing data row which calls out Differential Pair, otherwise individual members are added to group.</p>
<i>Run DRC and Update Shapes</i>	Choose to run design rule checks and update dynamic shapes after rules are imported in the Constraint Compiler.
<i>Report only (Validate)</i>	<p>Run the compiler in read-only mode to check for errors or compatibility with the design.</p> <p>A report is generated that displays all warnings, errors and the expected changes to the design.</p>
<i>Save for batch</i>	Choose to save the compiler settings in an XML-based (.accx) file. This file can be used to run compiler using batch command.
<i>Import</i>	<p>Click to run the compiler to apply constraints in the design.</p> <p>A report is generated that displays all warnings, errors and the changes made to the design.</p>
<i>Back</i>	Use this button to navigate in the compiler UI.
<i>Next</i>	Use this button to navigate in the compiler UI.
<i>Close</i>	Closes the compiler without applying constraints to the design

More Options Dialog Box

Options

<i>Do not report warnings</i>	Choose to not generate compiler warnings
<i>Mark auto-generated objects as read-only</i>	Choose to mark all auto-generated objects as read-only to prevent accidental modification in Constraint Manager.
<i>Create empty Net Classes</i>	Choose to create empty Net Classes if required.
<i>Continue with errors</i>	Choose to allow the compiler to continue in case of errors
<i>Create Ratsnest Bundles for auto-generated Net Group</i>	<p>Choose to create Ratsnest bundles for the Net Groups.</p> <p>By default, Net Groups created by the compiler set the Disable Automatic Ratsnest Bundle to <i>Off</i> under the Ratsnest Bundle Properties workbook in Properties domain of Constraint Manager.</p>
<i>Do not delete any objects, attributes, or relationships</i>	<p>Choose to allow only addition and modification in constraint information and restrict any deletions.</p> <p>By default, the compiler replaces all constraint information for objects which are processed in the selected specifications.</p>

Restore Default *Reset all compiler options to their default settings*

Setting the Constraint Library

To select rule/table files in Constraint Compiler, first set the path of constraint library using the following steps:

1. Open a design into which you want to run constraint compiler.
2. Choose *Setup – User Preferences*.

The User Preferences Editor opens.

3. Expand *Paths* folder and click to select *Config* folder.
4. Set the value for *acopath* variable to the location of the constraint library.
5. Click *OK* to close the User Preferences Editor.

Running Constraint Compiler

To apply constraints in a design using compiler, do the following:

1. Open Constraint Manager and choose *Tools – Constraint Compiler* from the menu bar.

The Constraint Compiler dialog box appears.

2. To select relevant library files, you can choose the following two ways:

- a. In the *Select Files* browser, expand library folders and select *.csv* / *.xml* / *.zip* files based on their names and dependencies.

- b. Click the *Load Selected Files* button.

Or

- a. In the *Select Files* browser, enable the top-level library folder to select all the files.

- b. Click the *Load Selected Files* button. In the *Keys* section, double-click a key name to select it for query filter.

- c. In the *Query Filter* pane, select the key and change its value in the *Define Query Filter* pane.

3. Optionally, click the *Save* button to save the query settings.
4. Click the *Next* button to view the query results.
5. Specify compiler options.
6. To set more compiler setting, click *More options*.
7. To validate the compiler results, enable *Report only (Validate)* compiler setting and click *Import*.

Compiler generates Allegro Constraint Compiler Report to review error and warnings before applying constraints.

8. Verify the report for new constraints and close the report.
9. Click the *Import* button to apply the constraints into a design.

Allegro X Constraint Manager Reference

Tools Menu Commands

10. Click the *Close* button to close the compiler and review the results in Constraint Manager.

Tools – Ravel – Delete All Markers

Use this command to delete all the DRC markers created when Ravel checks are performed in a design.

For more information on Ravel markers, see [File – Import – Ravel File](#) command.

Tools – Setup Property Definitions

Use this command to create a new attribute definition.

Setup Property Definitions Dialog Box

<i>Type</i>	Specifies the type of the attribute. Options are User-defined or Pre-defined.
<i>Filter</i>	
<i>Domain</i>	Provides a pull-down list of domain names for searching pre-defined attributes.
<i>Text</i>	Specifies the text for searching within the list of the pre-defined attributes.
<i>Create</i>	Displays the <i>Create Attribute</i> dialog box to create a user-defined attribute.
<i>View</i>	Displays the <i>Create Attribute Definition</i> dialog box for viewing definition of a pre-defined attribute.
<i>Edit</i>	Displays the <i>Create Attribute Definition</i> dialog box for editing definition of a user-defined attribute.
<i>Delete</i>	Removes an attribute definition of the selected user-defined attribute.
<i>OK</i>	Closes the dialog box.
<i>Cancel</i>	Closes without adding a column to the current worksheet, but preserves attribute definition creations or edits or deletes.

Create Attribute Dialog Box

<i>Name</i>	Specifies the name of a user-defined attribute.
<i>Copying Option</i>	
<i>Copy from attribute</i>	Copy name of an existing user-defined attribute.
<i>OK</i>	Opens <i>Create Attribute Definition</i> dialog box.
<i>Cancel</i>	Closes without applying any changes to the dialog box.

Create Attribute Definitions Dialog Box

<i>General</i>	Specifies general characteristics of an attribute.
<i>Name</i>	Specifies the name of the attribute. The maximum number of characters allowed are less than or equal to 64.
<i>Type</i>	Specifies data type of the attribute.
<i>Treat As</i>	Specifies that the attribute should be considered as: <i>Property</i> , <i>Min Constraint</i> , <i>Max Constraint</i> , <i>Target +/- Tolerance</i> , or <i>Target Tolerances</i> , or <i>Actual</i> .
<i>Description</i>	Specifies details about new attribute. The maximum number of characters allowed are 255.
<i>Display Name</i>	Specifies a display name of the attribute.
<i>DB Version</i>	Specifies the database version in which the attribute is created.
<i>Range</i>	Specifies the start- and end-point values.
<i>Measurement</i>	Specifies the measurement for the attribute to analyze.
<i>Flow - To Physical</i>	Specifies that the attribute passes to PCB Editor (via <code>netrev</code>).
<i>Flow - To Logical</i>	Specifies that the attribute passes to Allegro® System Architect or Design Entry HDL (via <code>genfeedformat</code>).
<i>Flow - Flags</i>	Only available when you launch Constraint Manager from Allegro Design Entry HDL.
<i>Flow - Flags - Package</i>	The property is transferred between Design Entry HDL and Allegro PCB Editor.
<i>Flow - Flags - PackageCreatePart</i>	<p>Creates a new physical part for each unique attribute value. If instances of a component have the property with the same value, Design Entry HDL packages the instances together. However, instances that do not have the property will not be packaged together with instances that have the same value for the property.</p> <p>For example: If you have two instances of 74LS00 in your design with a property, MYPROP, with two different values, ALT1 and ALT2, two physical parts are created: 74LS00-ALT1 and 74LS00-ALT2.</p> <p>If both instances of 74LS00 have the MYPROP=ALT1 property, Design Entry HDL packages them as a single physical part named 74LS00-ALT1.</p>

Flow - Flags -

PackageSameValueOnly

Packages instances with the same attribute value in one physical part. Design Entry HDL does not package together instances that have the same property but with different values. Instances without the property will not be packaged together with instances that have the same property and matching values.

For example: If you have two instances of 74LS00 in your design with the property, MYPROP, with two same values, ALT1, Design Entry HDL packages them as a single physical part named 74LS00-ALT1.

Flow - Flags -

PackageSameValue

Packages instances with the same attribute value in one physical part. If spare sections exist, packages the section that do not have the property. Design Entry HDL does not package together any instances that have different values for the same property.

For example: If you select this option for a property, MYPROP, and you have four instances of 74LS00:

- Instance i1 has MYPROP=ALT1
- Instance i2 has MYPROP=ALT2
- Instance i3 has MYPROP=ALT1
- Instance i4 does not have MYPROP

then Design Entry HDL packages instances i1 and i3 together because they have the same property value.

The 74LS00 component has four sections. As there are two spare sections instance i2 and i4, Design Entry HDL packages instance i4 with i2 because i4 does not have the MYPROP property.

Allegro X Constraint Manager Reference

Tools Menu Commands

<i>Flags</i>	Controls the attribute by enabling one of the following options: <ul style="list-style-type: none">■ NoInherit: Attribute is never inherited within the specified valid objects.■ ReadOnly: Attribute cannot be set through UI.■ UpperCaseValue: Attribute value is in uppercase.■ NoEditor: Attribute value is not editable in any UI.■ NoMerge: Attribute will not be processed when a hierarchical block is merged into a higher-level design.■ NoAliasMerge: Attribute is not processed when an alias is created or dissolved.■ NoReport: Differences for this attribute will not be reported.
<i>Objects</i>	Specifies valid objects for the attribute.
<i>OK</i>	Applies changes made to the attribute definition and closes the dialog box.
<i>Cancel</i>	Closes and cancels the changes made to the dialog box.
<i>Apply</i>	Click to apply the changes made to the dialog box.

Procedures

Creating User-defined Attribute

1. Choose *Tools – Setup property definitions*.

The *Setup Property Definitions* dialog box appears.

2. In the *Type* field, enable *User-defined* checkbox.
3. Click *Create*.

The *Create Attribute* dialog box appears.

4. Enter the name of attribute.
5. Optionally, enable *Copy from Attribute* checkbox.

The new attribute name is created with default `_COPY` name. You can edit the name.

6. Click *OK* to setup the attribute definitions.

The *Create Attribute Definition* dialog box appears.

7. Specify the attribute definitions and click *OK* in the *Create Attribute Definition* dialog box.

The attribute name is added to the list in the *Setup Property Definitions* dialog box.

8. Click *OK* to close the dialog box.

Setting up Attribute Definitions

1. In the *Type* field, specify an attribute's data type. The relevant units are displayed in the associated field.

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

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

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

4. In the *Display Name* field, enter a display name for the new attribute.

5. In the *Range* field, enter start- and end-points values for the new attribute.

6. Click the *Link* button associated to the *Measurement* field, for creating measurement that will be used to compute the results for the new attribute.

The *Select or Create Measurement* dialog box appears.

7. Select a measurement and click *OK* in the *Select or Create Measurement* dialog box.

8. In the *Flags* field, enable one or more of the following checkboxes:

- ☐ NoInherit
- ☐ ReadOnly
- ☐ UpperCaseValue
- ☐ NoEditor

- ☐ NoMerge
- ☐ NoAliasMerge:
- ☐ NoReport

9. In the *Objects* field, enable one or more of the available checkboxes. Some of the valid objects are:

- ☐ Design
- ☐ Part Defn
- ☐ Part
- ☐ Gate Defn
- ☐ Gate
- ☐ Pin Defn
- ☐ Pin
- ☐ Net
- ☐ Xnet
- ☐ Layer
- ☐ Region
- ☐ Electrical CSet
- ☐ DRC
- ☐ NetClass

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

Viewing a pre-defined attribute

1. Choose *Tools – Setup property definitions*.

The *Setup Property Definitions* dialog box appears.

2. In the *Type* field, enable *Pre-defined* checkbox.

3. Filter the name of the domain.

4. Select the attribute from the list of pre-defined attributes.

Allegro X Constraint Manager Reference

Tools Menu Commands

5. Click *View* to see the parameters of the attribute.

The *View Attribute Definition* dialog box appears.

6. Click *Cancel* to the *View Attribute Definition* dialog box.
7. Click *OK* to close the dialog box.

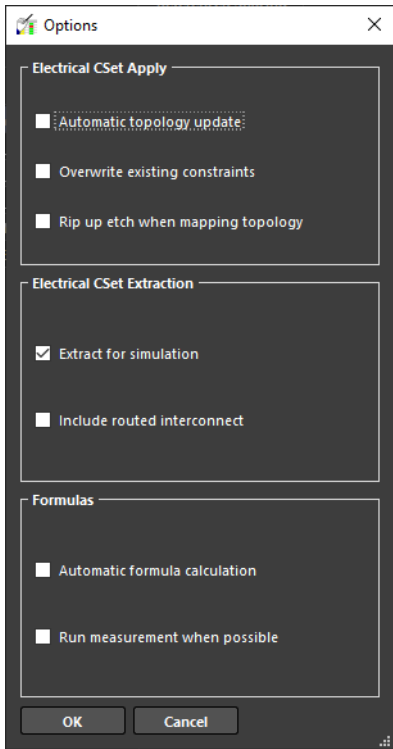
Tools – Precision

Use this command to change the precision value of constraints.

This option is available when Constraint Manager is launched from the schematic creation tools.

Tools – Options

Use this command to specify options that govern Electrical CSet extraction and application. This displays the Options dialog box.



Options Dialog Box

Checkbox Option	Function
<i>Electrical CSet Extraction</i>	Available when Constraint Manager is launched from back-end.
<i>Include Routed Interconnect</i>	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 <i>Edit – Transform – For Constraint Manager</i> .

Allegro X Constraint Manager Reference

Tools Menu Commands

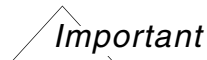
Checkbox Option	Function
<i>Extract for Simulation</i>	Indicates that the extraction will run simulation extraction using all available models.

Checkbox Option	Function
<i>Electrical CSet Apply</i>	Available when Constraint Manager is launched either from front-end or back-end.
<i>Rip up etch when mapping topology</i>	Controls whether etch (clines and vias) is removed when an Electrical CSet is reapplied and the schedule of the net changes.
<i>Automatic topology update</i>	<p>Controls how topology-related constraints are reapplied</p> <ul style="list-style-type: none">■ When the design changes (component placement, signal model updates) – or –■ When an Electrical CSet is initially referenced <p>When <i>enabled</i>, changes are applied as the design changes.</p> <p>When <i>disabled</i>, changes can be applied by choosing <u>Tools – Update Topology</u>.</p> <p>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.</p> <p>Note: Disabling automatic topology update may be necessary when design changes are frequent and complex ECsets are referenced.</p>

Allegro X Constraint Manager Reference

Tools Menu Commands

Checkbox Option	Function
<i>Overwrite existing constraints</i>	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.



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.

Checkbox Option	Function
<i>Formulas</i>	Available when Constraint Manager is launched from back-end.
<i>Automatic formula calculation</i>	<p>Controls whether formulas are <u>calculated</u> automatically. If this check box is selected, each formula is examined to automatically determine its dependencies and recalculation is triggered automatically when the formulas are found to be stale.</p> <p>When set to Off, formulas are not recalculated automatically. Dependency information is used to mark formulas stale, but does not trigger recalculation. When a formula is calculated, it uses the dependency information to determine if any other formulas needs to be recalculated first, and checks for cyclic dependencies.</p> <p>When the automatic formula calculation function is set from Off to On, a <u>message</u> is displayed informing you that all formulas must be recalculated to be up to date before the automatic formula calculation function is enabled.</p> <p>Note: In order for a calculated formula value to be accurate, all the cells it depends on must be up to date before the calculation is done. When a formula is selected for calculation, the dependency information is used to determine on which cells the formula depends.</p>

Allegro X Constraint Manager Reference

Tools Menu Commands

Checkbox Option	Function
<i>Run measurement when possible</i>	<p>Formulas may have dependencies on measurement values that have not yet been analyzed. Always triggering the analysis could be too time consuming when the results are not currently needed. A global Run measurement setting is used to indicate whether to perform these analysis.</p> <p>When the <i>Run measurement when possible</i> option is set to Off, analysis is not run, and any formula values that depend on unanalyzed measurement values is left empty. When the option is set to On, any Actual cells that use a measurement are analyzed when a formula is calculated, regardless of whether the calculation was triggered automatically or by using the <i>Calculate</i> command.</p> <p>The behavior when changing the Run measurement setting from Off to On depends on the value of the Automatic formula calculation function setting. If it is On when Run measurement is turned On, it results in all empty formulas being recalculated, this time forcing any required analysis to run. If Automatic formula calculation is Off, changing Run measurement to On does not have any immediate effect. However, the required analysis is forced to run when a formula is manually calculated</p>

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* check box 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* check box 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*.

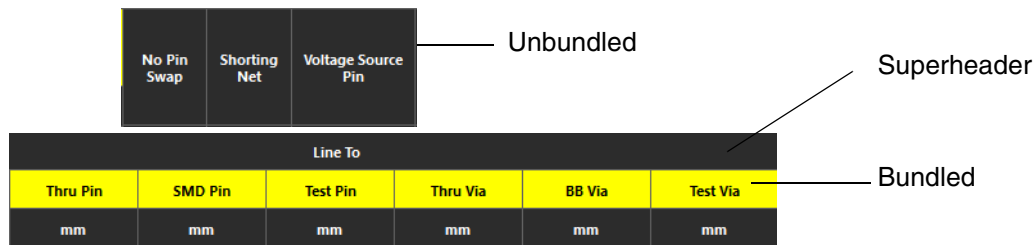
A column superheader icon appears as a rectangle over a circle; a column icon appears as a circle. A predefined column icon is gray; a customized column icon is blue; a hidden column icon is a silhouette of the column icon. See the [Customized Worksheets and associated icons](#) figure on page 275 for 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 275).

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 9-24 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. Enable *Predefined* or *User-defined* checkbox 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* dialog box appears.

- b. In the Name field, specify a unique name for the new attribute.**
- c. Optionally, enable Copying from attribute checkbox to copy the attribute definition from an existing attribute.**
- d. Click *OK*.**

The *Create Attribute Definition* dialog box appears.

- a. In the *General* section,**
 - ☐ In the *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.

Allegro X Constraint Manager Reference

Tools Menu Commands

- ❑ 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.

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

b. In the *Flow* section,

- ❑ Enable *To Physical* and *To Logical* checkboxes 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.

c. In the *Flags* section,

- ❑ Enable one or more of the following checkboxes:
 - NonInherit
 - ReadOnly
 - UpperCaseValue
 - NoEditor
 - NoMerge
 - AnAliasMerge
 - NoReport

d. In the *Objects* section,

☐ Enable one or more of the following checkboxes:

- ☐ Part Definition
- ☐ Part Instance
- ☐ Gate Instance
- ☐ Pin
- ☐ Xnet
- ☐ Net
- ☐ Layer
- ☐ CSet
- ☐ Diff Pair
- ☐ Bus
- ☐ Pin Pair
- ☐ Result
- ☐ Match Group
- ☐ Design Instance
- ☐ Net Group
- ☐ Net Interface
- ☐ Voltage Definition
- ☐ 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.

e. Click *OK*.

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

- f. 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.

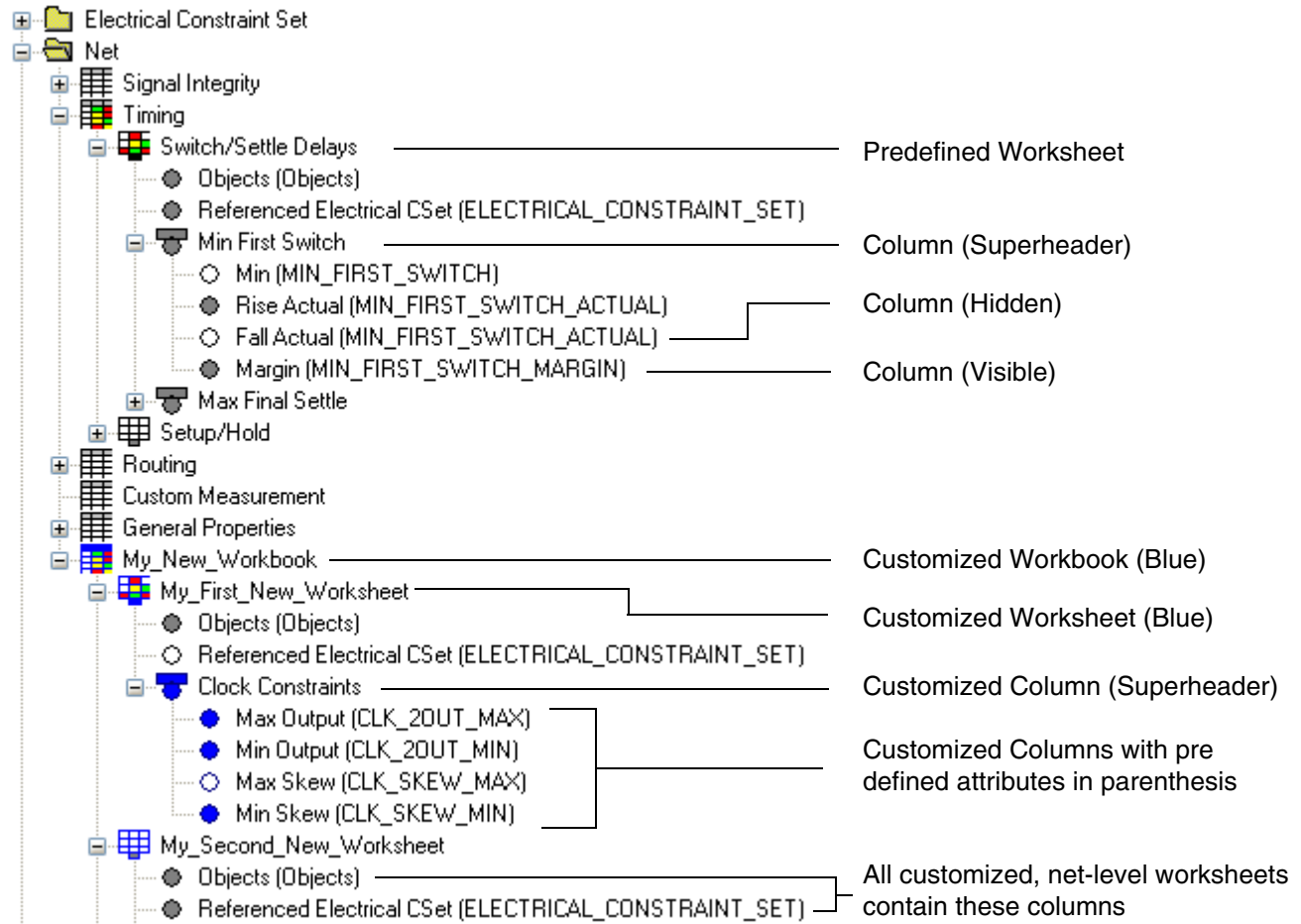
4. Click *Yes* to acknowledge the message.

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

Allegro X Constraint Manager Reference

Tools Menu Commands

Figure 9-25 Customized Worksheets and associated icons

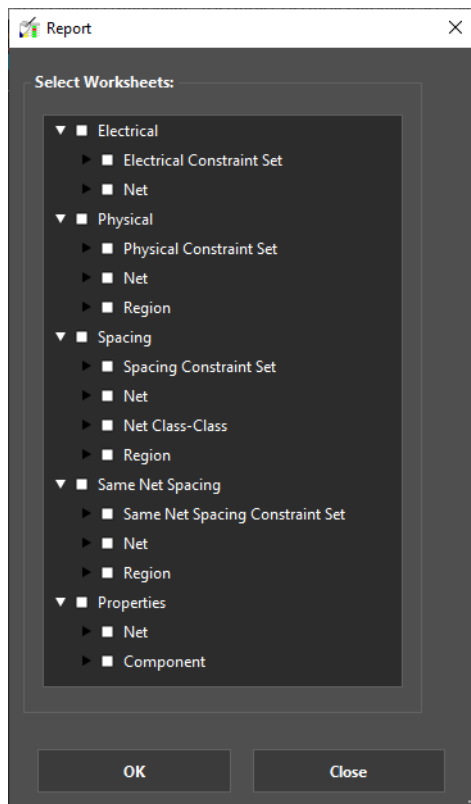


Tools – Report

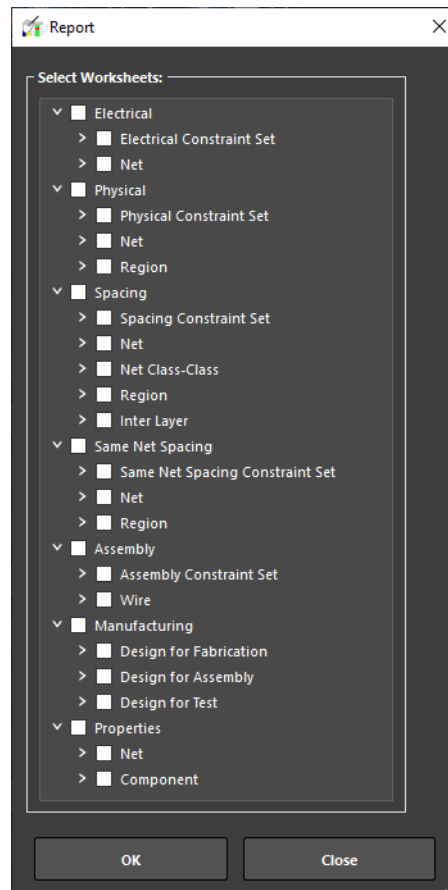
Use this command to specify which worksheets to include in the HTML report.

Procedures

Report Dialog Box



Report dialog box launched from Constraint Manager - Front-end.



Report dialog box launched from Constraint Manager - Back-end.

Allegro X Constraint Manager Reference

Tools Menu Commands

Use this field . . .

Select Worksheets

To . . .

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.

Procedure

Generating a HTML report

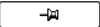



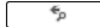
1. Choose *Tools – Report*.

The *Report* dialog box appears.

2. Select the worksheets for which you want to generate report from the *Select Worksheets* section.
3. Click *OK*.
4. Click *Save* to confirm when prompted to save the report file as .html file.

Constraint Manager displays a HTML report for the selected tab.

Report file : C:\D\tutorials\project\test_Electrical.html

   Search:   ☐ Match word ☐ Match case

Version : 17.400

Date/Time : 2020-04-03T10:57:53

1. Electrical\Net\Routing\Wiring															
Objects			Referenced Electrical CSet	Topology				Stub Length			Exposed Length				
Type	S	Name		Verify Schedule	Schedule	Actual	Margin	Max	Actual	Margin	Max	Actual	Margin		Max
*	*	*	*	*	*	*	*	*	*	*	*	*	*	*	
Dsn		start_allegro_cm													
Bus		A(23)													
Net		A<1>													
Net		A<2>													
Net		A<3>													
Net		A<4>													

Allegro X Constraint Manager Reference

Tools Menu Commands

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.

Allegro X Constraint Manager Reference

Help Menu Commands

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.

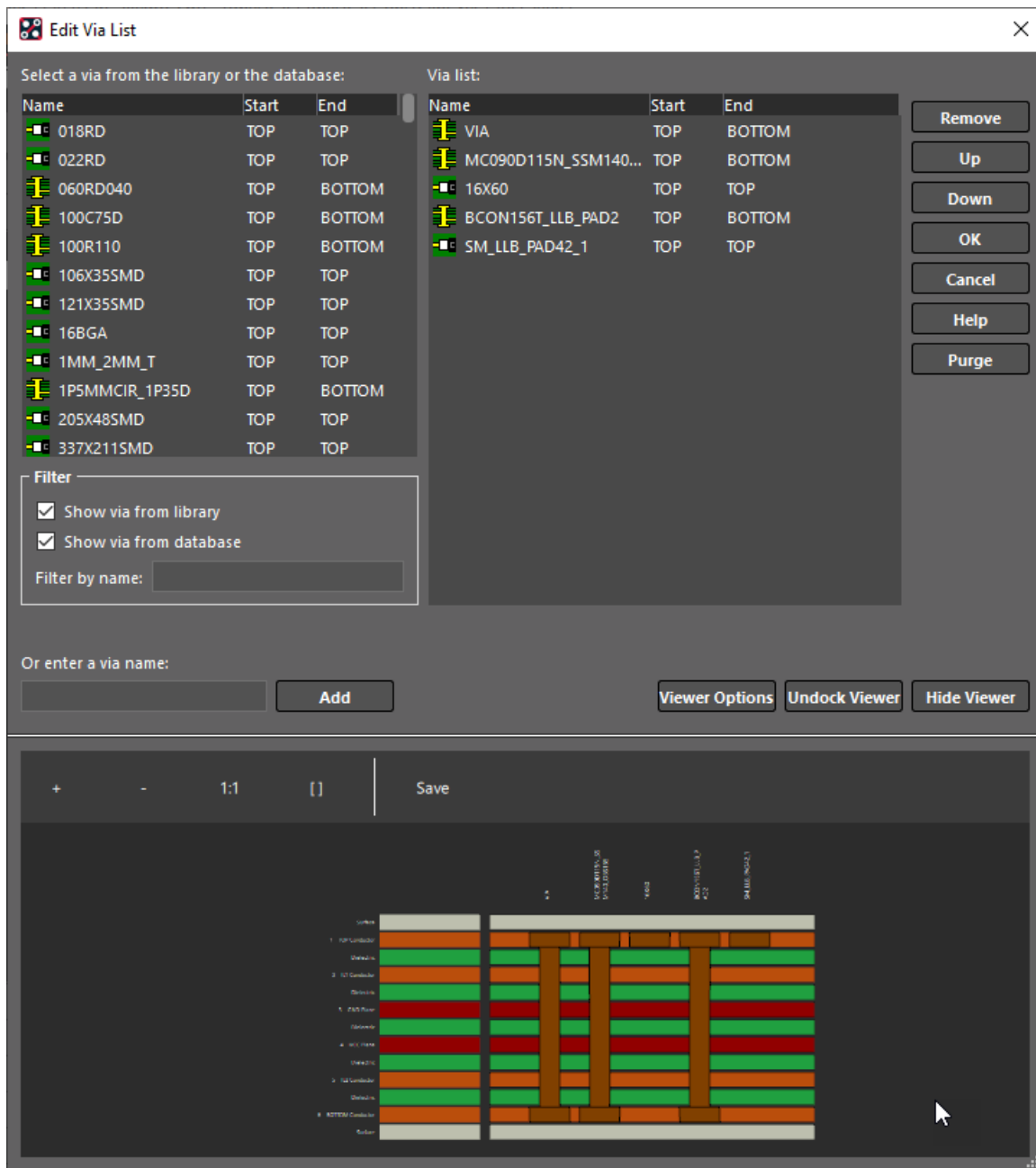
The list represents a selectable order of vias used by certain manual and auto routing commands, such as *Add Connect*. The via order allows an application to automatically select the first via in the list that meets the criteria. For example, if add connection needs to add a via from INT1 to INT2, the first via in the list that meets this criteria is presented as the default via.

Edit Via List Dialog Box

Use this field . . .	To . . .
<i>Select a via from the library or the database</i>	Display all padstacks in the library or database
<i>Filter: Show vias from the library</i>	Display all padstacks in the library
<i>Filter: Show vias from the database</i>	Display all padstacks in the database
<i>Filter by name</i>	Narrow your view of the list of vias in the library or database. This field accepts regular expression syntax.
<i>Or enter a via name</i>	Add a via that is not in the library or database to the <i>Via list</i>
<i>Via List</i>	Create a working list of vias for the design
<i>Remove</i>	Remove the via from the working via list (left pane)
<i>Up</i>	Reorder the vias in the Via List
<i>Down</i>	Reorder the vias in the Via List
<i>Purge</i>	Remove vias from the Via list that are not defined in the library or database. <i>Yes</i> removes all undefined vias, in all via lists, in the design. <i>No</i> removes all undefined vias, in the via list associated with the currently selected net, in the design
<i>Viewer Options</i>	Display the <i>Draw Options</i> dialog box.
<i>Undock Viewer</i>	Undocks the <i>Via List Viewer</i> .
<i>Dock Viewer</i>	Docks the <i>Via List Viewer</i> back to the original position.
<i>Hide Viewer</i>	Hides the <i>Via List Viewer</i> .
<i>Show Viewer</i>	Displays the <i>Via List Viewer</i> .

Allegro X Constraint Manager Reference

Appendix A: Dialog Box Help

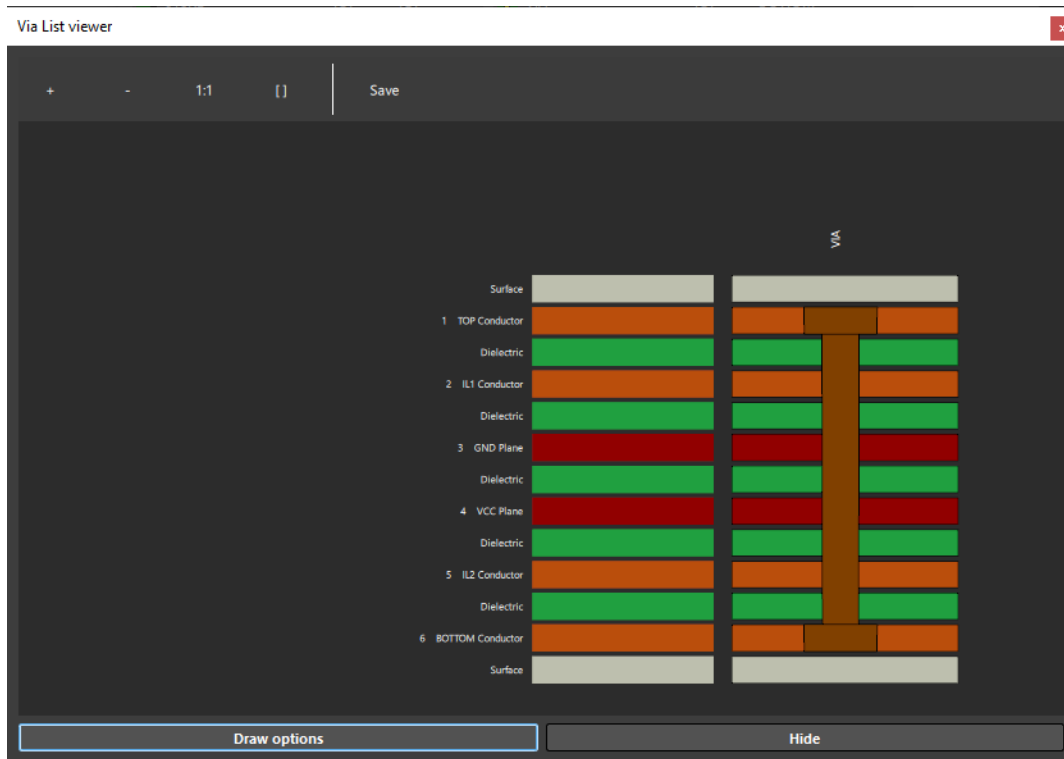


The *Via List Viewer* displays the graphical view of the vias associated with the respective Physical CSet.

Allegro X Constraint Manager Reference

Appendix A: Dialog Box Help

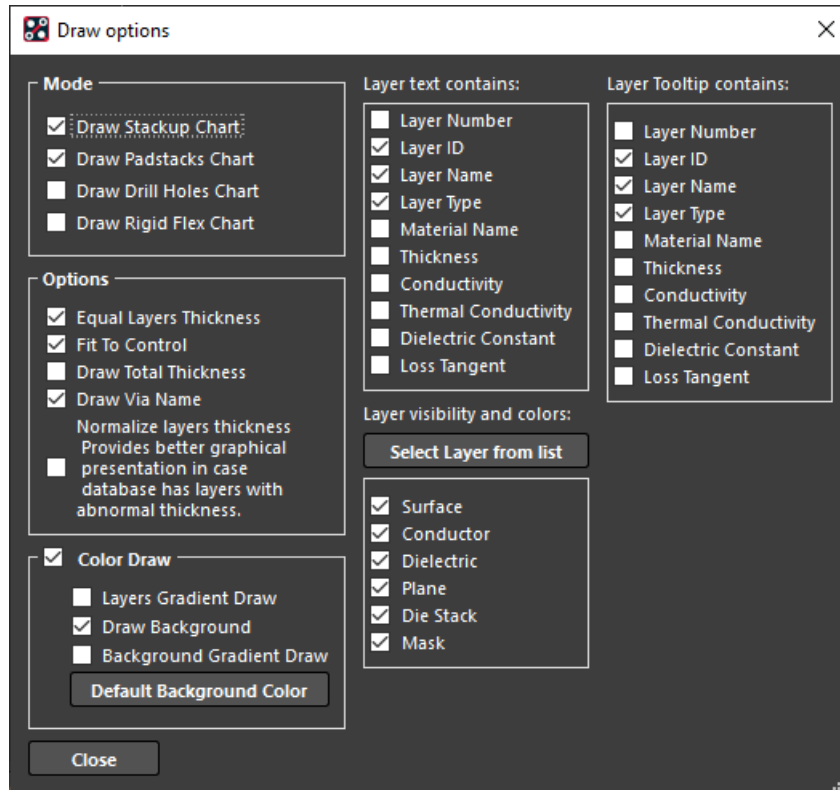
Note: The *Via List Viewer* is not available if Constraint Manager is invoked from Design Entry HDL tools.



Allegro X Constraint Manager Reference

Appendix A: Dialog Box Help

The *Draw Options* dialog box displays options for color selection, layer visibility, and tooltips.



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 by name* 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).

Edit Via Structure

Use this command to create a working list of via structures for your design. You can specify via structures from those defined in the library or database.

The list represents a selectable order of via structures used by `add connect` command. The via structure order allows an application to automatically select the first via structure in the list that meets the criteria.

Edit Via Structure List Dialog Box

Use this field . . .

Select via structure from the library or database

Start

End

Signal Paths

Return Paths

Filter: From the library

Filter: From the database

Filter by name

Via structure list

Remove

Up

Down

To . . .

Display all via structures in the library or database

Display name of the layer from where the signal path starts

Display name of the layer where the signal path ends

Display number of signal paths in the via structure

Display number of return path connections in the via structure

Display all via structures in the library

Display all via structures in the database

Narrow your view of the list of via structures in the library or database. This field accepts regular expression syntax.

Create a working list of via structures for the design

Remove the via structure from the working via structure list (left pane)

Reorder the via structures in the Via structure list

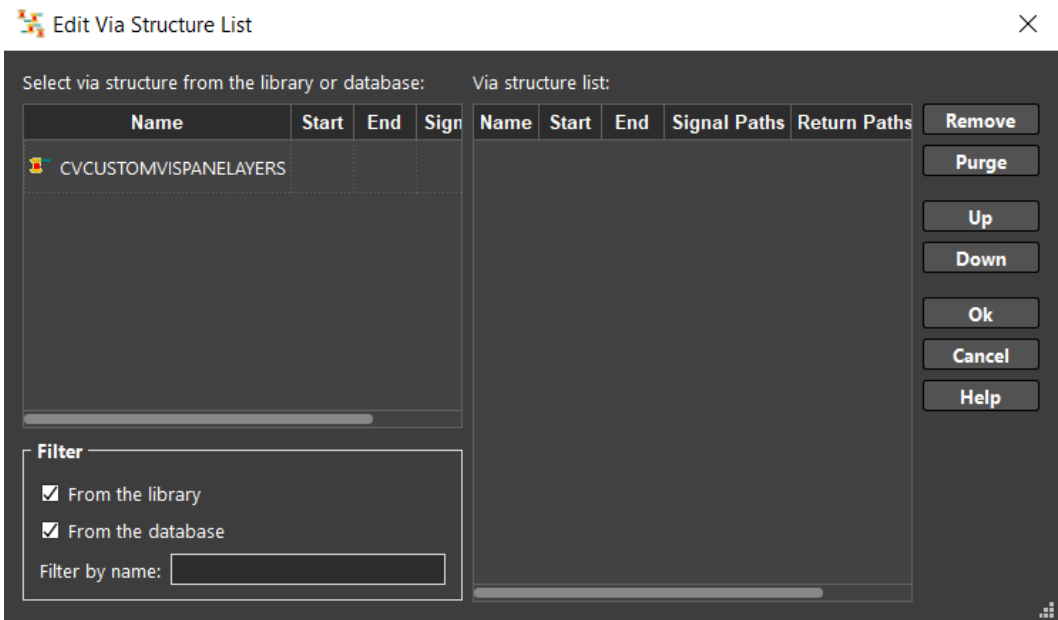
Reorder the via structures in the Via structure list

Use this field . . .

Purge

To . . .

Remove via structures from the Via structures list that are not defined in the library or database.



Edit Via Structure List Icons

The Edit Via Structure List dialog box displays icons to represent a via structure’s type. There are two types of via structures.



Standard via structure



High speed via structure

Procedures

Adding a via structure to the via structure list from the library or database

1. In *Electrical* domain, choose either

- ☐ *Vias* worksheet in the *Routing* workbook and click in the *Via Structures* cell.

--or--

- ☐ *Signal Integrity/Timing/Routing* worksheet from *All Constraints* workbook and click in the *Via Structures* cell.

Note: This option is available if High Speed product option is selected.

The *Edit Via Structure List* dialog box appears.

2. Click on a via structure in the *Select via structure from the library or database* field.



Tip

You can narrow the listing of via structures in the library or database by entering the via structure's name in the *Filter by name* field. This field requires regular expression syntax. As you type, qualifying names appear. You can also SHIFT-click to select a contiguous range of via structures or CTRL-click to select non contiguous via structures.

The via structures populates the *Via structure list* field.

3. If desired, continue to add additional via structures to the *Via structures* list.
4. Click *OK* to save your edits and dismiss the *Edit Via Structure List* dialog box.

Reordering the via structure list

1. In *Electrical* domain, select *Vias* worksheet in the *Routing* workbook, click in the *Via Structures* cell.

The *Edit Via Structure List* dialog box appears.

2. In the *Via structures list* field, click on a via structures and click *Up* or *Down* to relocate the via structures in the list.



Tip

Click *Remove* to remove the selected via structures from the list. This does not remove the via structures from the library or database.

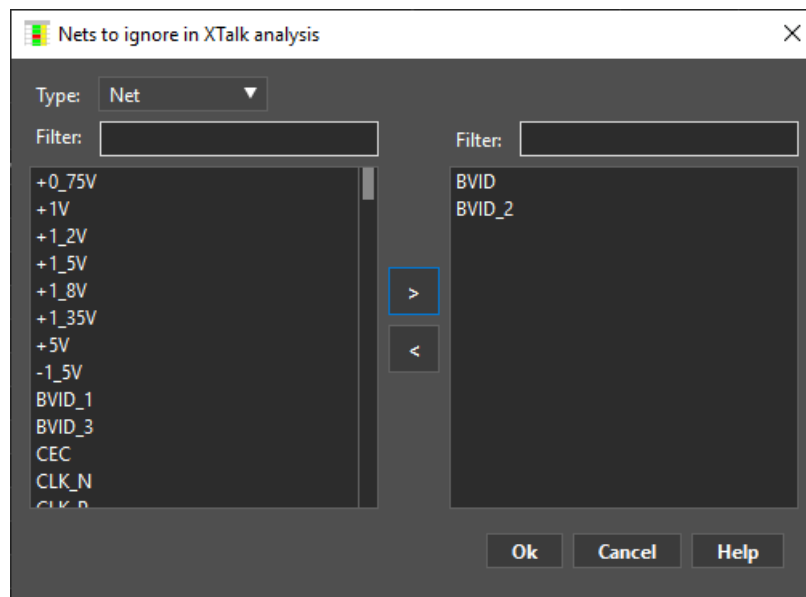
Ignore Nets

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.

For example, you can specify the following four rules:

- 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.