

# **C Commands**

**Product Version 23.1**  
**September 2023**

© 2024 Cadence Design Systems, Inc.  
Printed in the United States of America.

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

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

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

All other trademarks are the property of their respective holders.

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

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

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

1	10
C Commands	10
calc min airgap cancel capture image cd cdl out cdnshelp cdsdoc_search change change layer change radius change shape size change sym owner change void size change width check symbol chg origin class clear color clear external drcs clear pvs drcs cline change width cline to shape clpcopy clppaste cm dumpXMLForWorksheets cmgr cmgr_elec cmgr_phys cmgr_snpac cmgr_spac cmgr_xprobe cns_dummy_net cns design cns electrical cns onlinedfmdrc cns onlinedrc cns show color color192 colorview create colorview load colorview restore comp compare comp complete component assign component fix component height compose line compose shape compose symbol from geometry compress_route cond length report config package type config substrate layers confirm connect lines convert_gerber copy copy component copy fanout cputime create_devices create_sym create bounding shape create coupons create detail create device create fanout create module create net create nets create plot create short create structure create symbol create via structure csvpin in csvpin out ctab custom_route custom datatips Customize Toolbar custom smooth cut marks	10
calc min airgap	12
Calc Mins Airgap: Options Panel	13
Calculating Air Gap Between Design Objects	14
cancel	15
capture image	16
Capturing Design Canvas Image	17
cd	18
cdl out	19
CDL Netlist Out Dialog Box	19
cdnshelp	19
cdsdoc_search	19
change	20
Change Command: Options Panel	21
Changing Characteristics of Graphic Elements	22
Changing Line Width on a Line Segment	23
Changing Subclass of a Cline	24
Selecting a Padstack for Via Assignment	26
change layer	27

Changing Layer of Design Elements	28
change radius	29
Changing Radius of the Circle	30
change shape size	31
Change Shape Size Command: Options Panel	32
Changing Shape Size	33
change sym owner	34
Adding or Removing Objects from a Symbol	35
change void size	36
Change Void Size Command: Options Panel	37
Changing Void Size	38
change width	39
Changing Width of Nets and Lines	40
check symbol	41
Footprint Symbol Attribute Marker Viewer Dialog Box	42
Physical Symbol Attributes Check Dialog Box	43
Running Symbol Checks	44
chg origin	46
Changing Drawing Origin	47
class	48
clear color	49
clear external drcs	50
clear pvs drcs	51
cline change width	52
Changing Cline Width	53
Cline Change Width Command: Options Panel	54
cline to shape	55
Cline to Shape Command: Options Panel	56
Converting a Cline to Shape	57
clpcopy	58
Clpcopy Command: Options Panel	59
Copying Elements to a Clipboard File	60
clppaste	61
Clppaste Command: Options Panel	62
Pasting Elements from a Clipboard File	63
cm dumpXMLForWorksheets	64
cmgr	65

cmgr_elec	66
cmgr_phys	67
cmgr_snspace	68
cmgr_spac	69
cmgr_xprobe	70
cns_dummy_net	71
Assign Dummy Nets by Pin to NetClasses Dialog Box	72
Assigning NetClass to Dummy Pins	73
cns design	74
Design Constraints Dialog Box	74
Setting Up Spacing Constraints for Soldermask Openings	76
cns electrical	77
Assigning Electrical Constraint Sets to Nets, Differential Pairs, and Buses	78
Calculating Differential Impedance	78
Creating an Electrical Constraint Set	80
Defining Differential Pair Rules	81
Defining Net Values	82
Dialog Boxes to Manage Electrical Constraints	83
Differential Calculator Dialog Box	84
Electrical Constraints Dialog Box	87
Electrical Rule for Parallel Segments Dialog Box	89
Select Layer Sets Dialog Box	90
Setting DRC Modes for the Electrical Constraint Set	91
Tasks to Manage Electrical Constraints	92
Using Pin Delay	93
Using the Electrical Constraints Dialog Box	94
Using Z Axis Delay	95
cns onlinedfmdrc	96
Enabling Online DFM Checks	97
cns onlinedrc	98
cns show	99
Generating a Constraint Report	99
color	100
Color and Visibility Dialog Box	102
color192	103
Adding Subclasses to the My Favorites folder	104
Assigning Colors or Patterns to Subclasses	105

Assigning Custom Color and Highlighting from Nets and Net Elements	106
Assigning Stipple Patterns to Fixed Objects	107
Color Dialog Box	108
Controlling Class and Subclass Visibility	113
Customizing a Color	114
Importing a Customized Color Palette	115
Overriding Custom Colors	116
Removing the Highlight State from Nets and Net Elements	117
Saving a Customized Color Palette	118
Select Color Dialog Box	119
Setting Transparency for Shapes	120
Setting Transparency Globally	121
Tasks to Manage Colors in your Design	122
colorview create	123
Changing a Color Visibility View	124
Color Views Dialog Box	125
Creating a Color Visibility View	126
Deleting a Color Visibility View	127
colorview load	128
Loading a Color Visibility View	129
colorview restore	130
Applying the Previous Color Visibility View	131
comp	132
Using the comp Command	132
compare comp	133
Comparing Third-Party Package data with Symbol Footprint in Database	133
Component Compare Dialog Box	134
complete	136
component assign	137
Assigning Unassigned Components to Rooms	138
Moving Components Between Rooms	139
Removing Assigned Components From a Room	140
Room Assign Dialog Box	141
component fix	142
Fixing Components	143
component height	144
Adding Height Specifications to Component Keepout Areas	145

Adding Height Specifications to Place Bound Rectangles	146
Component Height Command: Options Panel	147
compose line	148
Compose Line Command: Options Panel	148
Composing Lines and Arcs into an Object	150
compose shape	151
Compose Shape Command: Options Panel	152
Composing a Shape	153
Before and After: Applying the compose shape Command	154
compose symbol from geometry	156
Compose Symbol from Geometry Dialog Box	156
Creating a Symbol Using Geometry Data Files	158
Place Die dialog box	159
compress_route	161
cond length report	162
Conductor Length Report Structure	162
Generating the Conductor Length Report	163
config package type	164
Config Package Type Command: Options Panel	164
config substrate layers	166
Defining the Substrate Layers of a Wirebond Package Design	167
Substrate Layer Config Dialog Box	168
confirm	169
connect lines	170
Connecting Lines and Arc Segments	171
convert_gerber	172
copy	173
Copy Command: Options Panel	174
Copying Elements in Radial Patterns	176
Copying Elements in Rectangular Patterns	178
Mirroring Elements on the Same Subclass	179
copy component	180
copy fanout	181
Copy Fanout Command: Options Panel	182
Copying a fanout	183
cputime	184
create_devices	185

create_sym	186
create bounding shape	187
Create Bounding Shape Command: Options Panel	188
Creating Shapes Around Boundary of Selected Objects	189
create coupons	190
Adding Coupons to a Drawing	191
Generating a Coupon	192
Select and Define Coupon Dialog Box	193
create detail	195
Creating a Detailed View	196
create device	198
Create Device Dialog Box	199
Generating Text File Data for a Device	200
create fanout	201
Create Fanout Command: Options Panel	202
Creating a Fanout	204
create module	205
Creating a Module	206
create net	207
Creating a New Net	208
create nets	209
Create List of Nets Dialog Box	210
Creating a List of Nets	211
Editing a List of Nets	212
create plot	213
Creating Plot Files for Negative Plane Layers	214
Creating Plot Files from an Active Design	215
Prerequisites for Creating Control Files (UNIX Only)	216
create short	217
Create Short Command: Options Panel	218
Creating a Short	219
create structure	220
Create Structure Dialog Box	221
Creating a Structure	224
Defining a High-Speed Via Structure	225
Defining a Standard Via Structure	226
Generating L Comp Structure	227



create symbol	228
Creating a Board Outline	229
Creating a Package Symbol Manually	230
Defining a Flash Symbol	232
Defining a Format Symbol	233
Defining a Mechanical Symbol	234
Defining Custom Pad Shapes	235
create via structure	236
Define Via Structure Dialog Box	236
csvpin in	236
CSV Pin In Dialog Box	237
Importing Symbol Pin Data from a CSV File	238
csvpin out	239
CSV Pin Out Dialog Box	240
Exporting Symbol Pin Data to a CSV File	241
ctab	242
custom_route	243
Using the Custom Route Command	244
custom datatips	245
Customizing a Datatip	246
DataTips Customization Dialog Box	247
Customize Toolbar	248
Assigning or Modifying Icon to a Command	249
Creating Your Own Toolbar	250
Customize Dialog Box	251
Deleting a Custom Toolbar	252
Hiding Toolbar Categories	253
Rearranging Buttons on Toolbars	254
custom smooth	255
Custom Smooth Command: Options Panel	256
Running Custom Smoothing	257
cut marks	258
Adding Cut Marks to a Board Outline	259
Cut Marks Options Dialog Box	260

## C Commands

calc min airgap	cancel	capture image
cd	cdl out	cdnshelp
cdsdoc_search	change	change layer
change radius	change shape size	change sym owner
change void size	change width	check symbol
chg origin	class	clear color
clear external drcs	clear pvs drcs	cline change width
cline to shape	clpcopy	clppaste
cm dumpXMLForWorksheets	cmgr	cmgr_elec
cmgr_phys	cmgr_snspace	cmgr_spac
cmgr_xprobe	cns_dummy_net	cns design
cns electrical	cns onlinedfmdrc	cns onlinedrc
cns show	color	color192
colorview create	colorview load	colorview restore
comp	compare comp	complete
component assign	component fix	component height
compose line	compose shape	compose symbol from geometry
compress_route	cond length report	config package type
config substrate layers	confirm	connect lines
convert_gerber	copy	copy component
copy fanout	cputime	create_devices
create_sym	create bounding shape	create coupons
create detail	create device	create fanout
create module	create net	create nets
create plot	create short	create structure

create symbol	create via structure	csvpin in
csvpin out	ctab	custom_route
custom datatips	Customize Toolbar	custom smooth
cut marks		

## calc min airgap

The `calc mins airgap` command calculates the minimum air gap for selected items on specified layers. You can select to calculate air gap between pins, vias, fingers, clines, lines, shapes and so on.

### Related Topics

- [Calculating Air Gap Between Design Objects](#)

## Calc Mins Airgap: Options Panel

### Access Using


- Menu path: *Display – Min Airgap*

<i>Two item mode</i>	Enables the two item mode, where you can select two items on different layers by clicking them. If not selected, you can window around items on the same layer to select them. In the two item mode, you can calculate minimum airgap between items on different layers.
<i>Layer for gap computation</i>	Specifies the layer for gap calculation.
<i>Secondary layer</i>	Specifies the layer for the second selection in two item mode.
<i>Include same net items</i>	Includes items on the same net while calculating gap.
<i>Results</i>	
<i>Items selected</i>	Displays the number of items selected.
<i>Items on layer</i>	Displays the number of selected items on the specified layers.
<i>Minimum gap</i>	Displays the minimum air gap between selected items on specified layers.
<i>Zoom display to minimum gap</i>	Zooms into the selected items if possible.
<i>Display minimum gap by object type</i>	Displays minimum air gaps by object type in the generated table.

## Calculating Air Gap Between Design Objects

To calculate air gap between design objects such as, pins, vias, fingers, clines, lines, and shapes, do the following:

1. Choose *Display – Min Airgap*.  
Alternatively, type `calc min airgap` in the Command window.
2. Configure the *Options* panel.
3. In the two item mode, click items to select them and specify the layers where the selected items are present. If not in the two item mode, window around items and specify the layer where the selected items are present.

 In both the modes (multi-select and two item), you need not reselect the layer(s) for gap calculation on the objects. For example, if you want to perform air gap calculations between each of the layers of two pins, you can select the two pins and then change the layers one after another to see the new results.

The minimum airgap is calculated and displayed under Results. A line is also shown between the objects for which the air gap is being calculated.

4. Right-click and choose *Done*.

## Related Topics

- [calc min airgap](#)

## **cancel**

The `cancel` command available on the pop-up menu when an interactive command is active. The Cancel command undoes all actions taken during the current interactive command since the last Done command was executed.

## capture image

The `capture image` command captures the screen shots of the selected part of the design canvas and save in Bitmap(\*.bmp) format. The other formats for saving an image are TIFFT (\*.tif), Jpeg (\*.jpg) and PNG (\*.png).

When you capture an image a file browser is opened to save the image at a desired location. By default, the command is saved in the working directory.

### ***Access Using***

- Menu path: *File – Capture Canvas Image*



## Capturing Design Canvas Image

To take screenshots of a selected area of a design, do the following:

1. Zoom to that part of the design which you want to capture.
2. Choose *File – Capture Canvas Image*.  
Alternatively, type `capture image` in the Command window.  
The *Capture Image* file browsers appears. By default, it shows the working directory.
3. Browse a directory location.
4. Enter name of the image file.
5. Choose an image type and click *Save*.  
The image file saves in the selected directory.

## cd

The `cd` command changes the current working directory.

### Syntax

There are three ways to use this command:

<b>cd</b>	Changes the current working directory to the directory that opens when you log in.
<b>cd ..</b>	Changes the current working directory to the next higher level directory.
<b>cd &lt;directory_name&gt;</b>	Changes the current working directory to the directory you specify. If the directory is not a subdirectory of the current working directory, you must provide the full path name.

### Examples

```
cd design
```

```
cd /home/abc/proj1
```

## cdl out

 Available only in Allegro X Advanced Package Designer (APD) with the *Silicon Layout* option.

The `cdl out` command exports the logic of the design as a CDL (circuit design language) netlist. The command opens the CDL Netlist Out dialog box. You can use the CDL netlist to run layout versus schematic (LVS) checks using Cadence® Physical Verification System (PVS).

### CDL Netlist Out Dialog Box

#### Access using:

- Menu path: *Si Layout - CDL Netlist Out*.
- Command: `cdl out`

Field	Description
<i>File Name</i>	Name of the CDL netlist. By default, the file is saved in the design folder with the same name as the design and <code>.cdl</code> extension.
<i>Top Cell Name</i>	Displays the name of an existing top cell of your design, which defines your design hierarchy.
<i>Add port pad (shape) on Component Geometry</i>	Select to add port pad on component geometry. Not selected by default.
<i>Add port label on Component Geometry</i>	Select to add port label on component geometry. Not selected by default.
<i>Create resistor elements for pins/vias</i>	Click in the <i>Resistor</i> column for a listed component to create resistor elements for the pins or vias of that component. No elements are selected by default.
<i>Generate</i>	Click to generate the CDL netlist.

## cdnshelp

The `cdnshelp` command brings up Cadence Doc Assistant that displays online documentation.

#### Access Using

- Menu path: *Help – Documentation*

## cdsdoc\_search

The `cdsdoc_search` command lets you search any word string in the Cadence Doc Assistant.


This command invokes a dialog box with a type-in field where you can specify a search string. The entire installed documentation is searched to produce the results.

### ***Access Using***

- Menu path: *Help – Search*

## **change**

The `change` command alters elements in a design. You can change line width, text size and justification, or the subclass to which an element is assigned.

 When using Allegro X Advanced Package Designer (APD), if you change the layer of an attached wire or cline, bondpads are no longer ripped up.

### **Related Topics**


- [Selecting a Padstack for Via Assignment](#)
- [Changing Characteristics of Graphic Elements](#)
- [Changing Line Width on a Line Segment](#)
- [Changing Subclass of a Cline](#)

## Change Command: Options Panel

### Access Using

- Menu path:
  - *Edit – Change* (Allegro X PCB Editor, APD)
  - *Route – Change* (Allegro SI)

The `change` command changes the characteristics of specific types of elements based on the settings in the *Options* panel.

 If you do not know an element's class, run the `show element` command to display the class, subclass, and other information about the element.

Fields that display color boxes to the left let you change the display color for that subclass while you complete your change actions.


<i>Class</i>	Must match the class designation of the element(s) that you are changing.
<i>New subclass</i>	Specifies the new subclass of the chosen element. Must be a valid subclass of the specified class.
<i>Act via</i>	Specifies the active via—that is, the via to use when switching etch/conductor to another subclass, if a via is necessary. The layout editor uses this field if a via is needed to maintain connectivity when connections are moved to a different subclass. The via must be an existing padstack definition. You can either enter a via name in the field or choose <i>Browse</i> and choose it from the Select a Padstack dialog box.
<i>Line width</i>	Specifies the width, in user-specified units, to which a chosen line changes.
<i>Text block</i>	Specifies the number of the text block to determine the parameters for the chosen text. This field lets you to change any element that is composed of text. To check the parameter values in all text blocks, use the Text Setup dialog box, available by choosing <i>Setup – Design Parameters</i> <a href="#">prmed</a> , the <i>Design</i> tab of the Design Parameter Editor, and <i>Setup Text Sizes</i> . You can also access the Text Setup dialog box by running the <code>define text</code> command.
<i>Text name</i>	Specifies the name of the text block.
<i>Text just</i>	Specifies how the chosen text should be aligned relative to the text marker. This field lets you change any element that is composed of text.

### Related Topics

- [Changing Characteristics of Graphic Elements](#)
- [Changing Line Width on a Line Segment](#)
- [Changing Subclass of a Cline](#)
- [prmed](#)
- [define text](#)

## Changing Characteristics of Graphic Elements

1. Choose *Edit – Change*.  
Alternatively type `change` in the Command window.
2. Configure fields in the *Options* panel.
3. In the Find panel, ensure that the type of design object(s) you want to change are enabled.

 If *Nets* are enabled, all connect lines and filled rectangles (if applicable) are changed.

4. Pick the elements you are changing in any of these ways:
  - Choose on one element in the design.
  - Choose *Temp Group* from the pop-up menu. Click each item in the temporary group or drag the cursor to choose objects that are next to each other. Then choose *Complete* from the pop-up menu.
  - Drag the cursor over a group of objects to choose them.

The chosen elements change according to their type and your settings in the *Options* panel.

1. Right-click and choose *Done*.

## Related Topics

- [change](#)
- [Change Command: Options Panel](#)
- [Changing Subclass of a Cline](#)

## Changing Line Width on a Line Segment

When you use the `change` command, the *Cut* option, available on the pop-up menu, enables you to apply a line width change on only a part of a line segment.

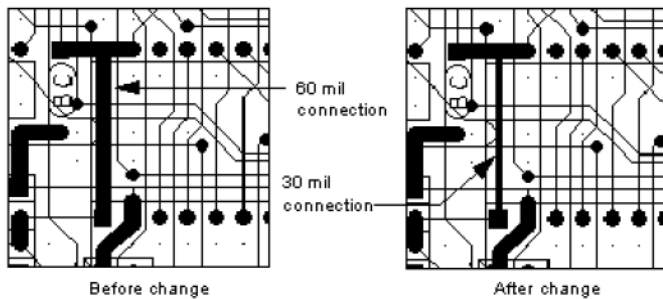
⚠ You can change the line width of an entire connect line and even all connect lines on a net. In those cases, you would not use the *Cut* option.

1. Choose *Edit – Change*.  
Alternatively type `change` in the Command window.
2. Configure fields in the *Options* panel.
3. In the Find panel, ensure that only *Cline Segs* is enabled.
4. Right-click and choose *Cut*.
5. Choose the start and end points on the line.

The line segment changes to the width that you choose in the *Options* panel.

The following example illustrates a change of line width from 30 mils to 60 mils to perform a DRC check to detect design rule violation additions and deletions.

Example of Changing Line Width



## Related Topics

- [change](#)
- [Change Command: Options Panel](#)
- [Selecting a Padstack for Via Assignment](#)

## Changing Subclass of a Cline

1. Choose *Edit – Change*.  
Alternatively type `change` in the Command window.
2. Set the Class field on the *Options* panel to *Etch/Conductor*.
3. In the *New subclass* box of the *Options* panel, specify the layer to which you are moving the connection.
4. If a via is necessary, enter its name in the *Act via* box. A via is necessary if the element that the cline or segment to which it connects is not on the new subclass.  
The layout editor uses this via if it is legal. If it is not a legal via, or you do not specify a via, the layout editor chooses another one using this criteria:
  - The via from the Current Via List for the chosen net that connects the two layers and spans the fewest number of layers is considered the best fit.
  - If multiple vias match the best-fit criteria, the layout editor uses the first appropriate one it finds on the Current Via List for the chosen net.
  - The Current Via List is located in the Physical worksheet of Constraint Manager (`cmgr_phys` command), under the *Vias* column heading.
  - If a different via is used than the one defined in the *Options* panel, the layout editor issues the following message, but does not change the *Options* panel:

Padstack <padstack name> used for via at (<x\_location>, <y\_location>)

If the connect line being changed spans two constraint areas, the command selects each required via separately, if the via you chose is illegal.

If no legal via is available, the operation is not completed and the following message appears:

No available via to connect to subclasses at (<x\_location>, <y\_location>)

5. In the Find panel, enable the design elements you are changing:
  - *Nets* moves all connect lines for that net to the new layer.
  - *Clines* moves the entire connect line to the new layer.
  - *Cline Segs* moves a chosen connect line segment to the new layer.
6. If you are moving a net, choose on any cline that is part of it. –or– If you are moving a cline, choose on it. –or– If you are moving just a line segment, choose *Cut* from the pop-menu and choose the beginning and ending points on the line.

The net, cline, or segment is highlighted.

1. Right-click and choose *Done*.

The following example shows steps to change subclass of a cline segment. To maintain connectivity, the `change` command adds a via at each end.

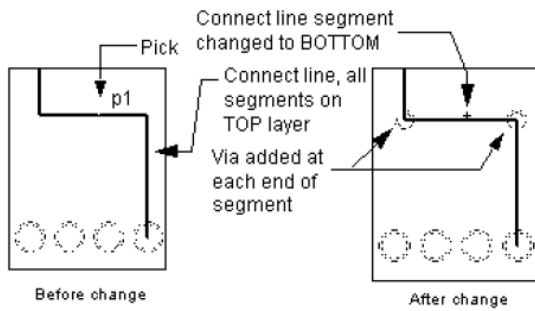
1. Run the `change` command.
2. In the *Options* panel, set the following fields:
  - *Class* to *ETCH/CONDUCTOR*
  - *New subclass* to *BOTTOM*

You could also choose a via in *Act Via*, but it is not necessary because the layout editor selects the one that best fits the connection.

1. In the Find panel, choose *All Off* and check *Cline Segs*.
2. Pick the horizontal segment at p1.

Example of Changing Subclass





## Related Topics


- [change](#)
- [Change Command: Options Panel](#)
- [Selecting a Padstack for Via Assignment](#)
- [Changing Characteristics of Graphic Elements](#)

## Selecting a Padstack for Via Assignment

You can use Select a Padstack dialog box to find and choose a padstack for via assignment when modifying the routing for a design element. All padstacks are listed in alphabetical order.

To choose a padstack,

1. Type the name of the padstack in the search field.  
Or highlight it in the list.  
Or enter a string followed by in the search field an asterisk (\*) and choose *OK*. For example, a search string of *MTG\** returns all elements beginning with MTG. The layout editor remembers the last search.
2. Choose *Library* to display the elements in the library. On reopening the dialog box, this mode remains selected for the duration of the design session, or until you deselect it. By default *Database* is selected. The library mode may include items already in the design because database items remain in the list when the *Library* option is chosen.

 If an object in the database has the same name as an object in the library but contains different content, the database object takes precedence in the list.

3. Click *OK*.

## Related Topics

- [change](#)
- [Changing Line Width on a Line Segment](#)
- [Changing Subclass of a Cline](#)

## change layer

The `change layer` command lists all the etch/conductor layers in the database, from which you may quickly choose a new layer on which elements are to reside. If the chosen layer is not visible, it becomes visible once you choose it.

This command functions in a pre-selection use model, in which you choose an element first, then right choose and execute the command.

Valid elements are:

- Nets
- Clines
- Line segments
- Cline segments
- Lines
- Shapes
- Text

### Changing Layer of Design Elements

1. Hover the cursor over an element or draw a window around the elements whose layer is to be changed.  
The layout editor highlights the element and a datatip identifies its name.
2. Right-click and choose *Change To Layer* from the pop-up menu.  
Alternatively, type `change layer` in the Command window.
3. Choose a layer from the list of etch/conductor layers that appears.  
The element moves to the new layer.

## change radius

The `change radius` command lets you change the radius of the specified circle.

This command functions in a pre-selection use model, in which you choose an element first, then right choose and execute the command. Valid objects are line circles and cline circles.

## Changing Radius of the Circle

To modify the radius of a circle do the following:

1. Hover the cursor over a circle.  
The layout editor highlights the object and a datatip identifies its name.
2. Right-click and choose *Change Radius* from the pop-up menu.
3. Enter new value for the radius.  
The radius of the circle changes.

## change shape size

The `change shape size` command lets you expand or contract a shape by a defined value.

### Related Topics

- [Changing Shape Size](#)

### Change Shape Size Command: Options Panel

<i>Expand/Contract</i>	Choose to increase or decrease the size of a shape. Enter the value in mils to expand or contract the void size. Use +/- to incrementally change the shape size.
------------------------	--



## Changing Shape Size

To expand or contract a shape, do the following:

1. Hover the cursor over a shape. Click to select the shape.
2. Right-click and choose *Expand/Contract*.  
Alternatively, type `change shape size` in the Command window.
3. Set value in the *Options* panel.
4. Use +/- buttons to expand or contract the shape.
5. Choose *Done* from the pop-up menu.

## Related Topics

- [change shape size](#)

## change sym owner

The `change sym owner` command is used either to add database objects as children of a specified symbol instance or to remove current children from their owning parent symbol instance. Note that only static shapes can be added as children.

Any operation, such as move or delete, performed on the parent symbol instance will also occur on the children.


### ***Access Using***

- Menu path: *Edit— Change Symbol Owner*

## Adding or Removing Objects from a Symbol

To add or delete design objects from a symbol instance, do the following:


1. Choose *Edit—Change Symbol Owner*  
Alternatively type `change sym owner` in the Command window.
2. Select the symbol for adding or removing object.
3. Right-click and choose the option to either *Add to symbol* or *Remove from symbol*.

 To be able to add an object owned by one symbol to another symbol, you must first remove the object from the current parent symbol.

4. Select the object to be added or removed.
5. Right-click and choose *Next* to perform change symbol owner for another object or choose *Done*.

## change void size

The `change void size` command lets you expand or contract a user-defined void outline by a defined value. You can use this command in General Edit application mode to change the size of a user-defined void.

 For dynamic shapes the boundary subclass must be visible.

### Related Topics

- [Changing Void Size](#)

### Change Void Size Command: Options Panel

<i>Expand/Contract</i>	Choose to increase or decrease the size of a void. Enter the value in mils to expand or contract the void size. Use +/- to incrementally change the void size.
------------------------	--

## Changing Void Size

To modify the shape of a void, do the following:

1. Hover the cursor over a void.  
The layout editor highlights the void and a datatip identifies its name. The boundary subclass is visible the datatip of a user-defined void.
2. Right-click and choose *Expand/Contract*.  
Alternatively, type `change void size` in the Command window.
3. Set value in the *Options* panel.
4. Use +/- buttons to expand or contract the void.
5. Choose *Done* from the pop-up menu.

## Related Topics

- [change void size](#)

## change width

The `change width` command displays the *Change Width* dialog box to quickly modify the width of specified elements. When you choose a net, all its clines change to the new width. The default width is that of the first element chosen.

This command functions in a pre-selection use model, in which you choose an element first, then right choose and execute the command. Valid elements are:

- Nets
- Clines
- Line segments
- Cline segments


### Changing Width of Nets and Lines

1. Hover the cursor over an element or draw a window around the elements whose layer is to be changed.  
The layout editor highlights the element and a datatip identifies its name.
2. Right-click and choose *Change Width* from the pop-up menu.  
Alternatively, type `change width` in the Command window.
3. Enter a new width in the *Change Width* dialog box that opens.
4. Click *OK* to change the width on the specified elements.



## check symbol

The `check symbol` command opens the *Physical Symbol Attributes Check* dialog box, where you can run predefined checks on physical symbols. You can customize this command by writing your own symbol rules using AXL-SKILL, the SKILL language extension for use with Cadence tools. This command reads the `rules_check_table.il` file.

 This command is available only in symbol editor mode.

### Related Topics

- [Defining and Developing Libraries](#)
- [Running Symbol Checks](#)

## Footprint Symbol Attribute Marker Viewer Dialog Box

This dialog box displays the contents of the `<design_name>_symchk.mkr`, stored in the current directory. Information appears here for checks that are not successful and for rules that require reporting data.

There are four columns, with a maximum of 100 rows in the grid with titles for each column. Each row is a line of results from the checks that were run.

The information in each row is contained in the following four columns:

<i>Short Message</i>	Gives a brief description of the listed item and why it is listed.
<i>Object</i>	Indicates the name or type of object that is being reported—for example, <i>VIA</i> , <i>PIN</i> , and <i>TEXT</i> .
<i>XY Location</i>	Provides the X and Y coordinates of the item.
<i>Class/Subclass</i>	Lists the class and subclass of the item.

The messages are color-coded to help you quickly recognize item status:

- Red: Error message
- Yellow: Warning message
- Green: Informational data

Choosing one of the rows (not the numbers of the rows) displays more details about that item in the box below the table.

## Physical Symbol Attributes Check Dialog Box

### Access Using

- Menu path: *Tools – Check Symbol*

This dialog box displays a tree structure. Each node of the tree is a set of rules of a specific type, such as REFDES, TOLERANCE and so on. These rules are read from the SKILL file `rule_check_tables.il` file, which you can edit to customize the rules listed.

<i>Run</i>	Performs a check against the rule(s) you select. You can choose the <i>All Checks</i> option, a particular type of rule (choose the folder for that group of rules), or individual rule(s).
<i>View Markers</i>	Displays the Footprint Symbol Attribute Marker Viewer dialog box, which lists information for checks that are not successful and for rules that require reporting data.
<i>View Log</i>	Displays the Symbol Check Log File window, which displays the <code>&lt;design_name&gt;_symchk.log</code> , located in the current directory. It summarizes the results of the symbol check.

## Running Symbol Checks

Once you start the `check symbol` command, these are the tasks you can do:

- [Running Checks against Rules](#)
- [Viewing the Log File](#)
- [Viewing the Marker File](#) to identify issues or problems in a symbol drawing

### Running Checks against Rules

Perform the following steps to run a check a symbol against design rules:

1. In PCB Librarian Expert, open a package or mechanical symbol.
2. Run the `check symbol` command.

On a chosen symbol, you can do one of the following:

- Run the check against all the rules.
- Run the check against a particular type of rule, such as checks against the REFDES rules. The REFDES rules dictate that each physical symbol must have a corresponding reference designator, and for each reference designator there should be a text block size and layer definition.
- Run a check against a specific rule from within the set of rules. For example, you can run the *COMPONENT VALUE Exist* check from within *COMPONENT VALUE Checks* and the *USER PART NUMBER Exist* check from within *USER PART NUMBER Checks*.

As each check is completed, a message in the console window displays its status.

After the checking routines are completed, you can view the log file or markers file, or exit the command by clicking *OK*.

### Running Checks against All Rules

- In the Physical Symbol Attributes Check dialog box, make sure that the *All Checks* option is chosen and choose *Run*.


When you execute the `check symbol` command for the first time, the *All Checks* option is chosen by default.

### Running a Particular Type of Check

1. In the Physical Symbol Attributes Check dialog box, deselect the *All Checks* option.
2. Choose the set of rules against which you want to run the check and choose *Run*.

### Running Checks Against a Specific Rule within a Set of Rules

1. In the Physical Symbol Attributes Check dialog box, expand the set of rules by either double-clicking on the folder or clicking on the + sign next to it.
2. Choose the rule(s) you want to use for the check and choose *Run*.

 You can choose multiple rules from different sets of rules. For example, you can the *TOLERANCE Exist* rule from the *TOLERANCE Checks* set of rules and the *SILK SCREEN Geometry* rule from the *GEOMETRY Checks* set of rules.

### Viewing the Log File

The `<design_name>_symchk.log` file records the results of the symbol check. It is stored in the current directory. After the checks are run, you can view the log file in one of the following ways:

- In the Physical Symbol Attributes Check dialog box, choose *View Log*.
- Use any text editor to view the log.

### Viewing the Marker File

If a check is unsuccessful, or the specific rule calls for reporting data, information for that rule is put into a marker file named `<design_name>_symchk.mkr`. This is a formatted file which is used by the marker viewing option to aid you in the identification of issues or problems in a symbol drawing.

You can view the marker file in one of the following ways:

- In the Physical Symbol Attributes Check dialog box, choose *View Markers*. The [Footprint Symbol Attribute Marker Viewer Dialog Box](#) appears.
- Use any text editor to view the marker file.

## Related Topics

- [check symbol](#)

## chg origin

The `chg origin` command lets you specify an exact point on the canvas as the location for the drawing origin. A circle with crosshairs designates the drawing origin.

To display the origin, enable the *Display Origin* parameter in the *Display* tab of the Design Parameter Editor, available by choosing *Setup – Design Parameters* (`prmed` command).

### ***Access Using***

- Menu path: *Setup – Change Drawing Origin*

## Changing Drawing Origin

To change the drawing origin in the design canvas, do the following:

1. Choose *Setup – Change Drawing Origin*.

Alternatively, type `chg origin` in the Command window.

The following message is displayed:

Pick new location for drawing origin

2. Choose a new location for the drawing origin either by left clicking or typing an existing XY location at the console command window. A circle with crosshairs designates the new location of the drawing origin.

✓ Consider using the right mouse button pop-up menu option *Snap pick to*, which snaps the origin to a database element such as a tooling hole or intersection.

## class

The `class` command changes the class field in the *Options* panel. The class name must be one of the classes specified in the layout editor.

### Syntax

```
class[--+] [--] [class]
```

<b>-+</b>	Increments to the next class.
<b>--</b>	Decrements to the previous class.
<b>class</b>	Indicates the name of the class to which you are changing.

### Examples

- Change the class to the *Etch/Conductor* class.

```
class Etch
```

- Increments to the next class.

```
class -+
```

- Uses the [funckey](#) command to create a function alias that increments to the next class.

```
funckey cl class -+
```




## **clear color**

An internal command.

## clear external drcs

The `clear external drcs` command removes all the DRC markers and associated geometry information imported using the `import external drcs` command. The command also removes the manufacturing class layers where the geometry information for the DRC markers was present, after the layers are empty.

 Available when the *Silicon Layout* option is selected in Advanced Package Designer.

This command provides a quick way to check the rules your design still violates. After all the `EXT_` layers are removed by this command, the next time you import external DRC markers, only the layers with active violations on them are created. The smaller the set of layers, the fewer rules the design violates.

### Access Using


- Menu Path: *Si Layout – DRC – Clear External DRCs*

### Related Topics

- [browse\\_drcs](#)

## clear pvs drcs

The `clear pvs drcs` command removes all the DRC markers and associated geometry information imported using the `import pvs drcs` command or the interactive `pvs` command. The command also removes the manufacturing class layers where the geometry information for the DRC markers was present, after the layers are empty.

 Available when the *Silicon Layout* option is selected in Advanced Package Designer.

This command provides a quick way to check the rules your design still violates. After all the `PVS_` layers are removed by this command, the next time you import PVS DRC markers, only the layers with active violations on them are created. The smaller the set of layers, the fewer rules the design violates.

### Access Using

- Menu Path: *Si Layout – DRC – Clear PVS DRCs*

### Related Topics

- [browse\\_drcs](#)

## cline change width

The `cline change width` command allows you to increase or decrease the width of specified routing cline segments. The layout editor creates a new vertex at the location of the width change. The layout editor does not check for DRCs during the width change operation, but does report any new violations as online DRC markers.

This command does not apply to non-conductor (geometry) lines, bond wires, and so on.

### Related Topics

- [Changing Cline Width](#)

## Changing Cline Width

To increase or decrease the width of routing cline segments, do the following:

1. Choose *Edit – Change Cline Width*.  
Alternatively, type `cline change width` in the Command window.  
The parameters are displayed in the *Options* panel.
2. Choose either *Single trace mode (two-pick)* or *Multi-trace mode (window)*.
3. To change the width, you can perform any one of the following:
  - Set the value for the new line within the *New width* box. This will change the width of the selected cline segment to the new line width.
  - Select *Maximize line width* to set the selected line width to the maximum possible without any errors.
  - Select *Remove selected part of cline(s)* to remove selected region of clines.
4. Make your selections in the drawing canvas.  
If you choose *Single trace mode (two-pick)*, pick the first point on the cline and then the second point. The length of the clines between these two points (following the cline path) changes.  
If you choose *Multi-trace mode (window)*, mark the two corners of the region with two picks.  
If the pick is not in the desired location, use the *Oops* option on the right-choose pop-up menu.
5. Right-click and choose *Done*.

## Related Topics

- [cline change width](#)

## Cline Change Width Command: Options Panel

### Access Using

- Menu path: *Edit – Change Cline Width*

<i>Single trace mode (two pick)</i>	Choose this button to pick two locations on a single cline and change the width of the cline that is between the two specified points to the new value you specify. This is the default setting.
<i>Multi-trace mode (window)</i>	Choose this button to draw a selection window in the main canvas and change all the clines inside that region to the new width that you specify.
<i>New width</i>	Specifies the new width for the modified pieces of the clines. This value defaults to the minimum line width on the TOP CONDUCTOR layer in the design.
Maximize line width	Check this option to have the system compute, and use, the maximum line width possible that will not cause DRC violations.
Remove selected part of cline(s)	Check this option to remove selected parts of one or more clines.

## cline to shape

The `cline to shape` command converts clines to shapes.

### Related Topics

- [Converting a Cline to Shape](#)

## Cline to Shape Command: Options Panel

### Access Using

- Menu path: *Tools – Convert – Cline / Line to Shape*

<i>Remove converted objects</i>	Select this option to remove converted objects and only retain the shapes. Selected by default.
<i>Retain nets on shapes</i>	Select this option to retain the nets connected to converted objects on the shape. Selected by default.
<i>Create dynamic shapes</i>	Select this option to specify that the shapes are to be dynamic.
End cap style	Choose whether you want the end caps of the created shape to be <i>Square</i> , <i>Octagon</i> , or <i>Round</i> .
Filter by line width	Select this option to specify minimum and maximum line width for objects that should be converted.
Create shape on line/cline layer	Select this option to create the shape on the same layer as the objects that are being converted. Selected by default. You can create the shape on a different layer by deselecting this option.



## Converting a Cline to Shape

To convert clines in your design to shapes, do the following:

1. Choose *Tools – Convert – Cline / Line to Shape*.  
Alternatively, type `cline to shape` in the Command window.  
The parameters are displayed in the *Options* panel.
2. Configure the *Options* panel.
3. Select the clines to convert to shapes.
4. Choose *Done* from the pop-up menu.

## Related Topics

- [cline to shape](#)

## clpcopy

The `clpcopy` command copies design elements from designs or symbol drawings to a clipboard (`.clp`) file for pasting into designs and drawings. You can also create a library of clipboard files, described in *Defining and Developing Libraries* in the user guide.

### Related Topics

- [Copying Elements to a Clipboard File](#)
- [clppaste](#)

## Clpcopy Command: Options Panel

### Access Using


- Menu path: *File – Export – Sub-Drawing*

<i>Preserve Refdes</i>	Choose to preserve the reference designator information in the clipboard file.
<i>Preserve Nets on Shapes</i>	Choose to allow shapes to retain their net associations when a .clp file is read in. Only databases with nets support this capability as symbols do not support the preservation of nets.
<i>Preserve Nets of Vias</i>	Choose to allow vias to retain their net associations when a .clp file is read in. Only databases with nets support this capability as symbols do not support the preservation of nets.
<i>Preserve Testpoints on Vias</i>	Choose to allow vias to retain their testpoint associations when a .clp file is read in.

## Copying Elements to a Clipboard File

If you copy a dynamic shape to another dynamic layer, any shape override properties attached to it are not maintained. For additional related information on working with dynamic shapes, see [Preparing the Layout in the user guide](#).

1. Choose *File – Export – Sub-Drawing*.  
Alternatively, type `clpcopy` in the Command window.
2. Choose elements for the clipboard file in the Find panel. Typically, you can only choose visible elements. The exceptions are:
  - Pins or vias. If any part of a pin or via is visible, all applicable information for the pin or via is chosen.
  - Symbols. All applicable elements of symbols are chosen.
3. Specify in the *Options* panel whether you want to preserve the reference designator information (*Preserve Refdes*) in the clipboard file.
4. Specify in the *Options* panel whether you want to allow shapes to retain their net associations in the clipboard file in the *Preserve Nets on Shapes* field.
5. To copy multiple elements to the clipboard file, do one of the following, otherwise proceed to [step 6](#).

 The clipboard copy command does not consider pin escapes (clines and vias) as part of the package/part symbol. To copy a package/part symbol with its pin escapes to a clipboard file use the select by group or window options to include the clines and vias.

To choose all elements within a specific area:

- a. Drag the cursor over the elements you want to select.
- b. Choose a coordinate in the work area as the origin point for the elements.
- c. Proceed to [step 6](#).

To choose only specific elements:

- a. From the pop-up menu, choose *Group*.
- b. Choose each element you want to copy.
- c. When you have chosen all the specified elements, choose *Complete* from the pop-up menu.
- d. Choose a coordinate in the work area as the origin point for the elements.  
The layout editor displays the coordinates of the last pick and displays the Clipboard Filename dialog box.
- e. Proceed to [step 6](#).

6. In the *Clipboard Filename* dialog box, enter the file's name and choose *OK*.
7. If you do not enter a filename, the layout editor uses the default `standard.clp`. If you enter a filename without an extension, the layout editor adds the default extension `.clp`.  
If you enter the name of an existing clipboard file, the layout editor displays a confirmation window and asks if you want to overwrite the existing file. Choose *Done* from the pop-up menu to store the element in the clipboard file.

## Related Topics

- [clpcopy](#)

## clppaste

The `clppaste` command pastes design elements from a clipboard (`.clp`) file into designs or drawings.


### Related Topics

- [clpcopy](#)
- [Preparing the Layout](#)
- [Pasting Elements from a Clipboard File](#)

## Clppaste Command: Options Panel

### Access Using

- Menu path: *File – Import – Sub-Drawing*

<i>Rotate angle increment</i>	Defines the angle of rotation for placing the clipfile's elements in the design when you choose <i>Rotate</i> from the pop-up menu before pasting the elements.
<i>Clip_drawing</i>	<p>Defines a CLIP_DRAWING name to identify the element or group of elements that make up the clipfile. When you paste elements from a clipboard, the layout editor attaches a CLIP_DRAWING property to each item (except text). The default property value is CLIP_<i>n</i>, where <i>n</i> represents the number of times the paste operation has been performed on the drawing. You can override the default value by entering a name here. The value can have no more than 32 characters. The exclamation point (!) and single quotation marks (') are not allowed.</p> <div> The layout editor maintains the current CLIP_DRAWING property value after you save, exit, and re-open the drawing.</div>
<i>Assign Refdes</i>	<p>Specifies whether you want to include reference designator information from the clipfile. This option appears only if the <i>Preserve Refdes</i> option was chosen when the clipfile was created.</p> <ul style="list-style-type: none"><li>• If a reference designator pasted into the design causes a duplicate, the layout editor does not assign the reference designator. Instead, it places a message in the <code>clipboard.log</code> file indicating the reference designator in conflict and the X,Y location of the pasted symbol.</li><li>• If the reference designator pasted into the drawing is not in the logical data, a message appears in the <code>clipboard.log</code> file indicating the reference designator could not be assigned and the X,Y location of the pasted symbol.</li></ul>
<i>Preserve Nets on Shapes</i>	Choose to allow shapes to retain their net associations when a <code>.clip</code> file is read in. Only databases with nets support this capability as symbols do not support the preservation of nets. This option appears only if the <i>Preserve Refdes</i> option was chosen when the clipfile was created.

## Pasting Elements from a Clipboard File

If you paste a dynamic shape to another dynamic layer, any shape override properties attached to it are not maintained. (For additional related information on working with dynamic shapes, see [Preparing the Layout](#) in the user guide.)

1. Run the `clppaste` command.
2. In the Select Subdrawing to import dialog box, enter or choose the name of the clipboard file whose contents you want to paste, and choose *OK*.  
A rectangle representing the boundaries of the elements in the clipboard file appears attached to the cursor, and the *Options* panel displays the clppaste controls.
3. Enter settings in the *Options* panel.
4. Position the cross-hair cursor where you want the origin point of the pasted element(s) to be, then choose to paste.  
The layout editor:
  - Performs a DRC check on the new elements.
  - Writes the `clipboard.log` file to the current directory.
  - Displays the Clipboard Log File window if any warning or error messages resulted from the paste operation.

## Related Topics

- [Clpcopy Command: Options Panel](#)
- [clppaste](#)

## cm dumpXMLForWorksheets

The `cm dumpXMLForWorksheets` command dumps the worksheets of Constraint Manager domains to a XML file. This command can only be used in a script file.

Before using the command in a script, ensure that:

- Constraint Manager is opened in the script.
- Focus of the window is set to Constraint Manager.

### Syntax

```
cm dumpXMLForWorksheets <path_to_xml>/<prefix> "<domain_name>" clearFilters
```

<b>path_to_xml</b>	Specify the path where you want the command to generate the XML files.
<b>prefix</b>	Specify a prefix that is attached to the name of the XML files.
<b>domain_name</b>	Specify the domain names for which you want the command to generate the XML files. You can specify multiple domains separated by comma. If not specified, XML files are generated for all the domains.
<b>clearFilters</b>	(Optional)Clears all the filters from the worksheets before dumping the XML files.

### Examples

Following is an example of a script that uses this command and generates two files named *data\_Electrical.xml* and *data\_Same Net Spacing.xml* at the location *C:\xmls*:

```
setwindow pcb

cmgr

setwindow cmgr

cm dumpXMLForWorksheets C:\xmls\data "Electrical,Same Net Spacing"

cm exit

exit
```

### Related Topics

- [script](#)
- [cmgr](#)



## cmgr

The `cmgr` command opens Constraint Manager, a spreadsheet tool where you create and modify electrical, physical, and spacing constraints for use in this tool. For designs that require less complex electrical constraints, you can also use the `cns electrical` command to set electrical constraints.

### Access Using

- Menu Path: *Setup – Constraints – Constraint Manager*
- Toolbar Icon:



### Related Topics

- [Working with Constraints](#)
- [cns electrical](#)

## cmgr\_elec

The `cmgr_elec` command opens the *Electrical* worksheet in Constraint Manager, where you create and modify electrical constraints that apply to elements in your design.

### ***Access Using***

- Menu Path: *Setup – Constraints – Electrical*

### **Related Topics**

- [Working with Constraints](#)

## cmgr\_phys

The `cmgr_phys` command opens the *Physical* worksheet in Constraint Manager, where you create and modify physical constraints that apply to objects in your design.

### ***Access Using***

- Menu Path: *Setup – Constraints – Physical*

### **Related Topics**

- [Working with Constraints](#)

## cmgr\_snspac

The `cmgr_snspac` command opens the *Same Net Spacing* worksheet in Constraint Manager, where you create and modify spacing constraints that apply to objects on the same net in your design.

### ***Access Using***

- Menu Path: *Setup – Constraints – Same Net Spacing*

### **Related Topics**

- [Working with Constraints](#)

## cmgr\_spac

The `cmgr_spac` command opens the *Spacing* worksheet in Constraint Manager, where you create and modify spacing constraints that apply to objects in your design.

### ***Access Using***

- Menu Path: *Setup – Constraints – Spacing*

### **Related Topics**

- [Working with Constraints](#)

## **cmgr\_xprobe**

An internal command.

## cns\_dummy\_net

The `cns_dummy_net` command displays the *Assign Dummy Nets by Pin to NetClasses* dialog box to assign dummy nets to Net Classes. You can assign dummy nets in either Spacing or Physical domain. You can also create a new Net Class and specify the constraint domain.

### Related Topics

- [Assigning NetClass to Dummy Pins](#)

## Assign Dummy Nets by Pin to NetClasses Dialog Box

### Access Using

- Menu Path: *Setup – Constraints – Dummy Net Assignment*

<i>Dummy Pin</i>	Lists the RefDes pin number of the pin on the dummy net.
<i>Spacing NetClass</i>	Choose to assign the available NetClass for Spacing domain to the dummy pin. To remove the dummy net from the NetClass assign a blank NetClass.
<i>Physical NetClass</i>	Choose to assign the available NetClass for Physical domain to the dummy pin. To remove the dummy net from the NetClass assign a blank NetClass.
<i>New NetClass</i>	Creates a new NetClass with the name specified.
<i>Spacing</i>	Choose to create new NetClass in the Spacing domain.
<i>Physical</i>	Choose to create new NetClass in the Physical domain.



## Assigning NetClass to Dummy Pins

To assign dummy pins to Net Class, do the following:


1. Choose *Setup – Constraints – Dummy Net Assignment*.  
Alternatively, type `cns_dummy_net` in the Command window.
2. On *Dummy Pin* row, choose an existing *Spacing NetClass* and *Physical NetClass* from the drop-down list.  
If the NetClass exists in both constraint domains it will appear in both columns.
3. To create a new NetClass, check the options *Physical* or *Spacing* or both to specify the constraint domain.
4. Enter new name in the *New NetClass* field.
5. Click *OK* to complete the assignments.

## Related Topics

- [cns\\_dummy\\_net](#)

## cns design

The `cns design` command displays the *Design Constraints* dialog box for you to set up DRCs that flag potential alignment and spacing problems for soldermask openings within a symbol or pin, pad soldermask to nearby pad or etch/conductor soldermask, and negative plane islands. The soldermask constraints are not area-dependent.

 This command is not available in the symbol editor mode.

### Related Topics

- [Creating Design Rules](#)
- [Setting Up Spacing Constraints for Soldermask Openings](#)

### Design Constraints Dialog Box

Use this dialog box to set online checks that flag potential alignment and spacing problems for design elements that are neither areas nor nets.

<i>On Off Batch</i>	Control when DRC is run for each constraint. For definitions of each of these options, see <i>Creating Design Rules in the user guide</i> .
<i>All</i>	Sets all of these constraints to the same DRC mode.
<i>Package to package</i>	Flags packages that overlap one another.
<i>Package to place keepin</i>	Flags packages that extend beyond place keepin.
<i>Package to place keepout</i>	Flags packages that extend inside a place keepout.
<i>Package to room</i>	Flags packages that are located in rooms to which they are not assigned—for example, package A is located in room B.
<i>Negative plane islands</i>	Flags electrically disconnected areas, or islands, formed by overlapping thermal reliefs and/or antipads on a negative layer shape. DRCs are reported between the shape and pins or vias forming the island.
<i>Oversize</i>	Scales up the pad geometry before the check for shape islands is done. Enter a value that takes into account inaccuracies that may be introduced in the manufacturing process. The default is 0.
<i>Soldermask alignment</i>	Specify the alignment tolerance (in design units) required for the proximity of Package Soldermask To Placebound or of Pad Soldermask To Pad Geometry.
<i>Tolerance</i>	Enter the constraint for soldermask-opening DRCs and applies globally to the board. The default for <i>Tolerance</i> is 0.
<i>Soldermask to soldermask</i>	Specify spacing checks between Pad Soldermask To Pad Soldermask Spacing, Symbol Soldermask To Symbol Soldermask Spacing, and between Symbol Soldermask To Pad Soldermask Spacing.
<i>Spacing</i>	Enter a spacing value to use between Pad Soldermask To Pad Soldermask Spacing, Symbol Soldermask To Symbol Soldermask Spacing, and between Symbol Soldermask To Pad Soldermask Spacing.
<i>Soldermask to shape</i>	Specifies soldermask checks for shapes. A DRC error is generated for a shape that comes within this specified clearance.
<i>Soldermask to pad and cline</i>	Specifies soldermask checks for pads and clines. A DRC error is generated for any pad or cline that comes within this specified clearance.
<i>Testpoint pad to component</i>	Specify spacing checks between the testpoint pad (edge) to components.

## C Commands

### C Commands--cns design

---

Spacing	Enter value to use for spacing between the testpoint pad (edge) to components.
Testpoint location to component	Specify checks for spacing between testpoint location (center) to components.
Spacing	Enter a value to use for spacing between testpoint location (center) to components.
Testpoint under component	Choose to flag test pads under components.

## Setting Up Spacing Constraints for Soldermask Openings

To set up spacing checks for design elements other than shape or nets, do the following:

1. Type `cns design` in the Command window.  
The *Design Constraints* dialog box opens.
2. If you want to set all of the constraints to the same DRC mode, choose the appropriate button in the *All* row. –or– If you want to set or change individual constraints, choose a mode in the constraint's row. For details, see [Design Constraints Dialog Box](#).
3. Choose *Apply* to save your changes and continue to work in this dialog box. –or– Choose *OK* to save your changes and close the dialog box.

When you save your changes, the layout editor runs a DRC check.

### Related Topics

- [cns design](#)

## cns electrical

The `cns electrical` command displays the *Electrical Constraints* dialog box for you to set up and maintain electrical constraint sets, including rules for differential pairs. This dialog box manages simple electrical constraints. For more complex ones, use Constraint Manager, which you open using the `cmgr` command.

### Related Topics

- [Creating Design Rules](#)
- [cmgr](#)
- [Tasks to Manage Electrical Constraints](#)

## Assigning Electrical Constraint Sets to Nets, Differential Pairs, and Buses

To assign an electrical constraint set to a net, differential pair, or bus, you must attach an ELECTRICAL\_CONSTRAINT\_SET property to the object with the type name you want. You can do this in the *Assign* tab.

⚠ You can also attach electrical constraint sets to nets using the ([property edit](#) or [cmgr](#) commands).

1. In the Electrical Constraints dialog box, choose the *Assign* tab. For details, see [Assign Tab](#).
2. In the *Electrical Csets* list, choose the constraint set you want to work on. –or– Create a new constraint set. For details, see [Creating an Electrical Constraint Set](#).
3. If necessary, disable the *Filter voltage nets* option at the bottom of the tab.
4. Choose from the *Available objects* list the objects to which you want to assign the constraint set.

⚠ To assign an electrical constraint set to a differential pair, you should choose the differential pair object. If you decide to assign the constraint set to the individual nets or Xnets that are part of the differential pair, be sure to assign it to both nets. Otherwise, you could get DRC errors due to the different values on each of the nets/Xnets.

To locate an object in the *Available objects* list, you can do any of the following:

- Scroll through the list.
- Type a name in the field above the list and choose on the list box. The name is highlighted.
- Narrow the objects displayed by using wildcard characters (\* and ?) when you type a name in the field above the list. When you choose on the list box, only the chosen objects appear.

To move an object into the *Assigned objects* list, do one of the following:

- Enable *Auto move*. When you choose a name in the *Available objects* list, it automatically moves into the *Assigned objects* list. (And when you choose a name in *Assigned objects*, it automatically moves back to *Available objects*.)
- If *Auto move* is not enabled, use the arrow keys to move highlighted names from one list to the other.

When you choose a name in the *Available objects* list, the object's current constraint set assignment, if any, appears below the list boxes.

1. Choose *Apply* to save your changes and continue to work in this dialog box. –or– Choose *OK* to save your changes and close the dialog box.

When you save your changes, the layout editor runs a DRC check. Also, the Constraint Apply Results (topology.log) window appears, listing details about the objects you assigned the electrical constraint set to or removed the constraint set from.

## Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## Calculating Differential Impedance

1. In the Electrical Constraints dialog box, choose the *DiffPair Values* tab.
2. Choose *Calculator* at the top of the tab.

The Differential Calculator Dialog Box appears. For details, see [Differential Calculator Dialog Box](#).

1. In the *Layer name* field, choose a layer.
2. Choose the button to the right of the field you want to recalculate. For example, you might choose to recalculate *Primary gap*.
3. Change other values in this dialog box and press Enter or Tab.

The target value is recalculated. In the example, you might change both the *Differential impedance* and *Line width* values. *Primary gap* would then be recalculated.

⚠ If you change the value of the field you chosen for recalculation by accident, the Multiple Choice Selection dialog box appears. Choose the field you want to recalculate and choose *OK*.

1. When you are done, choose *OK*.

### **Related Topics**

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## Creating an Electrical Constraint Set

You can create an electrical constraint set in any of the tabs of the Electrical Constraints dialog box except Modes. There are two ways to do this: create a cset from scratch or copy one.

### ***Creating a Constraint Set with no Initial Values***

1. In the *Electrical Csets* section, make sure that *Copy chosen cset to new cset* is not chosen.
2. Choose *New*.
3. Enter a new constraint set name in the dialog box that appears and choose *OK*.

### ***Copying a Constraint Set***

1. In the *Electrical Csets* section, choose a constraint set from the list.
2. Enable *Copy chosen cset to new cset*
3. Choose *New*.
4. Enter a new constraint set name in the dialog box that appears and choose *OK*.

## Deleting an Electrical Constraint Set

1. In the Electrical Constraints dialog box, choose the *Assign* tab.
2. In the *Electrical Csets* list, choose the name of the constraint set you want to delete.
3. Choose *Delete*.
4. Choose *Apply* to save your changes and continue to work in this dialog box. –or– Choose *OK* to save your changes and close the dialog box.

## Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)




## Defining Differential Pair Rules

1. In the Electrical Constraints dialog box, choose the *DiffPair Values* tab. For details, see [DiffPair Values Tab](#).
2. In the *Electrical Csets* list, choose the constraint set you want to work on. –or– Create a new constraint set. For details, see [Creating an Electrical Constraint Set](#).

The name of the constraint set you are working on appears below the *Electrical Csets* section, in the *Cset name* field, which is gray. If the constraint set is locked due to the importation of a techfile, the dialog box displays *Locked* in this area. You cannot edit such a constraint set's values.

1. Set values for the differential rules. The descriptions and syntax appear in the *Allegro Platform Properties Reference*.

To determine values for *Primary gap* and *Line width* based on a differential impedance, see [Calculating Differential Impedance](#) . Start at step 3.

 If a field in the *DiffPair Values* tab goes blank after you leave it, look at the error message in the status bar at the bottom of the dialog box for information.

1. Choose *Apply* to save your changes and continue to work in this dialog box. –or– Choose *OK* to save your changes and close the dialog box.

When you save your changes, the layout editor runs a DRC check.

## Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## Defining Net Values

1. In the Electrical Constraints dialog box, choose the *Net Values* tab. For details, see [Net Values Tab](#).
2. In the *Electrical Csets* list, choose the constraint set you want to work on. –or– Create a new constraint set. For details, see [Creating an Electrical Constraint Set](#).

The name of the constraint set you are working on appears below the *Electrical Csets* section, in the *Cset name* field, which is gray. If the constraint set is locked due to the importation of a techfile, the dialog box displays *Locked* in this area. You cannot edit such a constraint set's values.

1. Set values for the constraints. Descriptions of the rules and their syntax appear in the *Allegro Platform Properties Reference*.

 If a field goes blank after you leave it, look at the error message in the status bar at the bottom of the dialog box for information.

2. Choose *Apply* to save your changes and continue to work in this dialog box. –or– Choose *OK* to save your changes and close the dialog box.

## Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## Dialog Boxes to Manage Electrical Constraints

The following dialog boxes provide the tools to manage electrical constraints in your design:

- [Electrical Constraints Dialog Box](#)
- [Electrical Rule for Parallel Segments Dialog Box](#)
- [Select Layer Sets Dialog Box](#)
- [Differential Calculator Dialog Box](#)
- [Using the Electrical Constraints Dialog Box](#)

## Differential Calculator Dialog Box

In this dialog box, you perform what-if scenarios to determine what combinations of line width and primary gap values can help you obtain a particular differential impedance. These calculations occur only for edge-coupled differential pairs on a chosen ETCH/CONDUCTOR layer based on its properties:

- Material
- Thickness
- Electrical conductivity
- Dielectric constant
- Loss tangent
- Shield

The values of these properties appear in the Layout Cross Section spreadsheet.

When you close this dialog box, the values are not propagated to the *DiffPair Values* tab. Enter the values you want to use on the tab.

⚠ For broadside-coupling calculations, use the Layout Cross Section spreadsheet.

<i>Layer name</i>	Indicates the ETCH/CONDUCTOR layer for which you are running the calculation. By default, the TOP/SURFACE layer appears.
<i>Differential impedance</i>	Specifies the impedance of the differential pair, in ohms. Calculated for a pair of lines having the specified <i>Line width</i> and <i>Primary gap</i> on the layer.
Single-line impedance	Indicates the impedance of one line of etch/conductor on the chosen layer, in ohms. Changing this value automatically recalculates the <i>Line width</i> field. Each time the <i>Line width</i> field in this calculator changes—either when you directly modify it or when you choose it to be recalculated—this value is recalculated, too.
Line width	<p>Specifies the minimum width of each line of the differential pair, in design units. Changing this value automatically recalculates the <i>Single-line impedance</i> field. And changing the <i>Single-line impedance</i> field automatically recalculates this value. The layout editor assigns the default value from one of these fields in the following order:</p> <ol style="list-style-type: none"> <li>1. The <i>Line width</i> from the <i>DiffPair Values</i> tab for the current electrical constraint set, if specified.</li> <li>2. The <i>Min line width</i> from the DEFAULT physical constraint set for this layer.</li> </ol>
Primary gap	<p>Indicates the ideal edge-to-edge spacing between the pair that should be maintained for the entire length of the pair, in design units. The layout editor assigns the default value from one of these fields in the following order:</p> <ol style="list-style-type: none"> <li>1. The <i>Primary gap</i> from the <i>DiffPair Values</i> tab for the current electrical constraint set, if specified.</li> <li>2. The <i>Line To Line</i> value from the DEFAULT spacing rule set for this layer.</li> </ol> <div style="background-color: #fff9c4; border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> <p>⚠ Although the primary gap value maintains the spacing for a differential pair, it does not prevent any other conductive element from being placed within the primary gap as long as that element adheres to the proper design rules. You can prevent the data item from being placed within the primary gap with additional spacing constraints. For example, if you have a differential pair with a <i>Primary Gap</i> of 15 mils, a default constraint <i>Line to line</i> width of 5 mils, and a <i>Shape to Line</i> clearance of 5 mils, and you routed a cline between the differential pair traces, you would not get any DRC errors.</p> </div>
Buttons to the right of the fields	Indicate the value you want to recalculate. The default value is <i>Differential impedance</i> .

## Options Tab

When calculating delay, you use this tab to include the conducting portion of a via/pin, also known as Z Axis Delay, in any length-related constraint, or the pin delay from the internal package connection to the pin's mounting layer.

If you defined DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY in time units, the conducting portion of a via/pin value is multiplied by the Z Axis Delay *Propagation Velocity Factor*, which is a constant used to convert from time to length units.


You can use the PIN\_DELAY property to manage interchip delay or die-to-die timing paths across a printed circuit board and thereby factor inter-package delay

## C Commands

### C Commands--cns electrical

into timing requirements. Pin delay, as defined by associating the PIN\_DELAY property to component instance or definition pins, is the length of a through pin. When you use pin delay, it is multiplied by the *Pin Delay Propagation Velocity Factor*, which is a constant used to convert from time to length units for DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY.

⚠ *Pin-delay, Z Axis Delay, and Same Net Xtalk and Parallelism Checks*- related fields in the *Options* panel are unavailable in tiers other than Allegro X PCB Editor GXL and GXL. If you execute design rule checks on a board in Allegro X PCB Editor GXL and then load the board into a higher tier, DRCs caused by Z Axis Delay, pin delay, or same net crosstalk and parallelism appear. However, during DRC updates in lower tiers, PROPAGATION DELAY and RELATIVE PROPAGATION DELAY checks ignore via and pin delay in DRC calculations. Setting *Same Net Xtalk and Parallelism Checks* ignores the DRC calculations for crosstalk and parallelism that a net makes to itself.

<i>Drc Unrouted</i>	
<i>DRC Propagation Delay</i>	Choose to compute propagation delay based on the Manhattan distance of ratsnest connections.
<i>DRC Relative Propagation Delay</i>	Choose to compute relative propagation delay based on the Manhattan distance of ratsnest connections.
<i>Pin Delay</i>	
<i>Include in All Propagation Delays and in DiffPair Phase Checks</i>	<p>Choose to include the pin delay value defined by the PIN_DELAY property, which specifies the time delay/length from the internal package connection to the mounting layer of the pin, in DRC calculations for DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY. By default, pin delay is enabled in new designs and disabled in existing designs, in which case, these DRCs are verified only against etch/conductor.</p> <div style="background-color: #e8f5e9; padding: 5px; margin: 5px 0;">  You can import initial values for new boards using <a href="#">techfile in</a>.         </div> <p>If you defined DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY in time units, the pin delay is calculated by multiplying by the <i>Pin Delay Propagation Velocity Factor</i>.</p>
<i>Propagation Velocity Factor</i>	If you defined DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY in time units, to convert from time to length, enter a value appropriate for the physical structure and material the signal traverses. The default value is 1.524e+08 m/s. Pin delay is calculated by multiplying the delay in units of time by the conversion factor.
<i>Z Axis Delay</i>	
<i>Include in All Propagation Delays and in DiffPair Phase Checks</i>	Choose to add the length of vias in these DRC calculations for DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY. Z Axis Delay is excluded from total etch/conductor length calculations. This field is disabled by default.
<i>Z Axis Delay Propagation Velocity Factor</i>	If you defined DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY in time units, to convert from time to length, enter a value appropriate for the physical structure and material the signal traverses. The default value is 1.524e+08 m/s. The conducting portion of a via/pin is calculated by multiplying the delay in units of time by the conversion factor.
<i>Same Net Xtalk and Parallelism Checks</i>	
<i>Perform Xtalk and Parallelism checks within the same net</i>	Instructs nets in the design to perform DRC calculations for crosstalk and parallelism on themselves. (Turned off, crosstalk and parallelism checks are made only between one net to every other.) This option is automatically enabled if the variable has previously been set, otherwise the default is unchecked. <b>Note:</b> This feature can impact performance and convergence issues in the Allegro PCB Router if enabled during initial routing stages, therefore we recommend that it not be used during initial or early routing. The feature can be controlled in the router through the design or do file syntax.
<i>OK</i>	Updates the database with your changes, runs DRC, then closes the dialog box.

### **Related Topics**

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## Electrical Constraints Dialog Box

The *Electrical Constraints* dialog box lets you manage simple electrical constraints that may not require the complex functionality available with Constraint Manager. Use the Electrical Constraints dialog box to:

- Set DRC modes for electrical constraints
- Add, delete, and modify electrical constraint sets and their values, including differential pair rules
- Assign electrical constraint sets to nets, differential pairs, and buses


### Electrical Csets Section

You can use various buttons in this section, depending on the tab that is displayed. Nothing in this section is available for the *Modes* tab.

<i>Electrical Csets</i>	Lists the names of the electrical constraint sets in the design database. When you highlight a name here, the values or objects assigned to the constraint set appear.
<i>New</i>	Creates a new electrical constraint set with the name you enter in the small dialog box that appears.
<i>Delete</i>	Removes the chosen constraint set and its assignments to any objects in the design. This button is only available for the <i>Assign</i> tab.
<i>Copy chosen cset to new cset</i>	Specifies that the chosen constraint set be used as the basis of a new one. Use this in conjunction with the <i>New</i> button.
<i>Cset name</i>	Displays the chosen constraint set. If you import the constraint set from Constraint Manager, the phrase <i>Topology based</i> appears next to the name. If the constraint set is locked as part of a techfile import, the word <i>Locked</i> appears next to the name. In that case, you cannot edit the values of the constraint set.

### Modes Tab

In this tab, you set DRC modes for supported electrical constraints, which are described in the *Allegro Platform Properties Reference*.

 *All Differential pair checks* is not a property. 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.

For details of DRC modes, see *Creating Design Rules* in the user guide. You set these DRC modes at a global level—that is, they control all electrical constraint sets. For instance, you cannot run [DRC for parallelism on specific nets], except in Constraint Manager. Consequently, you cannot choose any fields or buttons in the *Electrical Csets* section of the dialog box.


The settings for DRC modes in designs that you uprev to Release 14.0 are determined by a "worst-case" scenario. For example, if one electrical constraint set is set with *Max parallel* on, parallelism is set to *On* at the global level.

The order of precedence is *On*, *Batch*, *Off*.

When you create a new design, the default settings for all electrical DRCs is *Off*.

### Net Values Tab

In this tab, you can create and copy electrical constraint sets and edit the values for the constraints. These sets contain relatively basic electrical constraints for nets. For more complex electrical constraints, you must run the [cmgr](#) command to use Constraint Manager.

 Constraint descriptions and syntax appear in the *Allegro Platform Properties Reference*.

... (button for <i>Max parallel</i> )	Displays the Electrical Rule for Parallel Segments dialog box which lets you define up to four length and distance (separation) maximum parallel constraints in individual fields, rather than as a single string in the <i>Max parallel</i> field. For details, see <a href="#">Electrical Rule for Parallel Segments Dialog Box</a> in the <i>Allegro PCB and Packaging Physical Layout Command Reference</i> .
... (button for <i>LayerSets</i> )	Displays the Select Layer Sets dialog box which lists the current available layer sets to add to the ECset. Choose <i>Define LayerSets</i> to display the Layer Sets dialog box that you use to define more layer sets. For more information, see <a href="#">Select Layer Sets Dialog Box</a> in the <i>Allegro PCB and Packaging Physical Layout Command Reference</i> .

### DiffPair Values Tab

In this tab, you can create and copy electrical constraint sets and edit their differential pair rules.

Differential rule descriptions and syntax appear in the Allegro Platform Properties Reference.

<i>Impedance to line width Calculator</i>	Displays the Differential Calculator dialog box. For details, see <a href="#">Differential Calculator Dialog Box</a> . Use the calculator to determine what combinations of line width and primary gap values can help you obtain a particular differential impedance for edge-coupled differential pairs.
---	--

 If you are defining differential pairs by layer, do not fill in the Primary Gap or Neck Gap. For information on defining differential pairs by layer, see *Creating Design Rules* in the user guide.

### Assign Tab

This tab lets you create and copy electrical constraint sets; attach them to nets, differential pairs, and buses in your design; and delete these csets. (You can also attach nets to electrical constraint sets using the [property edit](#) and [cmgr](#) commands.)

<i>Auto move</i>	Automatically moves a net, differential pair, or bus from one list to another when you highlight a name. By default, this option is disabled.
<i>Available objects</i>	Lists the nets, differential pairs, and buses to which you can assign an electrical constraint set. You can use wildcard characters (* and ?) to refine the list. If <i>Auto move</i> is disabled, highlight a name and choose on an arrow key to move it to the <i>Assigned objects</i> list.
<i>Assigned objects</i>	Lists all the nets, differential pairs, and buses associated with the currently chosen electrical constraint set. If <i>Auto move</i> is disabled, highlight a name and choose on an arrow key to move it to the <i>Assigned objects</i> list.
<i>Current object assignment</i>	Displays the electrical constraint set assigned to the chosen object in the <i>Available objects</i> list.
<i>Filter voltage nets</i>	Removes nets from the <i>Available objects</i> list that have the VOLTAGE property. By default, this option is enabled.
<i>OK</i>	Updates the database with your changes, runs a DRC check, then closes the dialog box.
<i>Apply</i>	Updates the database with your changes and runs a DRC check.

### Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)



## Electrical Rule for Parallel Segments Dialog Box

In this dialog box, you can define up to four length and distance (separation) maximum parallel constraints. The values are in design units.

<i>Length</i>	Maximum parallel length between different net segments.
<i>Distance</i>	Edge-to-edge distance between different net segments.

For details about how the layout editor applies these constraints, see the description of the MAX\_PARALLEL property in the *Allegro Platform Properties Reference*.

### Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## Select Layer Sets Dialog Box

In this dialog box, you choose layer sets to include in an ECset.

<i>Available Layer Sets</i>	Lists the names of the layer sets in the design database.
<i>Auto Move</i>	Automatically moves chosen layers to and from the <i>Assigned Layers</i> list box. Enabled by default.
<i>Assigned Layer Sets</i>	Displays the assigned layer sets after you move them from the <i>Available Layer Sets</i> list box. The layout editor displays an error message at the bottom of the dialog box if you assign layer sets with overlapping layers.
<i>Define LayerSets</i>	Displays the Layer Sets dialog box that lets you create new layer sets and modify existing ones.
<i>OK</i>	Updates the <i>LayerSets</i> field of the <i>Net Values</i> tab in the Electrical Constraints dialog box with the assigned layer sets, and then closes the dialog box.
<i>Cancel</i>	Closes the dialog box without saving any changes.

## Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## Setting DRC Modes for the Electrical Constraint Set

1. In the Electrical Constraints dialog box, choose the *Modes* tab. For details, see [Modes Tab](#).
2. If you want to set all of the constraints to the same DRC mode, choose the appropriate button in the *All* row. –or– If you want to set or change individual constraints, choose a mode in the constraint's row.
3. Choose *Apply* to save your changes and continue to work in this dialog box. –or– Choose *OK* to save your changes and close the dialog box.

When you save your changes, the layout editor runs a DRC check.

## Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## Tasks to Manage Electrical Constraints

The following tasks help manage the electrical constraints in your design:

- [Assigning Electrical Constraint Sets to Nets, Differential Pairs, and Buses](#)
- [Defining Differential Pair Rules](#)
- [Defining Net Values](#)
- [Setting DRC Modes for the Electrical Constraint Set](#)
- [Creating an Electrical Constraint Set](#)
- [Calculating Differential Impedance](#)
- [Using Pin Delay](#)
- [Using Z Axis Delay](#)

## Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)

## Using Pin Delay


1. Define pin delay using one of the following methods:
  - Define pin delay as library properties that you can write to Allegro Design Entry HDL L. Use Allegro PCB System Librarian L Part Developer to manually assign the PIN\_DELAY property and values to symbol pins or automatically import the PIN\_DELAY values through its *Import Wizard*.
  - Derive pin delay values from an external source or from Allegro X Advanced Package DesignerL. Using *File – Export – Board Level Component* ([allegro\\_component](#) command) in APD, you can generate the Package Pin Delay report, that contains the package pin number, delay value of the net assigned to the package pin, and net name.
  - Export pin delay values from one design using *File – Export – Pin Delay* ([pin\\_delay out](#) command) and then import them to another design and assign them to component instance pins using *File – Import – Pin Delay* ([pin\\_delay in](#) command).
  - Assign the PIN\_DELAY property and values to the component instance pins using *Edit – Properties* ([property edit](#) command).
  - Edit values in Constraint Manager. See *Analyze – Analysis Modes* in the Allegro Constraint Manager Reference for details.
2. In the *Electrical Constraints* dialog box, choose the *Options* panel.
3. Enable the *Pin Delay* Include in All Propagation Delays and in DiffPair Phase Check field.
4. Enter a value in the Propagation Velocity Factor field for use in converting between time and length in DRC calculations for DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY.
5. To view information about the pin-delay values:
  - Launch Constraint Manager or
  - Use *Display – Element* ([show element](#) command)
  - Choose *Show* from the *Edit Properties* dialog box

## Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## Using the Electrical Constraints Dialog Box

1. Run the `cns electrical` command.
2. In the Electrical Constraints dialog box, perform these procedures to specify electrical design rules:
  - a. [Assigning Electrical Constraint Sets to Nets, Differential Pairs, and Buses](#) (use the *Assign* tab)
  - b. [Defining Differential Pair Rules](#), if working with differential pairs (use the *DiffPair Values* tab)
  - c. [Defining Net Values](#) (use the *Net Values* tab)
  - d. [Setting DRC Modes for the Electrical Constraint Set](#) (use the *Modes* tab)

 To delete an electrical constraint set, you use the *Assign* tab. See [Deleting an Electrical Constraint Set](#) for instructions.

## Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## Using Z Axis Delay


1. In the *Electrical Constraints* dialog box, choose the *Options* panel.
2. Enable the *Z Axis Delay* Include in All Propagation Delays and the DiffPair Phase Check field.
3. Enter a value in the Propagation Velocity Factor field for converting between time and length in DRC calculations for DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY.
4. To view information about the via-delay values:
  - Use *Display – Element* ([show element](#) command)
  - Choose *Show* from the *Edit Properties* dialog box

## Related Topics

- [cns electrical](#)
- [Dialog Boxes to Manage Electrical Constraints](#)
- [Tasks to Manage Electrical Constraints](#)

## cns onlinedfmdrc

The `cns onlinedfmdrc` command lets you enable or disable on-line DFM checks through a single option without managing these checks in Constraint Manager or Analysis Modes dialog box. During the design editing process when you enable this command it controls all the *Manufacturing* category checks, such as DFF, DFA, and DFT. The DRC status of the design is marked out of date when on-line DFM checks are disabled.

 You can turn on DFM checks only when on-line DRC is enabled.

### ***Access Using***

- Menu Path: *Setup – Enable On-Line DFM checks*



## Enabling Online DFM Checks

To enable on-line DFM checks, do one of the following:

- Choose menu option *Setup – Enable On-Line DFM checks*.
- In the status bar of the layout editor, click DRC status and select *Enable On-Line DFM checks*.
- Open the Status dialog box and enable *On-Line DFM checks* check box.
- In the Analysis Modes dialog box, enable *On-Line DFM checks* check box.

## **cns onlinedrc**

The `cns onlinedrc` command enables you to turn on-line DRC on or off. On-line DRC provides you with immediate feedback when you violate a design rule. The on-line DRC function operates according to the settings you define in the *Analysis Modes* dialog box.

### ***Access Using***

- Menu Path: *Setup – Enable On-Line DRC*

## cns show

The `cns_show` command allows you to generate a report that provides details about constraints that apply to an object or pair of objects you select. The report appears in a separate window which offers print, search, and save-to-disk functions. The report includes:

- Net owner
- Net Class membership
- Net Class–Net Class membership
- any overrides
- applicable constraint areas

You are prompted to select a single object or a pair of objects. Legal objects are: nets, pins, vias, clines, etch/conductor shapes, and ratsnests.

For a single object, physical constraints are reported. For a pair of objects, spacing constraints are reported. For nets, all applicable areas are reported. For non-nets (physical objects), the applicable area is determined based on the pick location. All constraints are resolved and reported for physical objects based on their pick location.

### Access Using

- Menu Path: *Display – Constraint*
- Toolbar Icon:



## Generating a Constraint Report


To view the constraint applied to objects, do the following:

1. Choose *Display – Constraint*.  
Alternatively, type `cns_show` in the Command window.
2. Choose on a single object to select it. –or– Drag a window around a pair of objects to select them.  
The Show Constraints window appears with the constraint information for the object or pair of objects you chosen.
3. In the *Constraint Hierarchy* table, choose the blue colored text in a *Location* cell to jump to the object in the design window of the layout editor.
4. In the *Constraint Hierarchy* table, choose the blue colored text in any of the object cells to open the corresponding worksheets in Constraint Manager.
5. In the *Resolved Spacing Constraints* table, choose the blue colored text in any of the cells under the *Source Name* column to open the corresponding worksheets in Constraint Manager.
6. Use the Save, Print, or Search functions in the Show Constraints window to work with the report information.

## color

The `color` command is a console command that controls the following aspects of design display:

- Colors for:
  - Subclasses
  - Grids
  - Ratsnest lines
  - Design window background
  - Highlighted objects during interactive commands
- Visibility of classes and subclasses\*

 You can set and change *grids*, *ratsnest lines*, *design window background*, and *highlighted objects during interactive command* using command arguments or using the Color and Visibility dialog box.

The command, without any parameters, opens the Color and Visibility window.

Because you can set the colors and visibility for each subclass of your design with this command, you can also use the color command to view all the classes and subclasses, and the groups under which they are located.

Lastly, the tool provides a global color palette, which you can modify and save as a local or database (design-specific) palette with the color command.

For more details on customizing colors used in your design, see *Getting Started* in the user guide.

### Syntax:

```
color [-gridcolor | -ratscolor | -highlightcolor | -permcOLOR "<color_number>"]  
  
[-background "<RGB_value>"] [-group "<group_name>"] [-globvis "{off | on}"]  
  
[-vis "{<class_name> | <class/subclass_name>}"] [-invis "{<class_name> |  
<class/subclass_name>}"] [-toggle "<class_name> | <class/subclass_name>}"]  
  
[-help]
```

<b>-gridcolor "</b> <b>&lt;color_number&gt;"</b>	Sets the color of the grids in the design window. Choose the color from the current palette. This argument is the same as the Grids field in the Display section of the Color and Visibility dialog box. (Group is set to Display.)
<b>-ratscolor "</b> <b>&lt;color_number&gt;"</b>	Sets the color of the ratsnest lines. Choose the color from the current palette. This argument is the same as the Ratsnest field in the Display section of the Color and Visibility dialog box. (Group is set to Display.)
<b>-highlightcolor "</b> <b>&lt;color_number&gt;"</b>	Sets the color of objects that are temporarily highlighted during an interactive command, such as copy or move. Choose the color from the current palette. This argument is the same as the Temporary highlight field in the Display section of the Color and Visibility dialog box. (Group is set to Display.)
<b>-permcOLOR "</b> <b>&lt;color_number&gt;"</b>	Sets the permanent highlight color, the color of objects when you run the <code>highlight</code> command. Choose the color from the current palette. This argument is the same as the Permanent highlight field in the Display section of the Color and Visibility dialog box. (Group is set to Display.)

<b>-background "&lt;RGB_value&gt;"</b>	<p>Sets the color of the design window's background. The RGB value is the 8-bit representation of a color value, ranging from 0 to 255. For example:</p> <ul style="list-style-type: none"> <li>• 0xff0000 is red.</li> <li>• 0xffffffff is white.</li> <li>• 0x000000 is black.</li> </ul> <p>This argument is the same as the Background field in the Display section of the Color and Visibility dialog box. (Group is set to Display.)</p>
<b>-group "&lt;group_name&gt;"</b>	Opens the Color and Visibility dialog box, with the Group field set to the specified group name.
<b>-globvis "{off   on}"</b>	Controls visibility for all subclasses. This argument is the same as the Global visibility field in the Color and Visibility dialog box.
<b>-vis "{&lt;class_name&gt;   &lt;class/subclass_name&gt;}"</b>	Enables visibility for an entire class or subclass. This argument is the same as enabling the check box for a class or subclass in the Color and Visibility dialog box.
<b>-invis "{&lt;class_name&gt;   &lt;class/subclass_name&gt;}"</b>	Makes the specified class or subclass invisible. This argument is the same as disabling the check box for a class or subclass in the Color and Visibility dialog box.
<b>-toggle "{&lt;class_name&gt;   &lt;class/subclass_name&gt;}"</b>	Reverses the current visibility for the specified subclass. If the subclass is visible, this command makes it invisible. If the subclass is invisible, this command makes it visible. This argument is the same as clicking the check box for a class or subclass in the Color and Visibility dialog box.
<b>-help</b>	Displays command syntax in the console.

### Examples:

This example changes the color of ratsnest lines:

```
color -ratscolor "20"
```

The next example sets global visibility to off. Nothing appears in the design window.

```
color -globvis "off"
```

The next example sets the background color of the design to turquoise:

```
color -background 0x00ffff
```

This example sets the `ASSEMBLY_NOTES` subclass of the `BOARD GEOMETRY` class visible:

```
color -vis "BOARD GEOMETRY/ASSEMBLY_NOTES"
```

The final example changes the visibility of the `OUTLINE` subclass of the `BOARD GEOMETRY` class. If visibility for this subclass is enabled, this command disables it—and vice versa.

```
color -toggle "OUTLINE"
```

## Color and Visibility Dialog Box

Use this dialog box to set visibility and color of subclasses and their elements.

Group	Specifies a group. The subclasses in the selected group are listed.
Global visibility	Controls visibility for all subclasses of the specified group.
<i>Palette</i>	Lists the colors that can be applied.
<i>Modify</i>	Opens the Color dialog box where you can specify a custom color. This is same as the <a href="#">Select Color dialog box</a> .
Ok	Closes the dialog box and saves your modifications.
<i>Cancel</i>	Closes the dialog box and does not save your modifications.
Apply	Saves modifications without closing the dialog box.
Reset	Changes settings to the default.
Help	Opens online help.

## Related Topics

- [Select Color Dialog Box](#)

## color192

The `color192` command launches the Color dialog box, which supports 192 colors and comprises the Layers and Nets grids and tabs to customize display (*Display*), favorites (*Favorites*) and visibility (*Visibility Panel*).

- **Layers**  
Controls the color and visibility settings of classes and subclasses, along with levels of transparency for the design and shapes. Use the Layers/Display grid to also control shadow dimming, highlighting, ratsnest display, waived DRCs, and drill holes. Use this tab to create your own special colors or palettes, save to external `.col` files, and then apply to other designs. You can also assign stipple patterns to the layers. Assigning a pattern to a color is applied to all corresponding objects on that layer.
- **Nets**  
Customize color settings on nets or across their elements including pins, vias, clines, shapes, or rats. Apply colors at the bus, differential pair, xnet, and net level. Colors applied to hierarchical objects descend to their membership. Filtering and sorting controls are available to customize the display of nets. Custom color settings can be temporarily disabled, which reverts the color display back to layer- based settings while preserving the net coloring scheme for future use. Ratsnest lines can be hidden or shown on an individual net or a group of nets in a design using right-click options.
- **Display**  

Control the design window's appearance. The Display group has no associated classes or subclasses.
- **Favorites**  

Centralizes a user-defined group of frequently accessed subclasses. To add a subclass to the favorites list, right-click the color box associated with that subclass in the *Layers* tab and choose *Add to My Favorites*. Similarly, to remove a subclass from the favorites list; right-click the color box in Favorites tab and choose *Remove from favorites*. These options are also available on the ALL horizontal/vertical headers in the *Layers* tab form to add subclasses to the favorites tab.
- **Visibility panel**  

Use to customize the Visibility panel itself.

Classes added to the Visible Classes section become vertical columns on the Visibility panel. Use the options in the lower section to show or hide elements from the Visibility panel. Control the color boxes on the Visibility panel using the sliders.

## Related Topics

- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)

## Adding Subclasses to the My Favorites folder

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color dialog box appears.

1. In the Layers tab, right-click the color box associated with the subclass you want to add to *Favorites*.
2. Choose *Add favorites* from the pop-up menu.
3. Add as many subclasses as necessary.

 A subclass listed in *Favorites* folder can be removed by right-clicking and choosing *Remove from favorites* from the pop-up menu.

## Related Topics

- [color192](#)
- [Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)



## Assigning Colors or Patterns to Subclasses

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color Dialog box appears.

1. Make the *Layers* tab active.
2. Choose a folder from the left pane that contains the subclass whose color you want to change.
3. Choose a color from the *Available colors* section or a pattern from the *Available patterns* list.  
The Selected boxes show the color or pattern or a combination of both, depending on your choice.
4. Choose the color box next to the subclass whose color or pattern you want to change.
  - To set the color or pattern to an entire row or column, choose the color box next to the *All* column or *All* row.
  - To set the color or pattern globally, choose the color box next to the intersection of the *All* row and *All* column cell (*All/All* cell)
  - To set color or pattern for a specific object, choose the box for that object in the layers row

The color box changes to the color and pattern you chose.

For example, if you choose Stack-Up, to change the color of conductor on the Top layer, choose the color box for *TopCond*.

5. Click *Apply* to update the drawing and continue changing colors.
6. Click *OK* to save changes and close the dialog box.

The *Options* panel displays the color assigned to a subclass in a color box next to the subclass name.

## Related Topics

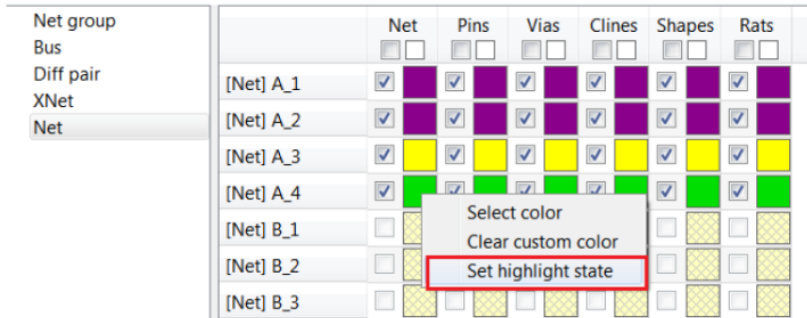
- [color192](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)

## Assigning Custom Color and Highlighting from Nets and Net Elements

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color dialog box appears.

1. In the *Nets* tab, right-click a box with no  $\checkmark$  in it to the left of the element's color box and then choose *Set highlight state*.



The element becomes highlighted in the design canvas, and its name displays in boldface in the Nets grid.

### Related Topics

- [color192](#)
- [Color Dialog Box](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)

## Assigning Stipple Patterns to Fixed Objects

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color Dialog box appears.

1. Make the *Display* tab active.
2. Click the *Pattern* box in the *Fixed objects* section and select a pattern.
3. Click *Apply* to update the drawing and continue changing patterns.
4. Click *OK* to save changes and close the dialog box.

## Changing grids, ratsnest lines, and highlighting colors

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color Dialog box appears.

1. In the *Display* tab, choose a color from *Available colors*.
2. In the *Display* section, choose the color box next to the item (*Grids*, *Temporary highlight*, *Waived DRC*, and so on) whose color you want to change.

The color box for this item changes to the color you chose from the *Color* section.

1. Click *Apply* to update the drawing and continue changing colors.
2. Click *OK* to save changes and close the dialog box.

## Related Topics

- [color192](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)

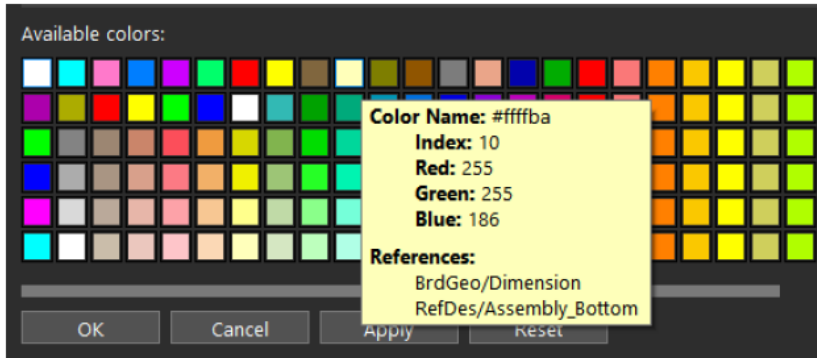

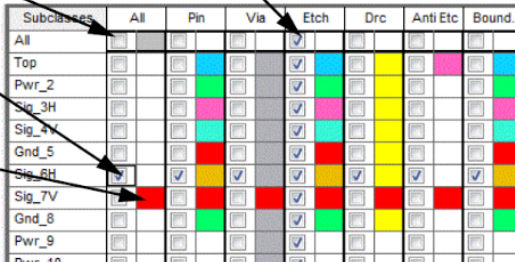
## Color Dialog Box

### Access Using

- Menu Path: *Display – Color/Visibility*
- Toolbar Icon:





OK	Closes the dialog box after saving changes.
Cancel	Closes the dialog box without saving any changes. All palette changes are retained in the database, if saved.
Apply	Saves changes.
Controls Common to All Tabs Except Visibility Panel	
Load	
Load default color palette	Loads the <code>\$HOME/pcbenv/allegro_192.col</code> and if not found, loads the Cadence defaults. Displays the default color palette, an array of 16 x 6 colors. The first column comprises popular colors typically used in designs.
Load color palette	Imports your customized color palettes from an external <code>.col</code> file and applies them to the current design. A file browser appears with the filter set to <code>*.col</code> and a list of all <code>.col</code> files available in the current local working directory. You can manually browse to other directories to open a color file.
Load color view	Opens Colorview Load to browse the color visibility view file ( <code>*.color</code> ) for loading into the design.
Load color parameters	Opens Imports Parameter File dialog to choose a database parameter record file <code>.prm</code> containing customized parameters into the design.
Save	
Save default color palette	Exports the current design's customized color palettes to the external <code>\$HOME/pcbenv/allegro_192.col</code> file.
Save color palette	Exports the current design's customized color palettes to an external <code>.col</code> file stored in your local working directory. A file browser appears with the filter set to <code>*.col</code> and a list of all <code>.col</code> files in the current local working directory. You can manually browse to other directories to specify an alternate save location.
Save color view	Opens Color View dialog to export the current layer visibility settings to a color visibility view file <code>.color</code> in your local working directory.
Save color parameters	Opens Export Allegro Parameters form to choose parameters of a design to exports to a <code>.prm</code> file in your working directory.

Available colors	<p>Select a color from a table of all the possible color swatches. The tooltip displays color information; color table index, RGB value, and color name string. The layers to which the selected color are assigned is also shown as references.</p> 																																																																																																
Highlight unused colors	Select to highlight colors that have not been used. Not selected by default.																																																																																																
Selected	Displays the selected color and pattern and allows customization of color.																																																																																																
	<p>Three boxes are displayed. The first box from the left displays the selected color and the third box from the left displays the selected pattern. The middle box displays the look on combining the selected color and pattern. To display only the pattern, remove selection from the first check box from the left. Similarly, to display only the color, remove selection from the second check box. Click the first box showing the color to open the Set Color dialog box and add a custom color.</p>																																																																																																
Available patterns																																																																																																	
Layers tab	<p>Displays each class associated with a group. The color and visibility of the subclasses associated with that class display horizontally. Each row lists a subclass. An X indicates the subclass is visible. The color box indicates the color assigned to the subclass element. Clicking the All column or All row enables visibility for the entire row or column. Clicking the intersection of the All row and All column cell (All/All cell) enables visibility globally. By default, subclasses are visible.</p> <div><p> For ICP products, the Color tab has a <i>Bond Wires Profiles</i> category. Choose this category to set the color and visibility of bond wires based on the profiles. Bond wires do not reside on any subclass. From this location, you can control the color for the bond wires or set the display, based on the profiles.</p></div> <div><p>Choose this box to enable visibility globally</p><p>Choose this box to enable visibility of entire column</p><p>Choose this box to enable visibility of entire row</p><p>Choose this color box to apply color across entire row</p><table><tr><th>Subclasses</th><th>All</th><th>Pin</th><th>Via</th><th>Etch</th><th>Drc</th><th>Anti Etc</th><th>Bound.</th></tr><tr><td>All</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr><tr><td>Top</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr><tr><td>Pwr_2</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr><tr><td>Sig_3H</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr><tr><td>Sig_4V</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr><tr><td>Gnd_5</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr><tr><td>Sig_6H</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr><tr><td>Sig_7V</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr><tr><td>Gnd_8</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr><tr><td>Pwr_9</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr><tr><td>Pwr_10</td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td><td><input checked="" type="checkbox"/></td></tr></table></div>	Subclasses	All	Pin	Via	Etch	Drc	Anti Etc	Bound.	All	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Top	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Pwr_2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Sig_3H	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Sig_4V	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Gnd_5	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Sig_6H	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Sig_7V	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Gnd_8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Pwr_9	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Pwr_10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Subclasses	All	Pin	Via	Etch	Drc	Anti Etc	Bound.																																																																																										
All	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										
Top	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										
Pwr_2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										
Sig_3H	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										
Sig_4V	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										
Gnd_5	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										
Sig_6H	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										
Sig_7V	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										
Gnd_8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										
Pwr_9	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										
Pwr_10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>																																																																																										

## C Commands

### C Commands--color192

Hide unused layers	Hide layers that are not used in the active design.
Filter layers	Searches every class and subclass for specific or groups of subclass layers.
Global Visibility	Control whether or not all classes and subclasses are visible. The options are <i>On</i> , <i>Off</i> and <i>Last</i> .
Nets tab	
	Alphabetically lists nets hierarchically by bus, differential pair, Xnet, or net. An $\checkmark$ indicates the custom color visibility state is enabled and visible in the design canvas. The color box indicates the custom color assigned to the net or net element.
Hide Custom Colors	Choose to display all elements in their Class/Subclass color and disable the display of any highlighting or custom colors throughout the design.
Sort ascending	Lists the nets in ascending order.
Sort descending	Sort descending Lists the nets in descending order.
Exclude default nets	Exclude default nets Excludes default nets from the list.
Filter nets	
Clear All Nets	Removes the custom color and state from all nets in the database, as individual color and state boxes are applicable only to nets visible in the <i>Nets</i> grid.
Display tab	
Display	
Temporary highlight	Specifies the color of elements that are temporarily highlighted when you run the <code>highlight</code> or <code>assign color</code> commands or during an interactive command, such as <code>copy</code> or <code>move</code> . The default is white.
Grids	Specifies the color of the grids. The default is white.
Rats top-top	Specifies the color of ratsnest lines that connect top-side only components (start-end pin on top).
Rats top-bottom	Specifies the color of ratsnest lines (one pin on top, other on bottom).
Rats bottom-bottom	Specifies the color of ratsnest lines that connect bottom-side only components (start-end pin on bottom). The default is aqua.
Waived DRC	Specifies the color of waived DRC error markers. The default is yellow.
Drill holes	Specifies the color of drill holes. The default is grey.
Backdrill holes	Specifies the color of backdrill holes. The default is green.
<i>Drill labels</i>	<p>Specifies the color of via span labels optionally displayed on the pads of blind and buried via(s). The default is white. The labels indicate the via hole extents. Pins, through hole vias, and single-layer vias remain unlabeled in the canvas. The label numbers map to design subclasses in order from top to bottom. Custom subclass names are ignored.</p> <div>  The visibility of via span labels is controlled using the <i>Drill Labels</i> parameter in the <i>Display</i> tab of the Design Parameter Editor. For details on the via label nomenclature, see the hover-over description for the <i>Drill Labels</i> parameter in the Design Parameter Editor dialog box. </div>
<i>Stacked drill label</i>	Specifies the color of stacked span of via labels. The default is white.
Background	Specifies the design window's background color. The default is black.
Alignment Guides	Specifies the color for alignment guides. The default is pink.
<i>Fixed objects</i>	

<i>Pattern</i>	Assigns stipple patterns to fixed objects.
Shadow mode	Highlights an individual element without affecting the visibility of that element's entire subclass. Refer to the <a href="#">shadow</a> command for more information on these functions.
<i>Enabled</i>	Activates and deactivates <i>Shadow mode</i> , which darkens the colors of objects and elements of your design. Use this with the <a href="#">highlight</a> command.
<i>Dim active layer</i>	Applies brightness to the colors of objects in the active layer, darkening the colors so that the highlighted objects are more prominent.
Brightness Control	Specifies the percentage of brightness applied to colors when Shadow mode is set to On.
Global Transparency	Assigns varying degrees of transparency to all elements in the entire design. Sliding the bar completely to the right represents a pre-16.0 graphics display. Sliding the bar completely to the left causes previously filled geometry, such as clines and pads, to display with less intensity.
Shapes Transparency	Assigns varying degrees of transparency to shapes only. Sliding the bar completely to the right represents a pre-16.0 graphics display. Sliding the bar completely to the left causes shapes to display with less intensity.
Object filter	Filters objects (vias, pins, shapes, voids, texts, DRCs, or embedded net names) from the design canvas. Objects smaller than the specified size (in pixel) are not displayed in the canvas when zooming or panning the design. Sliding the bar sets the objects size in pixel (px).
Favorites tab	
Filter favorites	Searches every class and subclass for specific or groups of subclass layers.
Remove all	Removes all listed favorites.
Visibility Panel tab	
Visible classes	Lists the classes that are visible in the Visibility panel in your workspace. Click the Up and Down buttons ( ) to add or remove classes. You can also drag and drop classes.
Available classes	Lists all the classes that are available and can be added to the list of visible classes.
Show global visibility	Displays the Global Visibility field in the Visibility panel. Selected by default.
Show view selection	Adds the View field to enable view listing. Selected by default.
Show stackup selection	Enables display of layer stackup information. Not selected by default. <div> Enabled only if design has more than one zone in stackup.</div>
Show conductors	Enables the display of Conductors under Layer. Selected by default.
Show planes	Enables the display of Planes under Layer. Selected by default.
Show masks	Enables the display of Masks under Layer. Selected by default.
Show layer numbers	Displays layer numbers under <i>All Layers</i> in the Visibility panel.
Show layer IDs	Displays layer IDs under <i>All Layers</i> in the Visibility panel.
Button size	Resizes the color boxes in the Visibility panel.
Spacing	Specifies the spacing between color boxes in the Visibility panel.

### **Related Topics**

- [Tasks to Manage Colors in Your Design](#)
- [highlight](#)
- [assign color](#)
- [shadow](#)



## Controlling Class and Subclass Visibility

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color Dialog box appears.

1. In the Layers tab, click the visibility box for the layer or object you want to change visibility.
  - To make all classes and subclasses visible or invisible, choose *On* or *Off* in the *Global visibility* field. To control visibility for an entire row or column, choose the box next to the *All* column or *All* row. All the subclasses in that class become visible, and an √ appears in each box.
  - To control visibility globally, choose the box next to the intersection of the *All* row and *All* column cell (*All/All* cell), and an √ appears in each box associated with that subclass.
  - To control the visibility of an individual subclass, choose its associated box, and an √ appears in the box.
2. Click *Apply* to update the drawing.
3. Click *OK* to save changes and close the dialog box.

## Related Topics

- [color192](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)

## Customizing a Color

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color Dialog box appears.

1. In the *Available colors* section, choose the color you want to change.
2. Click the color box under *Select* to open *Select Color*.
3. Choose a new color from the *Basic colors* section or the *Custom colors* section. Or, specify the RGB values to create a new shade and click Add to Custom Colors.
4. Click *OK* to save the changes and close the dialog box.

## Related Topics

- [color192](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)

## Importing a Customized Color Palette

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color Dialog box appears.

1. Choose *Load – Load Color Palette*.  
A file browser appears with the filter set to `*.col` and a list of all `.col` files available in the current local working directory. You can manually browse to other directories to open a color file.
2. Choose a customized color palette from the list and click *Open*.  
The customized color palette is applied to the current design.

 To revert to the default Cadence color palette, choose *Load – Load Default Color Palette*.

## Related Topics

- [color192](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)

## Overriding Custom Colors

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color dialog box appears.

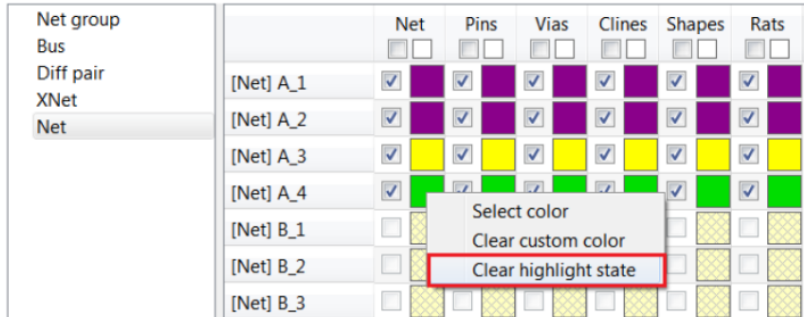
1. Make the *Nets* tab active.
2. Choose the new color in the Available *color* palette.  
The *Selected* color box shows the color you have chosen.
3. Choose the color box next to the net or net element whose color you want to override.  
The color box changes to the color you chose from the *Color* section. The custom color state is enabled, indicated by the ✓ that automatically appears in the box to the left of the color box.

## Related Topics

- [color192](#)
- [Color Dialog Box](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)

## Removing the Highlight State from Nets and Net Elements

1. Hover your cursor over an element's color box.
2. Right-click and choose *Clear Highlight State*.



The highlighting disappears from an element and its name displays in regular typeface in the Nets grid. Its custom color is preserved in the design canvas, and its custom color assignment remains in the Nets grid.

## Related Topics

- [color192](#)
- [Color Dialog Box](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)


## Saving a Customized Color Palette

After you customize a color palette, you can save these settings for use with other designs and for future use with the current design.

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color Dialog box appears.

1. Choose *Apply* after making your color changes.
2. Choose *Save – Save Color Palette*.  
A file browser appears with the filter set to `*.col` in the current local working directory. You can manually browse to other directories to save a color file.
3. Name the customized color palette and choose *Save*.  
The current design's customized color palette is saved.

 To revert to the default Cadence color palette, choose *Load – Load Default Color Palette*.

## Related Topics

- [color192](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)

## Select Color Dialog Box

Use this dialog box to customize shades and hues of color. After moving the control on the vertical sliding bar for luminosity away from the extremes of white or black, you can move the crosshair around the spectrum. All the fields in the dialog box reflect the correct number for the color in the crosshair. You can also type values in the fields to choose a color.

Basic Colors	Displays a selection of typical popular colors.
Custom Colors	Displays user-defined colors.
<i>Hue</i>	Represents the chosen color's hue.
<i>Sat</i>	Represents the chosen color's saturation.
<i>Lum</i>	Represents the chosen color's luminosity.
<i>Red</i>	Represents the amount of red in the chosen color.
<i>Green</i>	Represents the amount of green in the chosen color.
<i>Blue</i>	Represents the amount of blue in the chosen color.
Add to Custom Colors	Moves the color you created with the vertical sliding bar and crosshair to the <i>Custom Color</i> section of available user-defined colors.
Ok	Closes the dialog box and saves your modifications.
<i>Cancel</i>	Closes the dialog box and does not save your modifications.

## Related Topics

- [color192](#)
- [Tasks to Manage Colors in your Design](#)

## Setting Transparency for Shapes

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color Dialog box appears.

1. Use the *Shapes transparency* slider bar in the Display tab to vary the level of intensity.
  - a. Sliding the bar completely to the right (100%) represents a pre-16.0 graphics display.
  - b. Sliding the bar completely to the left (0%) causes previously filled geometry to display with less intensity.

The change takes effect once you have clicked *Apply*.

## Using Shadow Mode

Refer to the `shadow` command.

## Related Topics

- [shadow](#)
- [color192](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)



## Setting Transparency Globally

1. Run the `color192` command.  
Alternatively, choose *Display – Color/Visibility*.

The Color Dialog box appears.

1. Use the *Global transparency* slider bar in the Display tab to vary the level of intensity for the entire drawing.
  - a. Sliding the bar completely to the right represents a pre-16.0 graphics display.
  - b. Sliding the bar completely to the left causes previously filled geometry, such as clines and pads, to display with less intensity.

The change takes effect once you have clicked *Apply*.

## Related Topics

- [color192](#)
- [Select Color Dialog Box](#)
- [Tasks to Manage Colors in your Design](#)

## Tasks to Manage Colors in your Design

These tasks help manage the colors in your design:

- [Assigning Colors or Patterns to Subclasses](#)
- [Assigning Stipple Patterns to Fixed Objects](#)
- [Controlling Class and Subclass Visibility](#)
- [Customizing a Color](#)
- [Saving a Customized Color Palette](#)
- [Importing a Customized Color Palette](#)
- [Setting Transparency Globally](#)
- [Setting Transparency for Shapes](#)
- [Adding Subclasses to the My Favorites Folder](#)
- [Assigning Custom Color and Highlighting from Nets and Net Elements](#)
- [Removing the Highlight State from Nets and Net Elements](#)
- [Overriding Custom Colors](#)

## Related Topics

- [color192](#)
- [Select Color Dialog Box](#)

## colorview create

The `colorview create` command creates or changes a color visibility view, a collection of layer visibility settings that you can apply to subsequent designs using the *View* field on the Visibility form of the control panel. A color view can also display film record visibility settings stored in the current design, unless you suppress the film record names from the list of color views.

You save your settings in a `.color` file that is stored in the current directory.

 The procedures section for this command includes instructions for deleting a color visibility view.

For more details on using color visibility views, see *Getting Started* in the user guide.

### Related Topics

- [colorview load](#)
- [colorview restore](#)
- [Creating a Color Visibility View](#)
- [Creating a Color Visibility View](#)
- [Deleting a Color Visibility View](#)

## Changing a Color Visibility View

1. Run the `colorview create` command.  
Alternatively, choose *View – Color View Save*.

The Color Views dialog box appears.

1. In the *Save view* field:

Enter the name of the file for the color visibility view you want to edit. –or– Choose ... and browse to the file.

1. For *View Replacement Method*, choose a method.
2. If you chosen either of the *Partial update* view replacement methods, change visibility settings in the Color dialog box (using the `color192` command) or in the Visibility form of the control panel.
3. In the Color Views dialog box, choose *Save* and then *Close*.

## Related Topics


- [colorview create](#)
- [Color Views Dialog Box](#)

## Color Views Dialog Box

### Access Using

- Menu Path: *View – Color View Save*

Use this dialog box to create a color visibility view or change an existing one.

Save view	Enter the name of the color visibility view file to which you want to save the current layer visibility settings. The tool automatically appends the <code>.color</code> file extension and stores the file in the current working directory. Choose ... to browse for an existing filename and overwrite its contents.	
View Replacement	Specifies modes for creating a color visibility view.	
	Complete visibility update	Saves the current layer visibility settings to a color view file. When you load the file later, it completely replaces the design's visibility settings, analogous to how the film option to color views works.
	Partial update (changes only)	Allows a color view file to be created that stores only <i>changes</i> to visibility settings. A partial color view does not replace all of a design's visibility settings when loaded. It only replaces the items you changed when you created the color view file. For example, if you changed the color for all DRCs to visible, when you loaded that color view file into a design, only the DRCs would be affected, all changing to visible. All other visibility settings would remain unchanged.
	Partial update using visibility toggle	Functions the same as the <i>Partial update</i> view replacement method because the color view file only stores changes. However, settings that you change toggle when you load the color view file. Toggle means that if the visibility for a layer is on in a design, when you load the color view file, it is turned off. If off, it is turned on.
Preserve zoom level	Includes the zoom points of the current design with the color view.	
Preserve flip state	Includes the flip state of the current design with the color view	
Save	Saves a new or changed file.	
Close	Closes the dialog box without creating a new color view file or saving changes to an existing file.	
	<div> To save a new or changed file, you need to choose <i>Save</i> before clicking <i>Close</i>.</div>	

### Related Topics

- [Changing a Color Visibility View](#)
- [Deleting a Color Visibility View](#)

## Creating a Color Visibility View

1. Run the `colorview create` command.  
Alternatively, choose *View – Color View Save*.

The Color Views dialog box appears.

1. In the *Save view* field, enter the name of the color visibility view.
2. For *View Replacement Method*, choose a method.
3. If you chosen either of the *Partial update* view replacement methods, change visibility settings in the Color dialog box (using the `color192` command) or in the Visibility form of the control panel.
4. In the Color Views dialog box, choose *Save* and then *Close*.

## Related Topics

- [colorview create](#)
- [Deleting a Color Visibility View](#)

## Deleting a Color Visibility View

1. Locate the directory where the file for the color visibility view resides. It has a `.color` extension.
2. Delete the file.

## Related Topics

- [colorview create](#)
- [Color Views Dialog Box](#)
- [Creating a Color Visibility View](#)

## colorview load

The `colorview load` command loads a specified color visibility view. This command treats a `.color` file as a script to be replayed, allowing you to assign function keys to your favorite views.

For more details on using color visibility views, see *Getting Started* in the user guide.

Related commands are [colorview create](#) and [colorview restore](#).

### Access Using

- Menu Path: *View – Color View Load*
- Syntax: `colorview load [<filename>.color]`



### Loading a Color Visibility View

1. Run the `colorview load` command.  
Alternatively, choose *View – Color View Load*.
2. If you did not enter a file name in the command, choose the file from the Colorview Load file browser.
3. Choose *Save* to load the color visibility view.

## colorview restore

The `colorview restore` command restores the previous color visibility view you used in the current session. You can also toggle between two color views using this command. A color visibility view stores a collection of layer visibility settings.

For more details on using color visibility views, see *Getting Started* in the user guide.

Related commands are [colorview create](#) and [colorview load](#).

### Access Using

- Menu Path: *View – Color View Restore Last*

### Applying the Previous Color Visibility View

1. Run the `colorview restore` command to apply the color view that preceded the current color view.
2. To toggle back and forth between the two color views, rerun `colorview restore`.

## comp

The `comp` command selects the specified component in the canvas. The `comp` command is used in conjunction with an active command and is valid when components can be selected.

### Syntax

```
comp <component_name>
```

### Using the comp Command

Perform these steps to select a component in the design:

1. Run `place manual`.
2. Type `comp <component_name>` at the console window prompt.

The component specified is chosen for placement from the list of components in the Placement dialog box.

## compare comp

The `compare comp` command lets you compare third-party die pin or package ball data (LEF/DEF, die text file, BGA text file, die abstract, or OpenAccess (OA) data) to the corresponding footprint in the APD database (.mcm). You can specify either the third-party data or the APD database as your golden data. Golden data is the source against which differences are highlighted. The layout tool creates a report of the logical and physical differences such as pin location, pin name, pin number, cell instance name, pin use, size, shape, orientation, and net assignment.


If you apply the optical shrinkage value to a die, the log file for this command indicates the shrunken value.

### Related Topics

- [Comparing Third-Party Package data with Symbol Footprint in Database](#)

## Comparing Third-Party Package data with Symbol Footprint in Database

1. Run the `compare comp` command.
2. In the Component Compare dialog box, choose either *External File* or *Component* to specify the golden data.
3. From the drop-down list in the *File Type* field, choose the file type representing the external file you are using in the comparison.
4. In the *File* text box, enter the path name for the external file or choose *Browse* to find the path of the external file. If you are using OA, choose the library name, cell name, and view name.

 OpenAccess is available only on UNIX platforms.

If you are using a die abstract, be sure that you have set up the LEF library. Click *Library Manager*, if needed.

5. From the *Reference Designator* drop-down list, select the reference designator of either the die or the package.  
The contents in this list are based on the *File type* you chosen in the *External File* frame.
6. Determine the key for comparison and choose an option in the *Check by* drop-down list in the *Checks* frame. Then check the optional boxes.  
The default primary key for the die abstract file comparison is *Cell Instance*. It is appropriate to compare cell instance names, in cases of cell swaps that involve movement of port and net information, but with the cell block remaining at the same location.  
The default key for all other types is *Pin Location*.
7. Click *Report* to start the comparison check.
8. When you choose a die abstract for comparison, the tool runs and displays an ASCII report listing the logical and physical differences in pin pattern between the external file and component inside the package database. The tool will not make any checks for physical pins for an input die abstract file. This is because the die abstract file does not support physical pins. Physical pins are external pins (polygon or rectangle shapes on metal layer) that are defined in the Pins section of a DEF file.

### Related Topics

- [compare comp](#)

## Component Compare Dialog Box

### Access Using

- Menu Path (APD): *Tools – Component Compare*
- Menu Path (SiP Digital Architect/SiP Layout): *Reports – Component Compare*

<i>Golden data</i>	Specifies the design data against which changes are referenced. You can choose <i>External file</i> or <i>Component</i> .
<i>External File</i>	Specifies that the golden data is from an external file: <i>BGA (.TXT)</i> , <i>DIE (.DEF)</i> , <i>die abstract (.DIA)</i> , <i>DIE (.TXT)</i> , or <i>DIE (OpenAccess)</i> . <i>DIE (.TXT)</i> is the default setting for golden data.
<i>Component</i>	Specifies that the golden data is from the component in the package database. The setting you choose in the <i>File Type</i> field below determines whether it is a die or package component.
<i>External File</i>	
<i>File Type</i>	<p>Specifies the file type of the external file to compare to the design: <i>BGA (.TXT)</i>, <i>DIE (.DEF)</i>, <i>DIE (.TXT)</i>, or <i>DIE (OpenAccess)</i>. The default setting is <i>DIE (.DEF)</i>.</p> <p>⚠ OpenAccess is available only on UNIX platforms.</p> <p>If you choose <i>BGA (.TXT)</i>, only BGA components are listed in the <i>Reference Designator</i> drop-down list in the <i>Component</i> frame. If you choose <i>DIE (.DEF)</i>, or <i>DIE (.TXT)</i>, a list of die components appears in the <i>Reference Designator</i> drop-down list.</p>
<i>File has shrink applied</i>	<p>When you check this box, the tool does not apply existing die shrink information to the die text file for comparison. This field is available only when you choose the <i>Die (.TXT)</i> file type. By default, this check box is unchecked.</p> <p>⚠ The abstract file does not contain only die shrink data.</p>
<i>File</i>	<p>Specifies the path name of the <i>External File</i> (ASCII) containing the pin pattern that is compared to the package component or the <i>lib.defs</i> file if you are using OpenAccess (OA) data.</p> <p>⚠ OpenAccess is available only on UNIX platforms.</p>
<i>Browse</i>	<p><i>Choose this</i> button to locate the path name of the appropriate file with the pin pattern or the <i>lib.defs</i> file if you are using OA. If you chose a <i>.txt</i> file type in the <i>File Type</i> field in the Component Compare dialog box, the <i>Files of type</i> field in the Select file dialog box displays only a list of <i>.txt</i> files in the current working directory.</p>
<i>LEF Library</i>	Specifies the name of the current LEF library. This field displays a library name only if you chose <i>DIE (.DEF)</i> in the <i>File Type</i> list.
<i>Library Manager</i>	<i>Choose this</i> button to change the current LEF library. This button is available only if you chose <i>DIE (.DEF)</i> as <i>File Type</i> .
<i>OpenAccess</i>	The OpenAccess section appears only on UNIX platforms.
<i>Library Name</i>	Specifies the name of the current OA library. This field is grayed out and unavailable if the file type is not OA.
<i>Cell Name</i>	Specifies the name of the cell in the current OpenAccess library. This field is grayed out and unavailable if the file type is not OA.
<i>View Name</i>	Specifies the name of the view in the current OA library. This field is grayed out and unavailable if the file type is not OA.
<i>Component</i>	
<i>Reference Designator</i>	Specifies the reference designator of the die or package in the package database whose pin pattern is being compared. The contents in this list are determined by the <i>File type</i> you chose in the <i>External File</i> frame. For example, if you chose <i>DIE (.DEF)</i> as <i>File Type</i> , this field lists the only reference designators of die components in the database. The first reference designator in the list is selected by default.

## C Commands

### C Commands--compare comp

---

<i>Component Definition</i>	<i>Specifies the component definition name of the chosen reference designator. This field is read-only.</i>
<i>Symbol Definition</i>	<i>Specifies the symbol definition name of the chosen reference designator. This field is read-only.</i>
<i>Include Cell Info</i>	<i>When checked, the report includes information such as driver information connected to cover bumps, such as driver definition, instance and pin names.</i>
<b>Checks</b>	Lists the checks to be performed by the <code>compare comp</code> command.
<b>Check by</b>	<i>Specifies the primary key for comparing pins. By default, the <code>compare comp</code> command performs checks based on <i>Pin Location</i> for all file types except for die abstract file. It performs checks based on <i>Cell Instance</i> for the die abstract file. If you do not select a check on a pin attribute, for example pin use, and there is a difference in that attribute for that pin in the external file and the component, the difference is not reported.</i>
<i>All</i>	Selecting this check box automatically selects all check boxes in this frame. Deselecting this box automatically deselects all check boxes in this frame. The default is <i>Off</i> .
<i>Pin Location</i>	Specifies whether a comparison of pin location is to be performed. The default is on. This field is disabled if the <i>Check By</i> field is by <i>Pin Location</i> .
<i>Pin Number</i>	Specifies whether a comparison of pin numbers is performed. The default is <i>Off</i> .
<i>Pin Use</i>	Specifies whether a comparison of pin uses is performed. The default is <i>On</i> .
<i>Pin Size</i>	<i>Specifies whether a comparison of pin sizes is performed. The default is Off.</i>
<i>Pin Shape</i>	Specifies whether a comparison of pin shapes is performed. The default is <i>On</i> .
<i>Pin Orientation</i>	<i>Specifies whether a comparison of pin orientation is performed. The default is On.</i>
<i>Net Assignment</i>	<i>Specifies whether a comparison of net assignment for pins is performed. The default is Off.</i>
<b>Report</b>	Generates a report based on the selections you make in the <i>Checks</i> frame.
<b>Close</b>	Exits the Component Compare dialog box.
<b>Help</b>	Displays help for the Component Compare dialog box.

## complete

The `complete` option is available on the pop-up menu when an interactive command is active and you are selecting a temporary group of elements. The *Complete* command tells the tool that you have finished choosing elements.



## component assign

The `component assign` command lets you arbitrarily choose groupings to assign to rooms and to assign components to any single group. This is known as design partitioning. You can choose design partitioning by any parameter, such as low-speed, medium-speed, or high-speed groupings, or functional block partitioning.

### Related Topics

- [Assigning Unassigned Components to Rooms](#)
- [Moving Components Between Rooms](#)
- [Removing Assigned Components From a Room](#)

## Assigning Unassigned Components to Rooms

1. Run `component assign` from the console window prompt.  
Alternatively, choose *Logic – Room Assignment*.  
The Room Assignment dialog box appears.
2. Leave one Room field as Unassigned.
3. Leave the RefDes and Device filters for the unassigned room set to \* to list all components or use wildcards to list the target components.
4. Leave the other Room field as is or:
  - a. Enter a room name.or
  - a. Choose and choose a room name.
5. Leave the RefDes and Device filters for the other room set to \* to list all components or use wildcards to list the target components.
6. Choose an individual component refdes/device name.  
The chosen refdes/device name moves from the unassigned list box to the list box for the specified room.
7. To move all components from the unassigned list box to the list box for the specified room, choose All.

 The percentage figure following the Available Room Area Used label represents the amount of room area the assigned components require.

## Assigning Unassigned Components Using the Design

If components are already placed, you can use the design as follows:

1. To quickly find a component, enter its reference designation in the Locate Refdes field near the bottom of the dialog box.  
The room to which the component is assigned appears to the right of the Room label.
2. Choose *Highlight Unassigned*.  
All unassigned components in the design are highlighted.
3. Choose a highlighted component in the design.  
The component is dehighlighted. The refdes and device name are moved to the component list box of the specified room and highlighted.

## Related Topics

- [component assign](#)
- [Removing Assigned Components From a Room](#)

## Moving Components Between Rooms

- Run `component assign`.  
Alternatively, choose *Logic – Room Assignment*.  
The Room Assignment dialog box appears.

### ***Moving Components Using the Room Assign Dialog Box***

1. Specify two different rooms in the Room fields.  
Neither Room field should be set to Unassigned.
2. Choose an individual component refdes/device name.  
The chosen component moves from one list box to the other.
3. To move all components from one list box to the other, choose All.

### ***Moving Components Using the Design***

1. In the dialog box, specify two different rooms in the Room fields. Neither Room field should be set to Unassigned.
2. Choose a highlighted component in the design.  
The component is de-highlighted in the design. The refdes and device name are moved to the opposite list box and highlighted.

## Related Topics

- [component assign](#)
- [Room Assign Dialog Box](#)

## Removing Assigned Components From a Room

1. Run `component assign`.  
Alternatively, choose *Logic – Room Assignment*.  
The Room Assignment dialog box appears.
2. Choose a Room field and change it to Unassigned.
3. Change the other Room field to choose the room from which you remove one or more components.
4. Choose an individual component refdes/device name in the list box of the specified room.  
The component moves from the specified room list box to the Unassigned list box.
5. To move all components from the specified room list box to the Unassigned list box, choose ALL.

## Removing Assigned Components Using the Design

1. Choose Highlight for the room in which you want to work.  
In the design, all components assigned to the specified room are highlighted.
2. Choose a highlighted component in the design.  
The component is de-highlighted. The refdes and device name move to the Unassigned list box and are highlighted.

## Related Topics

- [component assign](#)
- [Room Assign Dialog Box](#)
- [Assigning Unassigned Components to Rooms](#)

## Room Assign Dialog Box

### Access Using

- Menu Path: *Logic – Room Assignment*

### Component Assignment Area

<i>Room</i>	Pull-down selects existing rooms or Unassigned parts.
<i>Refdes Filter</i>	Limits the search for reference designations.
<i>Device Filter</i>	Limits the search for devices.
<i>Highlight</i>	Highlights parts in room chosen in Room pull-down.
<i>Available Area Used</i>	Shows percent of room area parts occupy.

### Sort By:

<i>Refdes</i>	Sort parts allowed by filters by reference designators.
<i>Device</i>	Sort parts allowed by filters by device name.

### Move:

<i>All -&gt;</i>	Move all from left to right.
<i>&lt;- All</i>	Move all from right to left.

### Locate Component Area

<i>Refdes</i>	Text box to type in component reference designator you want to locate.
<i>Room</i>	Readout shows what room reference designator is assigned to.

### Related Topics

- [Moving Components Between Rooms](#)
- [Removing Assigned Components From a Room](#)

## component fix

The `component fix` command lets you identify critical components to be fixed during floorplanning. Fixed components cannot be moved during layout; they must first be unfixed before they can again be moved.

### Access Using

- Menu Path: *Place – Fix Component Location*

## Fixing Components

Follow these steps to fix components in the design:


1. Run `component fix` from the console window prompt.  
Alternatively, choose *Place – Fix Component Location*.  
The Component Fix Location dialog box appears.
2. Choose *View – Zoom By Points* and choose two (diagonally opposed) zoom window points such that components of interest are clearly visible.  
While the Component Fix Location dialog box is open, all fixed components are highlighted.
3. Do one of the following:
  - a. Choose an unfixed component in the design to mark it as fixed.

or

  - a. Choose a fixed component in the design to mark it as unfixed.
4. Choose *OK* to dismiss the Component Fix Location dialog box.

## component height

The `component height` command lets you attach properties defining a height restriction to a component keepout or place bound rectangle. Keepouts allow component symbols whose height is below a minimum or above a maximum to be placed in that area; place bound rectangles define component boundaries in terms of height and component geometry.

 The command is available only with Allegro X Advanced Package Designer (APD).

### Related Topics

- [Adding Height Specifications to Component Keepout Areas](#)
- [Adding Height Specifications to Place Bound Rectangles](#)



## Adding Height Specifications to Component Keepout Areas

1. Run `component height` from the Command line.  
Alternatively, choose *Place – Fix Component Location*.
2. In the *Options* tab, set the active class and subclass to COMPONENT KEEPOUT/ALL.
3. Choose a component keepout area, or choose *Add Rectangle* or *Add Shape* from the right-button pop-up to add one. (If the shape or rectangle does not appear, check to see that the visibility for that layer is on.)  
The *Options* tab is configured for the component height command. If you chose an existing shape, the current minimum and maximum heights are displayed, as well as a graphic view that illustrates how the height values are treated. (Note that the graphic is not dynamic; that is, it does not change according to the specific height values that you enter.)
4. Enter the minimum and maximum height information in the text fields of the *Options* window if you are creating a new keepout area.  
The heights that you enter are expressed as a range in user units (mils, inches, or millimeters) in the *Design* tab of the Design Parameter Editor, available by choosing *Setup – Design Parameters* (`prmed` command). This defines the vertical dimension of the keepout area from minimum value to maximum value. The minimum must be greater than or equal to 0. The maximum can be anything up to infinity (the default).
5. When you have attached the restrictions you want, choose *Done*.

 You can also add component heights using the property edit command.

## Related Topics

- [component height](#)

## Adding Height Specifications to Place Bound Rectangles

1. Run `component height` from the Command line.  
Alternatively, choose *Place – Fix Component Location*.
2. In the Options tab, make sure the active class and subclass is set to COMPONENT GEOMETRY/PLACE\_BOUND\_TOP or PLACE\_BOUND\_BOTTOM.
3. Choose a component shape, or choose *Add Rectangle* or *Add Shape* from the right-button pop-up to add one.  
The *Options* tab displays for the component height command. If you chose an existing shape, the current minimum and maximum heights are displayed, as well as a graphic view that illustrates how the editor treats the height values. (Note that the graphic is not dynamic; that is, it does not change according to the specific height values that you enter.)
4. If the place boundary object is new, enter the minimum and maximum height information in the text fields of the *Options* tab.  
The heights that you enter are expressed as a range in user units (mils, inches, or millimeters) in the Drawing Parameters dialog box ([drawing param](#) command). You can also edit them on the *Design* tab of the Design Parameter Editor by choosing *Setup – Design Parameters* ([prmed](#) command). This defines the vertical dimension of the shape area from minimum value to maximum value. The minimum must be greater than or equal to 0. The maximum can be anything up to infinity (the default).
5. When you have attached the restrictions you want, choose *Done*.

## Related Topics

- [component height](#)
- [Component Height Command: Options Panel](#)

## Component Height Command: Options Panel

### Access Using

- Menu Path: *Setup – Areas – Component Height*

<b>Active Class and Subclass</b>	Ensure that these classes are the same as the one on which you set the component keepout boundary. Allowable class/subclass combinations are: COMPONENT_KEEPOUT/ANYCOMPONENT_GEOMETRY/PLACE_BOUND_TOP and PLACE_BOUND_BOTTOM.
<b>Line lock</b>	Active when Add Rectangle or Add Shape is chosen from the pop-up. Defines the line segment and the angle of the corner when the segment changes direction.
<b>Minimum height</b>	Sets the minimum height restriction for the package or the keepout area, based on the chosen class.
<b>Maximum height</b>	Sets the maximum height restriction for the package or the keepout area, based on the chosen class.
<b>Clear</b>	Empties the height fields of user-edited values
<b>Reset</b>	Restores the values to their non-user edited values

### Related Topics

- [Adding Height Specifications to Place Bound Rectangles](#)

## compose line

The `compose line` command lets you create complex new construction lines by merging construction lines from a particular class and subclass. You can compose multiple lines that touch or almost touch, or that cross into one line object.

With the `compose line` command, you can compose one multi-segment path object from a set of disjoint paths. You can also use the Wirebond Select command and choose the option *Add* from the pop-up menu to create wire bond guide paths.

Specify the parameters in the *Options* panel and then run `compose line` multiple times on different selection of lines and arcs.

### Related Topics

- [Composing Lines and Arcs into an Object](#)

### Compose Line Command: Options Panel

#### Access Using

- Menu Path: *Tools – Compose Line*

<i>Line intersection</i>	
<i>Use auto gap</i>	Check this box when a gap exists between two lines. The tool automatically extends the line segments to form a closed angle. The gap used in extending the lines appears in the <i>Maximum gap</i> field. If the gap used in extending the lines is not large enough to close all the gaps, right-choose and choose <i>Close Gap</i> from the pop-up menu. Based on the number of lines being merged, you may have to do this multiple times to close all gaps. By default, the <i>Use auto gap</i> box is checked. When the <i>Use auto gap</i> box is unchecked, the tool enables the <i>Maximum gap</i> field. Any gap larger than the maximum gap value is not automatically closed.
<i>Maximum gap</i>	Enter a maximum gap value to close the gaps between lines. All gaps that are less than the maximum gap value are automatically closed by the command. You can edit this field only if you uncheck the <i>Use auto gap</i> box. The default value of the field is the maximum gap value last used or 0.0 when you first launch the command.
<i>Round corners</i>	Check this box to add arcs to create round corners. You must specify the arc radius to be used. By default, this box is unchecked. When rounding corners, the tool always extends or trims the line segments to form a closed corner, and then fillets the corner with an arc of the given radius. As a result, sometimes this option causes two new line segments and one arc to be added.
<i>Radius</i>	<i>Specifies the radius that the tool uses when rounding corners. The default value is the last value used or 254 when you first launch the command. You can edit this field only when you check the Round Corners box.</i>
<i>Active class</i>	Specifies the active class for the placement of the new line. The default class is <i>Substrate Geometry</i> .
<i>Add line to subclass</i>	Specifies the subclass for the placement of the new line. The default setting is the <i>Wb_Guide_Line</i> (wirebond) subclass.
<i>Delete original lines</i>	Specifies that the construction lines, which are on a different layer than the destination layer, are hidden and then deleted when you exit the command. If you do not check this box, the original lines, which are on a different layer, are preserved. The default setting is that the box is checked. If the chosen construction lines are on the same subclass as the subclass indicated in the <i>Add line to subclass</i> field, then the original construction lines are deleted and replaced by the single lines composed from the construction lines.

### Pop-up Menu

When you are in the `compose line` command and right-choose, a pop-up menu with these options appears:

## C Commands

### C Commands--compose line

---

<i>Done</i>	Commits the changes.
<i>Oops</i>	Rolls back the last action taken.
<i>Next</i>	Commits the current action and starts a new action.
<i>Cancel</i>	Rolls back all the changes, and exits from the command.
<i>Close Gap</i>	Increases the tolerance gap and closes the smallest gaps of all line gaps
<i>Temp Group</i>	Lets you pick multiple lines and arcs.

## Composing Lines and Arcs into an Object

1. Create lines and arcs by using commands such as *Add – Line*, *Add – Arc w/Radius*, and *Add – 3pt Arc*.
2. Type `compose line` at the console window prompt.  
Alternatively, choose *Tools – Compose Line*.
3. Set the parameters in the *Options* tab in the Control Panel for the methods used in merging and trimming existing lines.
4. Specify the layer on which the converted lines should reside.

⚠ If you use the compose line command with the custom wire bond feature, be sure to use the `WB_Guide_Line` subclass as the tool looks for the composed lines on that subclass.

5. Select the set of lines and arcs to merge:
  - a. Use window selection  
or
  - b. Right-choose and choose *Temp Group* from the pop-up menu and then pick the elements.

The lines are composed into one object, and they appear in the Design Window.

6. To close unwanted gaps between lines, right-choose and choose *Close Gap* from the pop-up menu.  
This closes the smallest remaining gap. You can do this several times to close more than one gap. Or you can increase the value of *Maximum gap* in the *Options* tab. Note that the tool does not close a gap if doing so, it forms a closed path.
7. To commit the result, right-choose and choose *Done* from the pop-up menu.

### Line Intersection Method

The extend and trim methodologies determine the intersection of two lines:

- If two lines cross each other, they are trimmed to form a closed angle.
- If two lines are parallel, an extra line segment is added to connect the two end points with the smallest gap.
- If a gap exists between two lines, this option automatically extends the line segments to form a closed angle.
- When you extend the lines and the resulting intersection point lies outside the drawing extent, then an extra line segment is added (for example, chamfer) to connect the two end points inside the drawing extent.

There are two options for handling corners:

- Chamfer or Trim: If two lines cross each other, they are trimmed to form a closed angle. If a gap exists between the two lines, a line segment is added to connect the ends of the lines.
- Rounded Corner: To add an arc, you must specify the arc radius to use. When adding an arc, the tool extends or trims the line segments to form a closed corner, and then fillets the corner with an arc of the specified radius.

### Related Topics

- [compose line](#)

## compose shape

The `compose shape` command converts a group of lines and arcs into a shape. These lines and arcs can be created from DXF or Gerber input. This feature is especially useful for laying out design-specific ground and power planes.

For more details on composing and decomposing shapes, see *Preparing the Layout* in the user guide.

A related command is [decompose shape](#).

### Related Topics

- [Composing a Shape](#)
- [decompose shape](#)

## Compose Shape Command: Options Panel

### Access Using

- Menu Path: *Shape – Compose Shape*

✓ The Design Parameter Editor is also available to set up shape parameters. To edit global dynamic shape parameters, static shape parameters, and split plane parameters, choose *Setup – Design Parameters* (`prmed` command), then choose the *Shapes* tab.

<i>Use auto gap</i>	Choose to automatically calculate a <i>Maximum gap</i> value based on the options you choose to form a closed shape. By default, this option is chosen. When you first run <code>compose shape</code> , the <i>Maximum gap</i> value becomes 0 and deselected as a result. Each time you execute the command to create a shape, the tool updates the <i>Maximum gap</i> field with the automatically calculated value. When this field is unselected, the tool uses the value you enter in the <i>Maximum gap</i> field to form a closed shape.
<i>Maximum gap</i>	<p>Enter a value that the tool uses to determine whether to extend two unconnected line segments up to a specified distance or add a new line segment to eliminate an existing gap and form a closed corner. This option is especially useful when selecting a group of lines and arcs. For an example of how this works, see "<a href="#">Connecting the Gap Between Lines</a>". If you choose <i>Use auto gap</i>, this field displays the last value previously calculated or entered and is deselected as a result. Each time you execute the <code>compose shape</code> command to create a shape, the tool updates this field with the automatically calculated value, which you can use as a baseline and then modify as necessary. If you disable the <i>Use auto gap</i> option, the tool uses the value you enter here to form a closed shape.</p> <div style="border: 1px solid #fff3cd; padding: 5px; margin-top: 10px;"> <p>⚠ For best results, ensure that the grid increment is close to the size of the largest gap you want to connect. Line segment gaps that exceed more than twice the grid increment may reduce the efficiency of the <code>compose shape</code> command.</p> </div>
Delete original lines	Set to delete source lines. Set by default.
Delete unconnected lines	Choose this option if you want to delete lines in a specified group that cannot be connected. Sometimes, line segments can be small and missed, and this option cleans up the shape design.
Round corners	Choose this option to connect two unconnected lines and replace an existing closed angle with an arc of a specified radius. For an example of how this works, see " <a href="#">Rounding a Corner and Trimming</a> ".
Radius	Specify the radius of the arc with which to connect lines or to transform angles.
Active class	Specify the class to which the shape is added when you choose <i>Done</i> . The default class is ETCH/CONDUCTOR.
Add shape to subclass	Specify the subclass of the class to which the shape is added when you choose <i>Done</i> . The default subclass is TOP.
Filled shape	Set for a filled shape. Set by default.
Assign net name	This field displays the net you specify or choose from the Net Names dialog box, which appears when you choose ... .



## Composing a Shape

1. Run the `compose shape` command.  
Alternatively, choose *Shape – Compose Shape*.
2. Set fields in the Options tab to resolve unconnected lines. Overlapping lines are automatically trimmed.
3. Do any of the following to choose the line and arc segments:
  - Choose on them
  - Outline an area to choose a group of segments.

The compose shape command connects or extends line segments as you choose them. See [Before and After: Applying the compose shape Command](#) for examples of before and after images of shapes created with the compose shape command.

For complex shapes, connections and extensions may change with each segment you pick.

Voids can be created by creating a shape within a shape. The `compose shape` command automatically determines voids and solids based on which shapes are nested inside other shapes.

1. If needed, undo several actions by selecting *Undo* from the pop-up menu. You undo one action at a time.

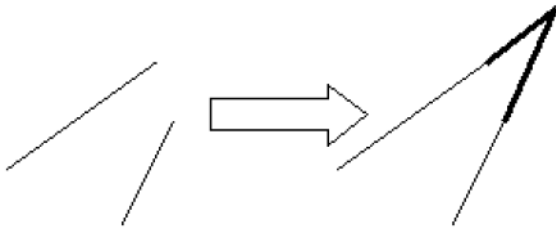
You can redo undone actions similarly by selecting *Redo* from the pop-up menu.

1. If you chosen unconnected elements, choose *Close Shape* from the pop-up menu. The tool creates a shape.
2. Choose *Done* from the pop-up menu.

### Examples

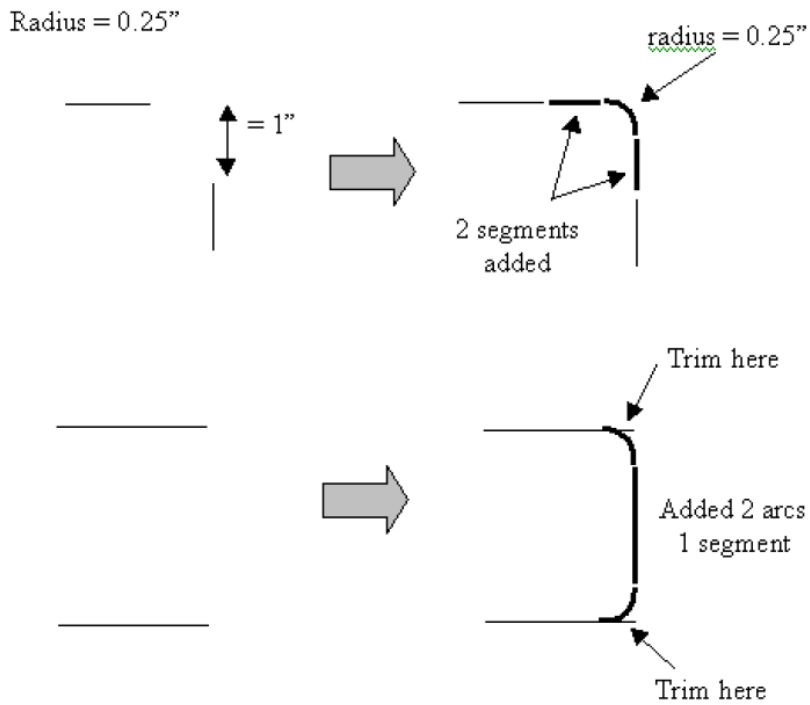
As an example of how the *Maximum gap* field in the Options tab works, the following figure shows how the tool adds the bold portions to extend the two lines a specified distance so that they meet.

**Figure: Connecting the Gap between Lines**



The following figure is an example of how the *Rounded corners* field in the Options tab works.

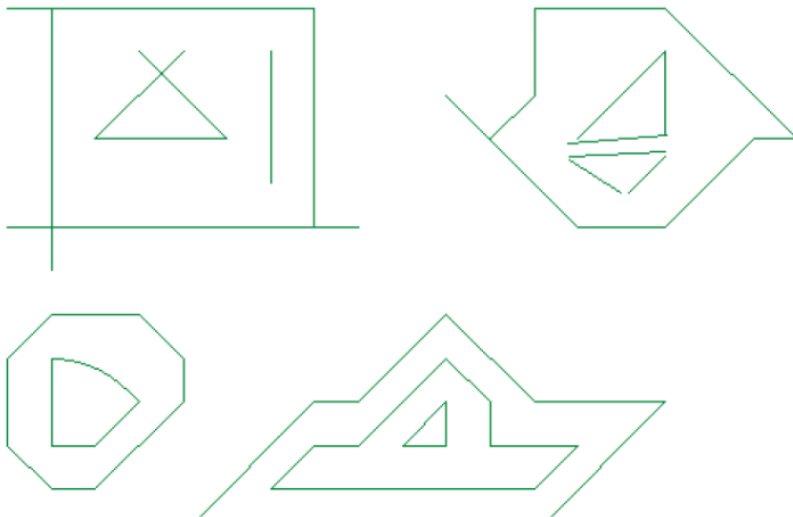
**Figure: Rounding a Corner and Trimming**



#### Before and After: Applying the compose shape Command

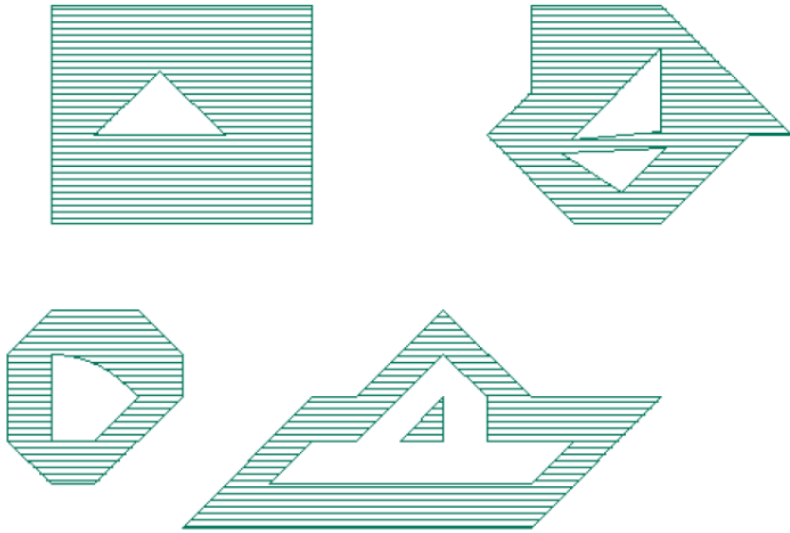
In the following figure, notice how these loosely defined shapes have gaps and crossed segments.

**Figure: Before Using Compose Shape**



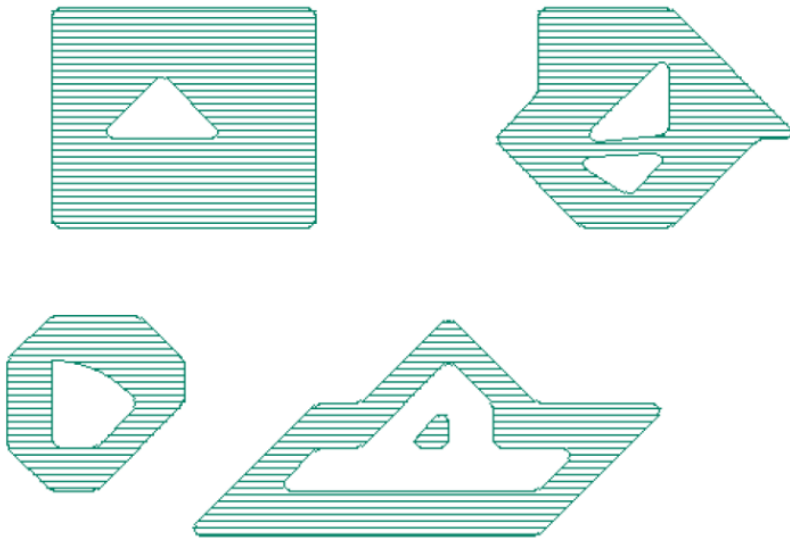
The following figure shows the effect of the `compose shape` command with the *Maximum gap* field in the *Options* tab set to 100.00 and the *Delete unconnected line* option set.

**Figure: After Using Compose Shape (Maximum Gap with Automatic Trimming)**



The following figure shows the effects of the Compose Shape tool with the *Round corners* option set to a *15.00* radius and the *Delete unconnected line* option set.

**Figure: After Using Compose Shape (Round Corner)**



## Related Topics

- [compose shape](#)

## compose symbol from geometry

The `compose symbol from geometry` command extracts information from the geometry data read from GDSII or other file formats supported by the application and creates a die symbol with die boundary and pins, along with pin names, pin numbers, and net names.

You can export IC layout data to a stream file and then import geometry from the file into APD to create a die using this command.

⚠ If the geometrical data is not formed into polygons, use the [compose shape](#) command.

### Related Topics

- [compose shape](#)
- [Place Die dialog box](#)
- [Creating a Symbol Using Geometry Data Files](#)

### Compose Symbol from Geometry Dialog Box

#### Access Using


- Menu Path: *Add – Compose Symbol From Geometry*

<i>Symbol Type</i>	Select the symbol type you want to generate. The options for symbol type are <i>Die</i> , <i>Bga</i> , and <i>Discreet</i> . Also select the outline class (Outline Class) and subclass (Outline Subclass) where the die outline is present.  ⚠ If no outline class or subclass is selected, it will take maximum size occupied by all polygons for pins
<i>Width (X)</i>	Shows the width of the die outline.  ⚠ Any geometry outside the outline is ignored for width and height.
<i>Height (Y)</i>	Shows the height of the die outline.
<i>Symbol Name</i>	Specify the <i>Symbol Name Class</i> and <i>Symbol Name Subclass</i> based on which you can select a Symbol Name from the list.
<i>Geometry From</i>	Specify the source geometry of the symbol, which can be either <i>IC</i> (definition) or <i>Placed in Package</i> (instance). For an instance ( <i>Placed in Package</i> ) source, you can flip the symbol and specify the rotation. <i>IC</i> is selected by default.
<i>Symbol Pins</i>	Select the Class and Subclass where pin location and geometries are defined.
<i>Pin Geometry</i>	Select the Class and Subclass where pin geometries are defined.
<i>Merge overlapping pins</i>	Choose this option to merge overlapping pins. The option is enabled by default.
<i>Pin Name</i>	Select the Class and Subclass where pin names are defined. A pin name is associated with a text located within the geometry of the pin.
<i>Delimiter</i>	Specify the delimiter character. This is an optional field. You can specify either <i>Colon</i> or <i>Semicolon</i> , or select <i>Other</i> to specify a different character. The default delimiter is <i>None</i> .

## C Commands

### C Commands--compose symbol from geometry

---

<i>Import Pin Numbers</i>	Choose this option to create pin numbers based on imported geometry data. You must also specify the pin number class and subclass. Otherwise, the pin numbers will be generated
<i>Pin Number Class/Subclass</i>	Select the Class and Subclass where pin numbers are defined. Pin number is associated with a pin if that text is located within the geometry of the pin.
<i>Import Nets</i>	Choose this option to create and add nets to pins based on imported geometry data.
<i>Net Name Class/Subclass</i>	Select the Class and Subclass where nets are defined. Net name is associated with a pin if that text is located within the geometry of the pin.
<i>Find Pins</i>	<p>Populates the pins grid table with pin data with valid pins on the class and subclass specified. The pin grid displays width, height, pin name, and pin use. If text does not exist for a pin geometry, the pin number is shown as the pin name.</p> <div> By default, the grid lists data in ascending order of pin names. You can change the order by right clicking on a column header and choosing an option from the pop-up menu.</div>
<i>Remove Geometry From Design</i>	Choose this option to delete all geometries on the layers where pin, pin text, and outline are located from the design after the die symbol is created. Not selected by default.
<i>Create Symbol</i>	Creates the die symbol and opens the Place die dialog box.
<i>Cancel</i>	Exits the command without making any changes.

#### Related Topics

- [Creating a Symbol Using Geometry Data Files](#)

## Creating a Symbol Using Geometry Data Files

To create a symbol in the layout editor using a geometry data file, perform these steps:

1. Import the stream file.
2. Invoke the Compose Die From Geometry dialog box either using the `compose die from geometry command` or by choosing *Add – Standard Die – Compose From Geometry*.
3. Specify the class and subclass information and delimiter, if needed.
4. Click *Find Pins* to generate the pin grid.
5. Click *Create Symbol* to open the Place Die dialog box
6. Specify the settings for the tabs.
7. Click *Place*.

## Related Topics

- [compose shape](#)
- [compose symbol from geometry](#)
- [Compose Symbol from Geometry Dialog Box](#)

## Place Die dialog box

General tab	
<i>Symbol Name</i>	Specify the name of the symbol.
<i>RefDes</i>	Specify the RefDes.
<i>Die Attachment</i>	Specify the type and orientation of the die. The options for type are, <i>Wire bond</i> and <i>Flip chip</i> (default). The options for orientation are, <i>Chip up</i> and <i>Chip down</i> (default).  ⚠ If chip down is selected, the placed symbol will be a mirror image of the geometry.

Placement tab	
<i>Pin Layer</i>	Select the layer from the list.
<i>Origin X</i>	Specify the X-origin of the die placement. By default, origin is the center of the design. Optional field.
<i>Origin Y</i>	Specify the Y-origin of the die placement. By default, origin is the center of the design. Optional field.
<i>Rotation</i>	Specify the rotation from the list.

### Symbol Outline tab

<i>Change Symbol Outline (pre-shrink/pre-scribe values)</i>	Check to change the pre-scribe/ pre-shrink value of the symbol outline. Not checked by default.
<i>Center around origin</i>	Select to center symbol around the origin with specified <i>Width</i> and <i>Height</i> .
<i>Extents</i>	Select to specify the symbol size with specified values for coordinates: <i>Bottom left X</i> , <i>Top right X</i> , <i>Bottom left Y</i> , and <i>Top right Y</i> .
<i>Apply IC fabrication optical shrink by</i>	Specify the percentage by which the die should shrink. If you do not specify a value in the editable field, 0% is assumed. Not selected by default.
<i>Add scribe width</i>	Specify the scribe width. If only <i>North</i> value is filled, the other sides are populated with this value. The default value for all fields is 0UM. Not selected by default.

Padstack tab	
<i>Create New Padstack Based on Source geometry</i>	Select to create a a padstack for each different geometry. Selected by default.
<i>Padstack Name</i>	Specify the padstack name. By default, <i>PADSTACK</i> is used. This is used as a seed name and incremental number is added to specify the different padstacks such as <i>PADSTACK1</i> , <i>PADSTACK2</i> , and son.
<i>Available padstack</i>	Select an available padstack from the list. One padstack is used for all pins on each die. The pins are placed at the origin of the padstack based on the polygon for pins in the input data.
<i>Load from disk</i>	Specify a file for the padstack. One padstack is used for all pins on each die. The pins are placed at the origin of the padstack based on the polygon for pins in the input data.

Pin Number tab	
<i>Ordering</i>	Specify the order in which you want the pins numbered. Use the drop-down list to choose the numbering scheme. The graphical display changes to display the numbering scheme you choose.
<i>Start at</i>	Specify from where the numbering is to start.

## C Commands

### C Commands--compose symbol from geometry

---

<i>Label with letters before numbers</i>	Select to specify the option to number alphanumeric patterns <i>A1</i> , <i>A2</i> ...
<i>JEDEC standard</i>	Select to specify the numbering that conforms to the JEDEC standard, omitting the letters I, O, Q, S, X, and Z when generating alpha or alphanumeric pin numbering.
<i>Pad letter with A's</i>	Select to specify that the tool pads lettering with A characters to ensure that all pin numbers have two or three letters in their names. The editor counts the number of pins, and depending on whether you chose this option, assesses if sufficient rows or columns need pin numbers whose letter field requires two or three letters. For example, if two letters are required, and you check the option, the pin number sequence is: AA01, AB01, . . . , AZ01, BA01, BB01, . . . , BZ01. If you do not choose the option, the above numbering sequence is: A01, B01, . . . , Z01, AA01.
<i>Pad number with zeros</i>	Select to specify that the tool inserts leading zeros. For example, if a package of 10x10 pins is generated using Number Right Letter Down ordering, the resulting numbers are: A001, A002 . . . . . A010B011, 012 . . . . . B020C021, C022 . . . . . C030J091, J092 . . . . . J100.
<i>Staggered pin configuration</i>	Select to enable staggered pin placement in the chosen grid.

#### Related Topics

- [compose shape](#)
- [compose symbol from geometry](#)



## **compress\_route**

An internal command.

## cond length report

 Available only in Allegro X Advanced Package Designer (APD).

The `cond length report` command generates a Conductor Length Report intended for wire bond packages (though you may also use it for flip-chip designs). It is designed to support your package analysis programs by reporting unique elements such as bond wires, bond fingers, vias, connect lines, and plating bars. The format of the report lets you easily import the data into any commercial spreadsheet for editing or other modification.

### Access Using:



- Menu Path: *Manufacture – Documentation – Conductor Length Report*

### Related Topics

- [Generating the Conductor Length Report](#)

### Conductor Length Report Structure

The conductor length report contains a header section and a data section. The header section records when the report was generated, the name of the current design, and the database units of the design. The data section is made up of columns that provide the following information:

<i>Net Name</i>	The nets in the design, sorted alpha-numerically.
<i>Finger</i>	The value of the property BOND_PAD assigned to each bond finger. A blank indicates there is no bond finger, or that the property has not been assigned or has not been given a value.
<i>Wire Length</i>	The three-dimensional length of the bond wire from the die pad to the bond pad or power/ground ring. Be sure to configure your wire profiles before running this report.
<i>&lt;Layer name&gt; Conductor</i>	<p>These columns reflect each conductor and plane layer as named in the <i>Layout Cross Section</i> dialog box. Each named column represents the sum of the length of all the connect line (cline) segments on that layer (for example, TOP_COND, BOTTOM_COND, and so on.)</p> <p> <b>Note:</b> BONDING_WIRE and DIELECTRIC layers are ignored as are fillet lines.</p>
<i>Total Conductor</i>	The values in this column contain the sum of the values in the layer name (conductor) columns.
<i>Via Length</i>	<p>The length of each via, derived from the thickness of the layers that the via penetrates. If a net has more than one via, the value is the sum of all the vias on the net.</p> <p> BONDING_WIRE layers are ignored.</p>
<i>Total Conductor + Wire + Via</i>	The values in this column are the sum of values in the columns for <i>Wire Length</i> , <i>Total Conductor</i> , and <i>Via Length</i> .
<i>&lt;Layer name&gt; Plating</i>	The values in this column are the sum of all connect line segments on a given layer (TOP_COND, BOTTOM_COND, and so on.) The clines are measured from the edge of the package to the termination point at a pin, via, plane, or T-connection.
<i>Total Plating</i>	The values in this column contain the sum of the values in the layer name (plating) columns.
<i>Overall Total</i>	The values in this column contain the sum of the values in the columns <i>Total Conductor + Wire + Via</i> and <i>Total Plating</i> .


Layers in your design that contain no conductors or plating bars will not generate a column in the report.

## Generating the Conductor Length Report

To generate a complete conductor length report, the following elements need to be present in your design:

- Die (component class IC)
- Package (component class IO)
- Rectangular package outline (component class SUBSTRATE\_GEOMETRY)
- Bond finger labels (In order to output this type of data, the property BOND\_PAD must be attached and include a value. See [bpa](#) for details.)

A plating bar (component class PLATING\_BAR) may also be present, but is optional.

 Database elements not included in the conductor length report are:

- Bond finger length
- Connect lines not associated with a net
- Shapes

Perform these steps to generate the report:

1. Run the `cond length report` command.

Alternatively, choose *Manufacture – Documentation – Conductor Length Report*.

You are prompted to create an output file name. (The default name is *conductor\_length.rpt*.) Output files conform to standard revision control; that is, *conductor\_length.rpt,1... conductor\_length.rpt,2*, etc.

2. Enter a file name and location for the report.
3. Choose *OK* to generate the report.

### Example

Design: example

Date: 6/3/99 10:45am

Unit: mils

Net Name	Finger	Wire Length	TOP_COND Conductor	BOTTOM Conductor	Total Conductor	Via Length	Total Conductor + Wire + Via	TOP_COND Plating	BOTTOM Plating	Total Plating	Overall Total
BAND_GAP_R	BF35	123.72	187.45	35.36	222.80		346.53	210.24	0.00	210.24	556.76
BUS_DATA_SENSE_0	BF70	118.24	200.27	35.36	235.63		353.87	209.99	0.00	209.99	563.87
BUS_DATA_SENSE_1	BF71	117.14	220.27	35.36	255.62		372.76	199.05	0.00	199.05	571.81
BUS_DATA_SENSE_2	BF72	118.67	131.56	35.36	166.92		285.59	276.98	0.00	276.98	562.57
BUS_DATA_SENSE_3	BF73	117.47	148.60	35.36	183.95		301.43	267.04	0.00	267.04	568.46

### Related Topics

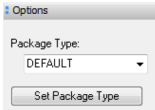
- [cond length report](#)
- [bpa](#)

## config package type

The `config package type` command lets you set the lead frame package type in the `IC_PACKAGE_TYPE` property for a design. The specified lead frame package type information is used by the `wirebond bond leads` command. When you run the `wirebond bond leads` command, the lead frame package type impacts which bond locations are wired on a lead. It also impacts the lead frame offset distances if the distances are defined in the wire profile set for the design. You need to set the lead frame package type only once for a design.

### Access Using

- Menu Path: *Route – Wire Bond – Set Package Type*



Package Type	Select the lead frame package type for a design. Shows the current lead frame package type if already selected for the design. If no package type is selected, it shows <i>DEFAULT</i> . The values available in the list for selection are <i>DEFAULT</i> , <i>QFP</i> , <i>QFN</i> , and package types defined for wire profiles in the design. You can also type in a value if none of listed types match the design type.
Set Package Type	Click to confirm the package type and save it in the design.

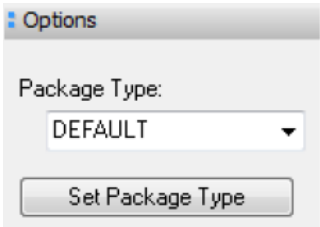
### Related Topics

- wirebond bond leads*

## Config Package Type Command: Options Panel

### Access Using

- Menu Path: *Route – Wire Bond – Set Package Type*



Package Type	Select the lead frame package type for a design. Shows the current lead frame package type if already selected for the design. If no package type is selected, it shows <i>DEFAULT</i> . The values available in the list for selection are <i>DEFAULT</i> , <i>QFP</i> , <i>QFN</i> , and package types defined for wire profiles in the design. You can also type in a value if none of listed types match the design type.
Set Package Type	Click to confirm the package type and save it in the design.

Wirebond select

#### Related Topics

- *wirebond bond leads*

## config substrate layers

The `config substrate layers` command defines the top and bottom substrate layers of a wire bond package design. If your design was created in a release earlier than Release 16.0 and the substrate layers are not defined, or are defined incorrectly, use this command to identify the correct substrate region of your cross-section. You can determine if a design's cross-section is set up correctly by running the [xsection](#) command. The layers that belong to the package substrate should appear in the red region, and the wirebond die layers should appear in the blue region. If no layers or the wrong layers appear in the blue region, run the `config substrate layers` command at the console prompt. Incorrectly defined substrate regions can result in problems such as the die stack objects being flagged as in a cavity.

### Related Topics

- [Defining the Substrate Layers of a Wirebond Package Design](#)

## Defining the Substrate Layers of a Wirebond Package Design

1. Open a pre-Release 16.0 wire bond design.
2. If design uprev messages appear in the console window, run the `uprev` command.
3. Run the `define lyrstack` command to check the cross section.
4. If the top and bottom substrate layers are not correctly defined, run the `config substrate layers` command to correctly define those layers.
5. Save the design.

## Related Topics

- [config substrate layers](#)

## Substrate Layer Config Dialog Box

### Access Using

- Command Line: `config substrate layers`

<i>Top substrate layer</i>	Specifies the top bounding physical layer of the substrate.
<i>Bottom substrate layer</i>	Specifies the bottom bounding physical layer of the substrate.
<i>OK</i>	Resets the top and bottom substrate layers to those that you specified in the dialog box.
<i>Cancel</i>	Cancels the command and dismisses the dialog box.
<i>Help</i>	Displays



## confirm

The `confirm` command pauses a script. Clicking *Yes* or *No* continues the running of the script.

### Syntax

```
confirm "<text>"
```

<b>&lt;text&gt;</b>	Text that appears in a confirmation box.
---------------------	--

## connect lines

The `connect lines` command creates new lines to connect existing line or arc segments.

### Access Using

- Menu Path: *Manufacture – Drafting – Connect Lines*

## Connecting Lines and Arc Segments

Follow these steps to connect lines or arc segments on your design with new lines:

1. Choose *Manufacture – Drafting – Connect Lines* or run the `connect lines` command.  
OR
2. Set *General Edit* application mode and select a line or an arc segment. Right-click and choose *Drafting – Connect Lines*.
3. Select a line or an arc segment.
4. Select another line or an arc segment.  
The potential connection lines are displayed.
5. Click to choose a connect line.  
A connect line is added between the selected elements.
6. Right-click and choose *Next* to continue or *Done* to complete the operation.

## convert\_gerber

The `convert_gerber` batch command converts Gerber data created in Allegro X PCB Editor database. You can convert lines, vias, or both.

You also have the option to convert figures to vias, if via information is not provided in the Gerber file. When this option is chosen, Allegro X PCB Editor matches the stacks of figures at an x,y location to a padstack definition, so the design converts with the same padstacks with which it began.

### Syntax

```
convert_gerber [-f|-v|-e|-q] <input>.brd <output>.brd]
```

<b>-f</b>	Converts only figures
<b>-v</b>	Converts both lines and vias
<b>-e</b>	If an error occurs when converting lines, terminates the command
<b>-q</b>	Displays minimal screen messages during execution

If neither `-f` nor `-v` are used, the default execution is to convert only lines. A log file, `convert_gerber.log`, describes the processing of the command.

## copy

The `copy` command creates duplicates of elements in your design. This command copies elements dynamically and paste them to multiple destination(s) in a single step. The copied objects are stored in a buffer for the duration of the session and can be pasted later to multiple destination(s). You can paste anywhere in design or paste to multiple destination objects, such as pins at a time. To paste to pins with certain net only, you can use *Find by Query* option.

Using this command, you can create step and repeat patterns of multiple copies of elements by pasting them in horizontal/vertical (rectangular) patterns spaced from a user-defined origin or in radial (polar) patterns around a user-defined point. In the *Options* tab, you set the quantity or number of copies to more than one.

You can also mirror elements on the same subclass by choosing *Rectangular* mode in the *Options* tab, and right mouse clicking to use the *Mirror Geometry* command from the pop-up menu that appears. All graphic elements in a design eligible for copying can be mirrored onto the same subclass, except symbols and stand-alone pins. However, if you choose a symbol or stand-alone pin (or if a symbol or stand-alone pin is included in group or window of elements), the `mirror geometry` command is not available on the pop-up menu. For more information on mirroring, refer to *Edit – Mirror* (`mirror` command) and the *Mirror Geometry* option.

Elements ineligible for use with the `copy` command generate a warning and are ignored.

Valid elements to copy are:

- Symbols
- Vias
- Clines
- Lines
- Shapes
- Figures
- Text
- Cline Segs
- Fingers
- Other Segs

Valid destination objects are:

- Pins
- Vias
- Clines
- Fingers

For more details, see *Allegro User Guide: Preparing the Layout*.

If you want to automatically snap the copied elements to the logical object target, see the `create structure` and `replace via structure` commands.

## Related Topics

- [Copying Elements in Rectangular Patterns](#)
- [Copying Elements in Radial Patterns](#)
- [Mirroring Elements on the Same Subclass](#)
- `create structure`
- `replace via structure`

## Copy Command: Options Panel

### Access Using

- Menu Path: *Edit – Copy*
- Toolbar Icon:





When prompted to select the element(s) to copy, set the following in the *Options* tab:

<i>Copy origin</i>	Indicates the point on the element used to calculate distance for rows and columns. If you rotate the element before pasting it, this is also the point about which the element is rotated. The choices are:	
	<i>Symbol</i>	The 0,0 point of the element.
	<i>Body Center</i>	The point at the center of an invisible boundary that the tool draws around the edge of the symbol.
	<i>User Pick</i>	A point you clicked with the mouse.
	<i>Sym Pin #</i>	A pin number you choose. You specify the number in the <i>Symbol pin #</i> field.
Symbol pin #	Specifies a pin number as the element's origin. Appears when you set <i>Copy origin</i> to <i>Sym Pin #</i> .	

When prompted to select destination, set the following in *Options* tab. The options vary depending on whether you set paste mode to Rectangular or Polar.

<i>Paste mode</i>	Specifies the type of pattern you are pasting. These are the choices:	
	<i>Rectangular</i>	Pastes copies in a rectangular grid array.
	<i>Polar</i>	Pastes copies around a user-defined point in angular increments.
Rectangular Options		
<i>Qty X</i>	Defines the number of columns to be created.	
<i>Qty Y</i>	Defines the number of rows to be created.	
<i>Spacing</i>	Indicates row and column (X and Y) spacing. <i>Spacing X</i> indicates the amount of distance in user units between items in a row. <i>Spacing Y</i> indicates the amount of distance in user units between items in a column.	
<i>Order</i>	Indicates the direction in which the tool should paste the copies in each row and column. <i>Order X</i> specifies the direction that rows are pasted. <i>Right</i> is the default. <i>Order Y</i> specifies the direction that columns are pasted. <i>Down</i> is the default.	
Rotation angle	Specifies the angle at which each element is placed if you choose <i>Rotate</i> from the pop-up menu. Enter a number between 0 and 360 or choose a number from the pop-up. The tool allows accuracy for up to three decimal places. <div style="border: 1px solid #f0e68c; padding: 5px; margin-top: 10px;">           ⚠ <i>Rotate</i> works with all elements that you can copy except figures. Figures appear to rotate to any angle, but when you choose a location point, the tool snaps the figure to the nearest 90-degree increment.         </div>	
Polar Options		
<i>Direction</i>	Specifies a direction for pasting the elements: <i>Cww</i> (counter-clockwise) or <i>Cw</i> .(clockwise).	
<i>Copies</i>	Specifies how many copies of the element the tool pastes. The default value is 1.	
<i>Rotation angle</i>	Specifies the incremental angle to be used when pasting the elements. Enter a number between 0 and 360 or choose a number from the pop-up. The tool allows accuracy for up to three decimal places.	
<i>Retain net of vias</i>	Allows vias to either retain their source nets or inherit net of the destination object. When disabled, via inherits the net. This option is disabled by default. When via touches a shape, the via inherits the net of the destination shape. If a via touches multiple nets from different shapes, a random net will be assigned to the via.	

<i>Retain net of shapes</i>	<p>Allows shapes to either retain their source nets or inherit net of the destination object. When enabled, shapes retains source nets. This option is enabled by default. When disabled, the shape inherits the net of the destination object based on the objects hierarchy. For a shape to inherit the net of a</p> <ul style="list-style-type: none"><li>• Pin: The shape must touch center or pin</li><li>• Via: The shape must touch center of via</li><li>• Cline: The shape must touch segment vertex</li><li>• Shape: The shape must touch any part of shape</li></ul> <div> If a shape touches multiple nets of the same hierarchical objects, a random net will be assigned to shape.</div> <div> If there is no net to assign to a shape, a dummy net will be assigned to the shape.</div>
<i>Ignore FIXED objects</i>	<p>Specifies that objects with the FIXED property will not be copied.</p>

### Related Topics

- [Copying Elements in Radial Patterns](#)
- [Mirroring Elements on the Same Subclass](#)

## Copying Elements in Radial Patterns

For details about using the polar copy mode, see *Preparing the Layout* in the user guide.

1. Choose *Edit – Copy*.

Alternately, you can click on the toolbar icon for the *Copy* command or type `copy` in the Command line.

You can set the *Copy origin* property in the *Options* panel.

⚠ The option to set *Copy origin* is not available when you run the *Copy* command from the right-click pop-up options after selecting the component.

2. Select the element to be copied.

The following message is displayed in the command window:

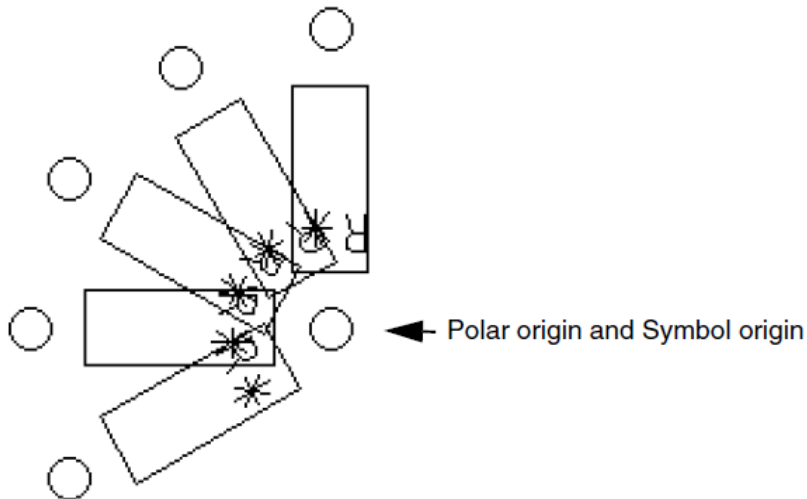
Copied object(s) saved to Paste buffer. Previous Paste buffer content overwritten.

3. Right-choose and choose *Options* or in the *Options* tab, choose *Polar* and complete the other entries.
4. Left choose to specify the polar origin for the copied element.
5. Choose the new location for the element. Each left choose pastes another copy of the element on your design. The tool creates a pattern of copies around the point of origin.
6. Choose *Done* from the pop-up menu.

### Example

In the following figure, the tool generates a radial pattern of copies:

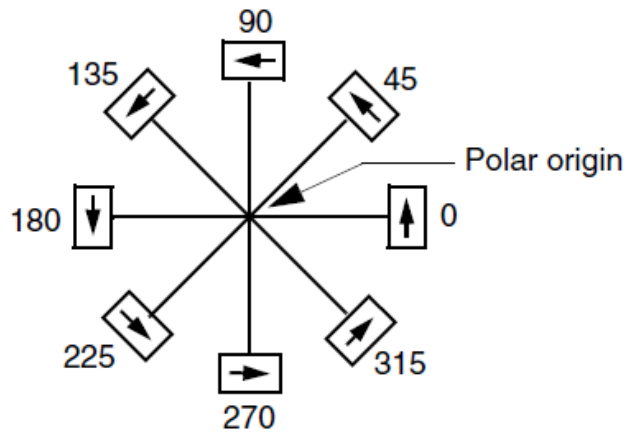
**Figure:** Copied Elements in a Radial Pattern



The following figure uses an angle value of 45 degrees and shows the possible locations for this setting:

**Figure:** Incremental Angles





### Related Topics

- [copy](#)
- [Copy Command: Options Panel](#)

## Copying Elements in Rectangular Patterns

1. Choose *Edit – Copy*.

Alternately, you can click on the toolbar icon for the *Copy* command or type `copy` in the Command line.  
You can set the *Copy origin* property in the *Options* panel.

⚠ The option to set *Copy origin* is not available when you run the *Copy* command from the right-click pop-up options after selecting the component.

2. Select the element to be copied.

The `paste` command invokes automatically and the following message is displayed in the command window:

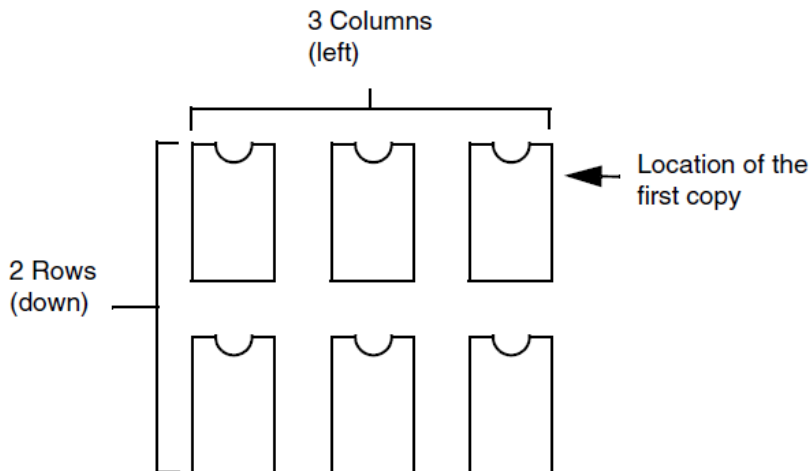
Copied object(s) saved to Paste buffer. Previous Paste buffer content overwritten.

3. Right-click and choose *Options* from the pop-up menu, or in the *Options* tab, choose *Rectangular* and complete the other entries.
4. If you entered a value in the *Angle* field:
  - a. Choose *Rotate* from the pop-up menu.
  - b. Rotate the elements to the angle you need and choose to lock them in that position.
5. In *Find* filter, enable only desired destination objects to paste to.
6. Click in the design canvas to paste either to a click location or use window select (window/polygon/lasso/path), Temp group, or Find by Query to paste to selected pads or clines.
7. Right-click and choose *Done* from the pop-up menu.

### Example

In the following figure, the number of columns and direction is 3, Left and the number of rows and direction is 2, Down.

**Figure:** Copied Elements in a Rectangular Pattern



### Related Topics

- [copy](#)
- [Mirroring Elements on the Same Subclass](#)

## Mirroring Elements on the Same Subclass

1. Hover your cursor over an element. The tool highlights the element and a datatip identifies its name.
2. Right-click and choose *Copy* from the pop-up menu.  
The following message is displayed in the command window:  
  
`Copied object(s) saved to Paste buffer. Previous Paste buffer content overwritten.`
3. Right-click and choose *Options* from the menu, or in the *Options* tab, choose *Rectangular* and complete the other entries.
4. Choose the element(s) to be copied. For a group of elements, window select an area to choose a group.
5. Right-click and choose *Mirror Geometry* from the pop-up menu that appears.
6. Move the cursor away from the original element. The original element remains in place and a mirror image of the original, around the Y-coordinate of the copy origin, moves with the cursor.
7. Move the mirrored copy to its new position, and choose to place it there.
8. Click in the design canvas to paste either to a click location or use window select (window/polygon/lasso/path), Temp group, or Find by Query to paste to selected pads or clines.
9. From the pop-up menu, choose *Done*.

## Related Topics

- [copy](#)
- [Copy Command: Options Panel](#)
- [Copying Elements in Rectangular Patterns](#)


## copy component

The `copy component` command copies a component, creates a new of the same type and adds to the design netlist. You can use this command for finalize the placement without waiting for a netlist change from the schematic.

The reference designator of the new component has the same prefix as the copied component and uses the next available unused number. To assign nets to the new component, use the `net logic` command. These changes cannot be fed back to schematic and require manual updates in the logic design.

By default, this command is disabled. To enable this command set an environment variable `logic_edit_enabled` in User Preferences Editor. When set, this command is available in General Edit and Placement Edit application mode as right-click option.

To run this command, in Find filter, enable *Symbols*. Hover over a component, right-click and choose *Copy component*. The copied component is attached to the cursor. Click to paste the component in the canvas.

 You cannot copy component to a different zone using this command.

It is recommended to not use this command for designs that are front-to-back flow-driven because it adds components to the netlist.

## copy fanout

The `copy fanout` command duplicates a fanout pattern from one component to all other instances of that symbol definition. Duplication occurs on the same subclass on one side of a board only. For symbols on a different subclass, you can create a fanout for one of them using the [create fanout](#) command, and then copy it to the remaining symbols on that subclass.

Copying a fanout automatically replaces any existing fanout on the chosen component unless the FIXED property has been assigned, or the fanout is tied to a different component. A copy occurs even if you have modified a symbol pin's padstack on one instance, as long as you don't move the pin from the chosen component's symbol definition. Duplicated fanouts preserve the padstacks and line widths of the chosen symbol, and DRC occurs after fanout duplication.

Shapes are considered part of the fanout if they connect only to the chosen symbol and are copied along with any associated clines and vias. No copy occurs for shapes tied to more than one symbol.

This command functions in both the noun-verb (pre-selection) mode and verb-noun mode. In the pre-selection use model in the Etch Edit application mode, you choose an element first, then from the pop-up menu (right-click) choose and execute the command. Elements ineligible for use with the command generate a warning and are ignored. Valid element:

- Symbols

### Related Topics

- [Copying a fanout](#)

## Copy Fanout Command: Options Panel

### Access Using

- Menu Path (PCB Editor): *Route – Copy Fanout*
- Menu Path (APD): *Route – Via Structure – Copy Fanout*

<i>Copy Fanout to</i>	
<i>Same device types</i>	Choose to copy to all symbols with identical package names and the same device type.
<i>Same package names</i>	Choose to copy the chosen symbol's fanout to all symbols with identical package names whether or not the device type matches.

## Copying a fanout

Perform these steps to copy a fanout:

1. Choose *Setup – Application Mode – Etch Edit* to access the etchedit application mode.
2. Hover your cursor over a symbol instance. The tool highlights the element and a datatip identifies its name.
3. Right choose and choose *Copy Fanout to* from the pop-up menu and whether to copy to symbols with the same device types or the same package names. The fanout is copied to all other instances of that symbol. The command then exits, and you may choose another symbol instance to copy.

## Related Topics

- [copy fanout](#)

## cputime

The `cputime` command offers electronic timing options within the tool. Command line options offer capability similar to that of a handheld stopwatch. Normally, these are embedded within a script for timing purposes.

This command displays the time used by the program, eliminating the effects of other running processes and the program's IO waits. The IO waits category includes the time spent waiting for disk or video access and the wait for spawned programs to return. In most cases, executing this command offers better repeatability when timing Allegro commands.

### Syntax

`cputime[<option>]`

<b>reset</b>	Returns the clock to zero and starts the clock.
<b>stop</b>	Stops the clock and displays elapsed time.
<b>lap</b>	Reports time elapsed and keeps clock running.
<b>start</b>	Starts the clock without resetting (for example, continues from previous time).
<b>normal</b>	Reports the time in hh:mm:ss:ff format.
<b>second</b>	Reports the time in seconds and milliseconds format.
<b>verbose</b>	Displays database status messages as the command executes.



## create\_devices

The `create_devices` batch command that creates a device file for every device type in the design. The command writes these files to the current directory:

- device files (*<devicename>.txt*)
- log file (`create_devices.log`)
- map file (`devices.map`)

You can store the extracted device files locally or in a library. You can also edit the device files using a text editor, as with any other ASCII file. Run the `viewlog` command to look at the log file.

### Syntax

```
create_devices <design_name>
```

<b>&lt;design_name&gt;</b>	Name of the <code>.brd</code> file from which you want to create device files.
----------------------------	--

## create\_sym

The `create_sym` batch command creates a symbol file from a drawing file. The command writes these files to the current directory:

- symbol file
- `<symbol_name>.log`

You can store the extracted symbol file locally or in a library. Run `viewlog` to look at the log.

 Before creating a package/part symbol, refer to *Defining and Developing Libraries* in the user guide.

## Syntax

```
create_sym <options> <name1> <name2>
```

<b>&lt;options&gt;</b>	Specifies the type of symbol you are creating. Use only one option: <code>-p</code> creates a package symbol ( <code>.psm</code> ) [Default] <code>-m</code> or <code>-b</code> creates a mechanical symbol ( <code>.bsm</code> ) <code>-f</code> or <code>-o</code> creates a format symbol ( <code>.osm</code> ) <code>-s</code> creates an pad shape symbol ( <code>.ssm</code> ) <code>-t</code> creates a thermal flash shape symbol ( <code>.fsm</code> )
<b>&lt;name1&gt;</b>	Indicates the name of a <code>.dra</code> drawing file. The first name in the command is assumed to be the name of the <code>.dra</code> file. You do not have to include the extension.
<b>&lt;name2&gt;</b>	(Optional) Indicates the name of the symbol file. If a second name is not present, the symbol file name is created from the drawing file's name and the appropriate symbol extension.

## create bounding shape

The `create bounding shape` command creates shapes around the boundary of the selected objects on the specified layers. You can create and modify one or more shapes simultaneously on multiple layers. Valid objects are: pins, vias, fingers, and clines. Boundary-based shape generation is useful in high-density and high-speed designs where bond fingers and RF components are widely used.

### Related Topics

- [Creating Shapes Around Boundary of Selected Objects](#)

## Create Bounding Shape Command: Options Panel

### Access Using

- Menu Path: *Shape – Create Bounding Shape*

To create shape select parameters from the *Options* panel.

<i>Object Selection</i>	Display options to select objects.
<i>Source layer</i>	Selects a layer for bounding shape from the drop-down list.
<i>Sync with shape layer</i>	Synchronizes the shape with the specified source pad or segment layer.
<i>Select by net</i>	Lets you select the objects that are on a selected net.
<i>Select subclasses to copy to</i>	Select layers to copy shape.
<i>Reset All</i>	Clears the layers selection from <i>Select subclasses to copy to</i> .
<i>Shape Options</i>	Specify settings to create the bounding shape.
<i>Type</i>	Select the type of shape from the drop-down list. Following shape types are available: <ul style="list-style-type: none"><li>• <i>Dynamic Copper</i></li><li>• <i>Dynamic Crosshatch</i></li><li>• <i>Static Solid</i></li><li>• <i>Static Crosshatch</i></li></ul> You cannot choose dynamic shape type if both conductor and non-conductor layers are enabled to copy shape.
<i>Regenerate shape</i>	Modifies the shape with different parameters. By default, this option is enabled.
<i>Auto assign to net</i>	Assigns the net to the shape that is created on a conductor layer.
<i>Distance from Pad/Cline segment edge</i>	Specifies the distance from the pad or segment edge. By default, no value is set.
<i>Max separation of grouped items</i>	Specifies the maximum distance between sets of objects. By default, no value is set.

## Creating Shapes Around Boundary of Selected Objects

To create shapes around the boundary of selected objects, perform these steps:

1. Choose *Shape – Create Bounding Shape* command.  
Alternately, type `create bounding shape` in the Command line.  
The following message appears in the command window:  
  
Select the objects around which to create a bounding shape on the specified layer(s)
2. Select objects in the *Find* filter.
3. Choose *Source layer* in the *Options* tab.
4. Enable *Select by net* to select the objects that are on the same net. Choose a net from the drop-down menu.
5. Select classes and subclasses on which the shape has to be copied.
6. Specify *Distance from Pad/Cline segment edge*.
7. Optionally, enable *Max separation of grouped items* and specify the distance in drawing units.
8. Drag a shape around the selected objects.
9. Right-click and choose *Done* to complete the command.

## Related Topics

- [create bounding shape](#)

## create coupons

The `create coupons` command displays the *Select and Define Coupon* dialog box, where you choose the test coupon types you want generated for a Allegro X PCB Editor drawing.

For more details on generating test coupons, see *Allegro User Guide: Completing the Design*.

### Related Topics

- [Generating a Coupon](#)
- [Adding Coupons to a Drawing](#)

## Adding Coupons to a Drawing

The coupon generation program creates the chosen coupons as database entities in the drawing. When the program is complete, the first coupon symbol appears attached to the cursor. You must then place the coupons in the drawing.

To place the coupons on the drawing:

1. Choose to place the coupon at the specified location in the board/substrate drawing.  
The *Rotate*, *Mirror*, and *Ops* options are available in the pop-up menu.  
After you place the first coupon, the symbol remains attached to the cursor to let you place another copy of the same symbol in another location.
2. Choose *Next* in the pop-up menu.

The next defined coupon symbol appears attached to the cursor. Position the coupon as defined in step 1.

If no more coupon symbols are defined, *Done* is assumed. No cycling is done.


1. To stop adding coupons to the drawing, choose one of the following options from the pop-up menu:
  - *Cancel* deletes all coupon symbols placed after the last *Next* operation and any coupon symbol entities in the database that are not placed. The program is complete.
  - *Done* checks for any coupon symbol entities in the database that are not placed and deletes them. The program is complete.

## Related Topics

- [create coupons](#)
- [Select and Define Coupon Dialog Box](#)

## Generating a Coupon

1. Run the `create coupons` command.  
Alternately, choose *Manufacture – Create Coupons*.
2. In the Select and Define Coupon dialog box, choose the coupon types and settings you need.
3. Choose *OK* to begin the coupon generation program. The coupons are generated one by one in the same order as they appear in the form.

 If the coupon symbol already exists in the drawing, an error message appears, asking if you want to delete all instances of that symbol.

- *Yes* deletes all existing instances of the same coupon in the layout and recreates it. To regenerate a coupon, all existing instances of the coupon in the drawing must be deleted.
- *No* is equivalent to selecting *Next* in the pop-up menu. The generation of that coupon is skipped.

When the program is complete, the Coupon Generation Log appears.

4. Choose *Close* to close the log.
5. Continue with [Adding Coupons to a Drawing](#).

## Related Topics

- [create coupons](#)
- [Adding Coupons to a Drawing](#)



## Select and Define Coupon Dialog Box

### Access Using

- Menu Path: *Manufacture – Create Coupons*

Use this dialog box to specify the test coupons you want generated by the coupon generation program.

<i>Coupon Type</i>	Indicates the coupon type to be generated. By default, Coupons A, B, and F are chosen.
<i>Default Padstack</i>	Indicates the default padstack for each coupon type. Type a padstack name or choose one from the list.  ⚠ You can type any padstack name listed in the \$PADPATH environment setting.
<i>X Spacing</i>	Indicates the X spacing between padstacks for the coupon type. The value is in design units.
<i>Y Spacing</i>	Indicates the Y spacing between padstacks for the coupon type. The value is in design units.

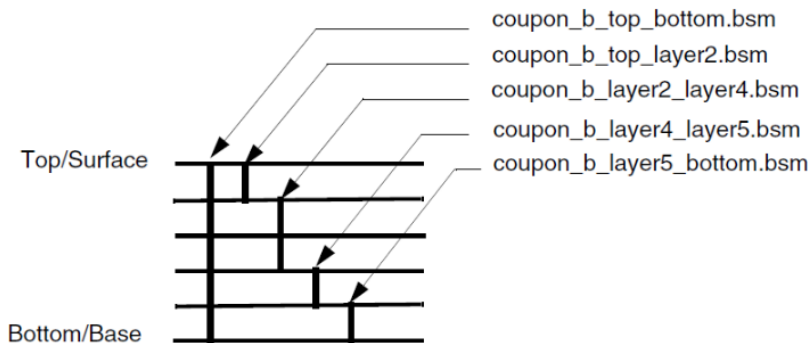
When you run the coupon generator command, the following default settings are shown for each coupon type:

- Coupon A is set to *on*.  
The padstack name, defined with the smallest through drill hole greater than zero, and smallest pad size definition which is greater than the hole size, used as a component pin in the drawing is the default padstack. The default X,Y spacing between padstacks is 100 mils. The symbol name is `coupon_a.bsm`.
- Coupon B is set to *on*.  
The padstack name, defined with the smallest through drill hole greater than zero, and smallest pad size definition which is greater than the hole size, used as a component pin or via in the drawing is the default padstack. The default X,Y spacing between padstacks is 100 mils.

⚠ One additional B coupon must be created for each blind/buried padstack used in the drawing.

The following figure shows example naming conventions for a 6-layer board/substrate.

**Figure:** Sample Naming Conventions for Coupon B



- Coupon C is set to *off*.  
No padstack is specified as none is required. The symbol name is `coupon_c.bsm`.
- Coupon D is set to *off*.  
The padstack name, with a through drill hole greater than zero, and pad size greater than the drill hole size, of the most commonly used component pin or via in the drawing is the default padstack. The symbol name is `coupon_d.bsm`.
- Coupon E is set to *off*.  
The padstack name, with a through drill hole greater than zero, and pad size greater than the drill hole size, of the most commonly used component pin or via in the drawing is the default padstack. The default X spacing is 150 mils and the Y spacing is 300 mils between padstacks. The symbol name is `coupon_e.bsm`.
- Coupon F is set to *on*.  
The padstack name, with a through drill hole greater than zero, and pad size greater than the drill hole size, of the most commonly used component pin or via in the drawing is the default padstack. The default X,Y spacing between padstacks is 100 mils. The symbol name is `coupon_f.bsm`.

- Coupon G is set to *off*.

The padstack name which has a definition of a square pad on the top/surface etch/conductor layer only, no internal pad or solder\_mask definition, and no drill hole definition. The pad size definition for this padstack is to be 50 mils square. This padstack is named *square050top*. The symbol name is `coupon_g.bsm`.

## Related Topics

- [Adding Coupons to a Drawing](#)

## create detail

The `create detail` command creates an enlarged portion of a chosen area in your design. You can locate the detailed view anywhere on your drawing.

For more details on [creating detailed views](#), see *Preparing Manufacturing Data* in the user guide.

### Access Using

- Menu Path (*Layout Mode*): *Manufacture – Dimension/Draft – Create Detail*
- Menu Path (*Symbol Mode*): *Dimension – Create Detail*
- Toolbar Icon:



## Creating a Detailed View

Follow these steps to create a detailed view in your design:

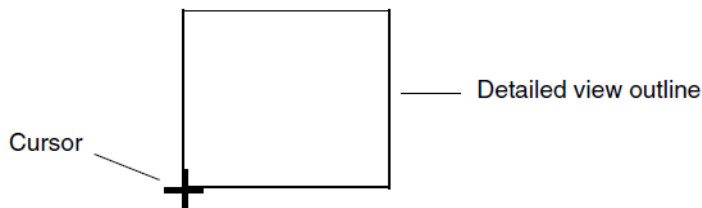
1. Run the `create detail` command.
2. In the *Options* tab, adjust the *Scaling factor*, if necessary.  
The *Options* tab displays the DETAILS subclass, which you leave as is, and a default scaling factor of 2.
3. In the *Find* filter, specify the types of items you want included in the detailed view.
4. Locate the area you want to enlarge:
  - If you want to include all the items in the area, choose two points with mouse clicks to define the selection window (area).
  - If you want to exclude some items in the area, do the following:
    - Choose *Group* from the pop-up menu.
    - Choose each element you want to remove from the selection window.
    - When you finish selecting elements, choose *Complete* from the pop-up menu.

Excluding items you have deselected, the tool copies visible elements in the selection window, clips items (rectangles, lines, arcs, shapes, text, and voids) to the window, and highlights visible elements in the window. Other element types, such as symbols or pins, must lie completely within the selection window to be copied.

The tool enlarges the copy of the selection window to create the detailed view, but only the outline of the detailed view appears when you move the cursor. You cannot see the actual detail until you position the view.

The bottom left corner of the detailed view outline attaches to the cursor, as follows

**Figure 1.1:** Outline of a Detailed View



5. Before positioning the outline in your drawing, change the scaling of the view, rotate it, or mirror it, if necessary.

**To change the scaling:** In the *Options* tab, change the *Scale factor* field.

**To rotate by 90-degree increments:**

- a. From the pop-up menu, choose *Rotate*. The outline rotates 90 degrees counterclockwise, as the cursor's new position shows. For an example, see the following [Rotated Detailed View Outline figure](#).
- b. To continue rotating the view, choose *Rotate* again.

**To mirror:** From the pop-up menu, choose *Mirror*. Notice the new cursor position on the outline. For an example, see the following [Mirrored Detailed View Outline figure](#).

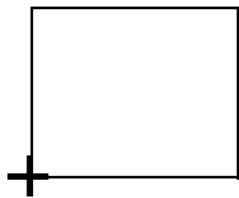
1. Position the detailed view outline within the drawing extents and choose to place it.

The tool displays the detailed view. The outline of the detailed view remains attached to the cursor so you can alter and position additional views in other locations.

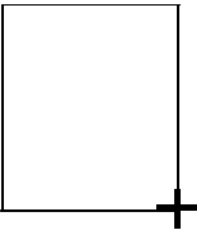
1. If you want to create a different detailed view, choose *Next* from the pop-up menu and repeat steps 2–6.
2. From the pop-up menu, choose *Done*.

## Examples

Rotated Detailed View Outline

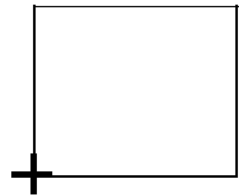


Initial detailed view outline

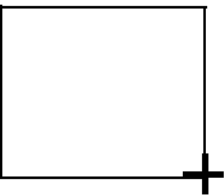


Rotated detailed view outline

Mirrored Detailed View Outline



Initial detailed view outline



Mirrored detailed view outline

## create device

The `create device` command, available in symbol editor mode, allows you to generate a text file of a device of type IC, IO, or DISCRETE. For additional information about preparing device files, see *Defining and Developing Libraries* in the user guide.

### Related Topics

- [Generating Text File Data for a Device](#)

### Create Device Dialog Box

<i>Device Type</i>	Choices are IC, IO, and DISCRETE. Any option is selectable, regardless of the actual device type of the <i>.dra</i> .
--------------------	---

### Access Using

- Menu Path: *File—Create Device (in the Symbol Editor)*

## Generating Text File Data for a Device

1. Run the `create device` command.  
The *Create Device File* dialog box appears.
2. Choose the device type of the *.dra* file.
3. Choose *OK*.  
A text file of the same name as the *.dra* design is written to your current working directory.

### **Sample**

The following is the text file generated for a design drawing titled econ62\_100.dra.

(DEVICE FILE: econ62\_100)

PACKAGE econ62\_100

CLASS DISCRETE

PINCOUNT 62

PINORDER econ62\_100 62 31 61 30 60 29 59 28 58 27,

57 26 56 25 55 24 54 23 53 22,

52 21 51 20 50 19 49 18 48 17,

47 16 46 15 45 14 44 13 43 12,

42 11 41 10 40 9 39 8 38 7,

37 6 36 5 35 4 34 3 33 2,

32 1

PINSWAP econ62\_100 62 31 61 30 60 29 59 28 58 27,

57 26 56 25 55 24 54 23 53 22,

52 21 51 20 50 19 49 18 48 17,

47 16 46 15 45 14 44 13 43 12,

42 11 41 10 40 9 39 8 38 7,

37 6 36 5 35 4 34 3 33 2,

32 1

FUNCTION econ62\_100 econ62\_100 62 31 61 30 60 29 59 28 58 27,

57 26 56 25 55 24 54 23 53 22,

52 21 51 20 50 19 49 18 48 17,

47 16 46 15 45 14 44 13 43 12,

42 11 41 10 40 9 39 8 38 7,

37 6 36 5 35 4 34 3 33 2,

32 1

END

### **Related Topics**

- [create device](#)




## create fanout

The `create fanout` command creates clines and vias and connects them to the chosen pins or symbols. Use this command after component placement but prior to routing to ensure the vias extend to the appropriate routing layer. Creating fanout clines and vias with this command associates them with the symbol instance of the chosen pins or components. Consequently, the fanouts move along with the symbol instances when *Edit – Move* executes.

Generating a fanout automatically replaces any existing fanout on the chosen elements. If pins connect to a different component, fanouts are not modified. No fanout generates for shared vias; multiple vias for voltage pins; pins already having a hole; or pins whose padstack name contains FID, assumed to be fiducials. If multiple via padstacks are associated with a net, then fanout is created with the first via in the list. DRC occurs only after fanout creation. You must verify enforcement of DRC rules.

To create fanouts in designs containing many nets and highly regular patterns of ball grid arrays (BGAs), you can use the `define via structure` command to create via structures, which combine design elements into a single via entity. A via structure may comprise a single via or connect line, a via and a connect line, or multiple vias of different sizes and multiple connect lines of different widths. You can then copy the via structure using the `create fanout` command and use it locally in the current design.

 This command cannot be used for wire bond dies in SiP Layout and APD. For wire bond dies use any of the following methods.

- Run the `wire bond escape` command for wire bond dies and then run the `mark fanout` command.
- Select *Routing Stubs* in the Options pane for wire bond commands or menu selections.  
Refer to the *Routing Stubs* section in the *APD and SiP: Wire Bonding Toolset* chapter of the *Allegro User Guide: Routing the Design* manual for more details

This command functions in both the noun-verb (pre-selection) mode and verb-noun mode. In the pre-selection use model of the Etch Edit application mode, you choose an element first, then from the pop-up menu (right-click) choose and execute the command. Elements ineligible for use with the command generate a warning and are ignored. Valid elements are:

- Pins
- Symbols

### Related Topics

- [Creating a Fanout](#)
- [define via structure](#)
- [wire bond escape](#)

## Create Fanout Command: Options Panel

### Access Using

- Menu Path (PCB Editor): *Route – Create Fanout*
- Menu Path (APD): *Route – Via Structure – Create Fanout*
- Toolbar Icon:



### Options Tab for the create fanout Command

☐ Include Unassigned Pins  
☐ Include All Same Net Pins  

Surface

▼

Start

Base

▼

End

☐ Via Structure  

Symbol

No avail via structure

▼

Rotation

0.00

▼

☐ Mirror-Geo

☒ Via 

Net Default

▼

Via Direction

BGA Quadrant Style

▼

☐ Override Line Width 

0.00

▼

Pin-Via Space

75.00

▼

Min Channel Space

75.00

▼

☐ Curve 

Cw

▼

?

▼

Curve Radius

75.00

▼

When you access the command in the pre-selection use model from the right-mouse button pop-up menu, the Options tab is not available for you to set parameters, instead, set them prior to using the command on the Route tab of the Design Parameter Editor and choose Create Fanout, available by choosing Setup – Design Parameters ([prmed](#) command).

When you run the command in menu-driven editing mode, you may set parameters by right- clicking to display the pop-up menu from which you may choose:

- *Fanout Parameters*, which displays the same *Options* tab information in a dialog box format, useful when the foldable *Options* tab window is pinned
- *Options*

<i>Include Unassigned Pins</i>	Choose to create fanouts for unassigned pins. Disabled by default.
<i>Include All Same Net Pins</i>	Choose to choose one pin and automatically include all other same-net pins on the same component for fanout. Useful for power or ground fanouts, where the via or via structure must be identical for all pins in the same net. Disabled by default.
<i>Start and End Subclasses</i>	Defines the range for the displayed via padstack and via structure symbol names. Defaults to the top-most and bottom-most subclasses on the design.
<i>Via Structure</i>	Choose to add a via structure, which lets you capture complex fanout patterns that may include multiple vias or clines, to symbol or pins you choose. You must have defined the via structure using the <a href="#">create structure</a> command prior to creating a fanout. Disabled by default.
<i>Symbol</i>	Specify the via structure symbol to add from the list of all via-structure symbols in the database that span the specified start and subclasses. Available only if you enabled <i>Via Structure</i> .

## C Commands

### C Commands--create fanout

---

<i>Rotation</i>	Specify the placement angle of the symbols added with <i>Via Structure</i> . Values of 0, 45, 90, 135, 180, 225, 270, and 315 indicate the number of degrees for a counter-clockwise rotation. Available only if you enabled <i>Via Structure</i> . Defaults to 0 degrees, meaning unrotated.
Mirror-Geo	Duplicates the symbol on the current subclass layer that is mirror image of the original, occurring around the Y axis.
<i>Via</i>	Choose to create a fanout using the parameters and via padstack you specify. The via padstacks in the list are those stored in the database that span the specified <i>Start</i> and <i>End</i> subclasses. Use <i>Net Default</i> to ensure a via padstack uses the default net constraints in the Net worksheet in Constraint Manager.
<i>Via Direction</i>	<p>Creates the fanout pattern relative to the pin location.</p> <ul style="list-style-type: none"><li>• <i>Via in Pad</i>: Centers a via at the pin location with no cline between the via and the pad.</li><li>• <i>BGA Quadrant</i>: Default style in which vias are created for each pin in the direction away from the symbol center. Two unused channels remain (one vertical, one horizontal) that pass through the symbol center. BGAs by default fan out using 45 degree angles.</li></ul> <p><i>North, South, East, West</i> specify compass-point directions.</p> <ul style="list-style-type: none"><li>• <i>NE</i> (northeast), <i>NW</i> (northwest), <i>SE</i> (southeast), and <i>SW</i> (southwest) control 45 degree angles.</li><li>• <i>Inward, Outward</i>, and <i>In/Out</i> are useful for SOICs and other non-BGA components.<ul style="list-style-type: none"><li>◦ <i>Inward</i> allows vias underneath the component.</li><li>◦ <i>Outward</i> allows vias outside the component.</li><li>◦ <i>In/Out</i> allows fanouts to alternate between <i>Inward</i> and <i>Outward</i> orientation.</li></ul></li></ul>
<i>Override Line Width</i>	Changes the line width that defaults from the Net worksheet in Constraint Manager. Disabled by default, and the value is zero by default.
<i>Pin-Via Space</i>	Specifies distance between the edge of the pin-pad and the edge of the via-pad. Value defaults from the default constraint set. Zero or negative numbers are valid entries. Choose <i>Centered</i> to locate vias equidistant between pins. The <i>Centered</i> option is valid when <i>Via Direction</i> is <i>BGA Quadrant Style, North, South, East, West, NE, NW, SE, or SW</i> .
<i>Min Channel Space</i>	Maintains a minimum space from vias to other vias with an <i>Inward/Outward</i> via direction. Distance spans via-pad to via-pad on the diagonal. The value defaults from the via-to-via space in the default constraint set. Available only if <i>Via Direction</i> is set to <i>Inward, Outward, or In/Out</i> .
<i>Curve</i>	Creates a fanout cline with two segments and an arc either clockwise (cw) or counter-clockwise (ccw). Available only if <i>Via Direction</i> is <i>BGA Quadrant Style, NE, NW, SE, or SW</i> . Use this option to alleviate thermal stress that may occur when repairs necessitate reheating.
Curve Radius	Specifies the arc radius when you use the <i>Curve</i> option.
?	Visually depicts the effects of the <i>Curve</i> option.

## Related Topics

- [prmed](#)
- [create structure](#)

## Creating a Fanout

To create a fanout, perform the following steps:

1. Choose *Setup – Application Mode – Etch Edit* to access the etchedit application mode.
2. Choose *Setup – Design Parameters* to access the *Route* tab in the Design Parameter Editor.
3. Choose *Create Fanout* from the *Commands* section and then choose *Create Fanout Parameters* to display the Create Fanout Parameters dialog box.
4. Set all desired parameters.
5. Choose *OK* to save the parameters.
6. Hover your cursor over pins or symbols. The tool highlights the elements and a datatip identifies their name.
7. Right choose and choose *Create Fanout* from the pop-up menu to automatically launch the command and create the fanouts using the parameters you specified.  
The command then exits, and you may choose other pins or symbols from which to create another fanout.

## Related Topics

- [create fanout](#)

## create module

The `create module` command creates a module—that is, a design element that is made up of various physical entities. You save the module in an `.mdd` file and can use it in different designs.

For more details on creating modules, see the *Allegro User Guide: Placing the Elements*.

### Access Using

- Menu Path: *Tools – Create Module*


## Creating a Module

1. Choose *Tools—Create Module*.  
Alternately, type `create module` in the Command line.
2. Choose multiple design elements:
  - Choose *Temp Group* from the pop-up menu. Choose on each item in the temporary group or drag the cursor to choose the physical entities that comprise the module definition that are next to each other. Then choose *Complete* from the pop-up menu.
  - Drag the cursor over a group of pins to choose them.
3. Choose the location for the module definition by clicking on that point. This origin is used when you place the module.
4. In the *Save As* dialog box, enter a file name and save the file to a location of your choice.

The tool creates the `mod` file for the module.

## create net

The `create net` command creates a new net from chosen pins in your design.

 When you backannotate this design in Allegro Design Entry HDL L, the logic is not updated.

### Access Using

- Menu Path: *Logic – Create Net*

## Creating a New Net

1. Run the `create net` command.  
Alternately, choose *Logic – Create Net*.
2. In the dialog box that appears, enter the name of the net you are creating and choose OK.
3. If you want to reassign previously assigned pins to the new net, enable the *Re-assign pin allowed* setting on the *Options* tab.

✔ You can select the *Propagate to connected items* to easily select a branch and assign all items in the branch to same branch as selected item(s).

4. Choose a pin for the new net.  
The tool highlights the pin and adds it to the net. If the chosen pin is currently assigned and pin reassignment is allowed (as indicated in the Options tab), the layout editor remove the pin from the old net and adds it to the new net.

⚠ If the reassigned pin had existing connections, DRC errors may occur. If you choose a currently assigned pin, but pin reassignment is not allowed, you need to choose a different pin.

5. Assign more pins to the net.
6. From the pop-up menu, choose *Done*.



## create nets

The `create nets` command lets you create and modify a text file containing a list of nets. This saves you the trouble of having to type in a list of nets using a text editor.

### Related Topics

- [Creating a List of Nets](#)
- [Editing a List of Nets](#)

## Create List of Nets Dialog Box

### Access Using

- Menu Path: *Logic – Create List of Nets*

Use this dialog box to create or edit a list of nets.

<i>List file name</i>	Displays the path of the file that has been opened, the default name of the file to be created, or a pathname you enter.
<i>Open</i>	Opens a .lst file.
<i>Save, SaveAs</i>	Saves a .lst file.
<i>Net Filter</i>	Displays nets according to the Net Filter.
<i>Available Nets</i>	Displays the names of the available nets.
<i>Selected Nets</i>	Displays the names of the chosen nets.
<i>All- &gt; , &lt;- All</i>	Moves all the nets from one list box to another.

### Related Topics

- [Editing a List of Nets](#)

## Creating a List of Nets

If you want to create a list of nets, follow these steps:

1. Run `create nets` from the console window prompt.  
Alternately, choose *Logic – Create List of Net*.  
The *Create List of Nets* dialog box appears.

Leave the Net Filter set to `*` or use wildcards to list a subset of nets.

1. In the Available Nets list box, choose the nets that you want included in the text file or choose *All* -> to choose all available nets for inclusion in the text file.
2. In the List file name field, enter a file name for the netlist.
3. Choose *Save*.
4. To save to a different directory, choose *Save As* and use the File browser.
5. Choose *Close* to close the dialog box.

## Related Topics

- [create nets](#)

## Editing a List of Nets

Perform the following steps to edit a list of nets:

1. Run `create_nets` from the console window prompt.  
Alternately, choose *Logic – Create List of Net*.  
The *Create List of Nets* dialog box appears.
2. Choose *Open*.
3. From the File browser that appears, choose and open a `.lst` file.
4. Leave the Net Filter set to `*` or use wildcards to list a subset of nets.
5. Use the Available Nets and Selected Nets list boxes to add more nets to, or remove nets from, the list of nets.
6. Choose the nets that you want included in or removed from the text file or choose *All* to include or remove all available nets.
7. Choose *Save*.
8. To save to a different directory, choose *Save As* and use the File browser.
9. Choose *Close*.

## Related Topics

- [create nets](#)
- [Create List of Nets Dialog Box](#)

## create plot

The `create plot` command creates intermediate plot (IPF) and control files from a current, active design. These files are required for plotting the displayed area of your layout. You can create IPF and control files on a UNIX workstation or a Windows PC, but you must run them on a UNIX workstation to print a plot.

For more details on plotting, see *Preparing Manufacturing Data* in the user guide.

### Access Using

- Menu Path: *File – Export – IPF*

### Related Topics

- [Creating Plot Files from an Active Design](#)
- [Creating Plot Files for Negative Plane Layers](#)

## Creating Plot Files for Negative Plane Layers

To differentiate between thermal relief and antipads on a negative plane layer plot:

1. Choose *Setup – Design Parameters* ([prmed](#) command) to display the Design Parameter Editor.
2. Choose the *Display* tab and in the *Enhanced Display Modes* section:
  - Enable Filled pads (in Windows) or Filled pads and cline endcaps (in UNIX).
  - Enable Thermal pads .
3. Follow the steps in [Creating Plot Files from an Active Design](#).

### Related Topics

- [create plot](#)
- [Prerequisites for Creating Control Files \(UNIX Only\)](#)
- [Creating Plot Files from an Active Design](#)
- [prmed](#)

## Creating Plot Files from an Active Design

1. Adjust visibility and color priorities using the `color` command, if necessary.
2. Run the `plot setup` command to display the Plot Setup dialog box.
3. In the IPF setup section, set the parameters of the IPF file. Choose *Vectorize text* and specify a line width in the *width* field.
4. Display the part of the drawing you want to output to the IPF file by using the zoom commands as needed.
5. Run the `create plot` command.  
Alternately, choose *File – Export – IPF*.
6. In the Create Plot dialog box, enter the name of the IPF file that you are creating, if it is different from the name of the `.brd` or `.mcm` file.
7. Choose OK .  
The tool creates the IPF and control files, `<filename.plt>` and `<filename.ctl>`.

## Related Topics

- [create plot](#)
- [color](#)

## Prerequisites for Creating Control Files (UNIX Only)

The control file reads the Plot Preference dialog box and assigns a pen number to each color. To use a different color-to-pen-to-priority correspondence than currently set, you must edit the control file before creating the IPF files. (Color-to-priority is dependent on settings that you make in the Layer Priority dialog box.)

To set pen colors for plotting, see [Plotting Your Design on UNIX](#). You can cancel out of the Plot dialog box rather than producing a plot at this point.

## Related Topics

- [Creating Plot Files for Negative Plane Layers](#)
- [Plotting Your Design on UNIX](#)



## create short

The `create short` command places blind vias to create the shorting scheme (defined either in the netlist or interactively using the `define shorting scheme` command) at any of the following locations:

- Package pin
- BGA (ball grid array) die pin
- Bond pad
- Via
- End of trace

 The command is available only with Allegro X Advanced Package Designer (APD).


### Related Topics

- [Creating a Short](#)
- [define shorting scheme](#)

## Create Short Command: Options Panel

### Access Using

- Menu Path: *Route – Create Short (APD)*

<i>Via name</i>	Choose the padstack to serve as the template for the shorting vias created between power and ground planes.
<i>Do not trim via</i>	When chosen, adds an unmodified via; when deselected, trims the via according to the shorting scheme. The trimmed, derived padstack is saved as a blind via. The default is off.
<i>Offset on bond pad: X,Y</i>	Lets you place the shorting via in a position other than the center/origin of the pad. This is useful for bondfingers where the via is not permitted in the same area as that of the bond wire attachment point. The default is 0,0. The tool determines the exact location based on the bond pad origin, offset, and pad rotation.
<i>Perform check only</i>	<div>Choose to verify that the shorts can legally be placed in the defined locations, without actually changing the design database.  To place the vias, you must turn this option off.</div>


## Creating a Short

Follow these steps to create a short:

1. Run the `create short` command.  
Alternately, choose *Route – Create Short*.
2. Choose the items for which you want to create shorts in the *Options* panel.  
The tool modifies the blind via from the generic padstack to match the layers in the shorting scheme. The tool also attaches the `SHORTING_SCHEME` property to all the blind vias created.

For standalone via locations (where traces are not connected to the via), the tool replaces an existing blind via with a new one, unless the original via has the same pad specifications for the layer stackup.

For locations at the end of a trace (which is connected to a pin or via with the `SHORTING_SCHEME` property attached), the tool adds and displays the blind via at the end of each trace.

 If you are only checking the placement of the blind vias (*Perform check only*), the tool immediately displays the Create Short Report in a window. This report is also stored in a file called `cr_short.log`.

1. If you performed a check in the previous step:
  - a. Close the Create Short Report window.
  - b. In the *Options* panel, deselect *Perform check only*.
  - c. Choose items that were identified as legal for which you can create shorts.
2. From the pop-up menu, choose *Done*.

## Related Topics

- [create short](#)

## create structure

The `create structure` command lets you create structures by combining vias, clines, and shapes into a single reusable design elements. The structure symbols are saved as XML-formatted text files. You can reuse the structure files from your current design or from a library of similar files. Structures provide a fast way of adding, refreshing, and replacing a piece of etch in various designs, for example, antennas, coils, HDI structures, and RF structures.


### Types of Structures

- **Standard Via Structure:** You can combine patterns of vias and connect lines (clines) to create a standard via structure symbol. Chosen vias and clines must all be connected to each other and belong to the same net. They may be connected to only one pin, whose location becomes the symbol origin. Duplicate symbol names are not created. A via structure must start and end on the subclasses specified in the `create fanout` command's *Start* and *End* subclasses fields when used in creating fanouts.
- **HighSpeed Via Structure:** High-speed via structures combine vias, connect lines (clines) or traces, static shapes without voids, and route keepouts. High-speed via structure can comprise of differential pairs with return path vias and route keepouts for custom voiding. Standard via structures are defined only for single net items but high-speed via structures can be defined for multiple net items.

 The HighSpeed Via Structure is available only when High-Speed option is enabled in Product Choices dialog box.

Via structures provide an efficient way to create fanouts, particularly in designs containing large numbers of nets and highly regular patterns of pins and ball grid arrays (BGAs). Once created, you can apply via structures to pins for design reuse using the `create_fanout` command, then copy the fanout containing the structure using the `copy_fanout` command.

- **L Comp Structure:** The inductive compensation loop structures (L Comp) structures are created by combining connect lines and route keepouts. The L Comp structure is 3/4 turn circular loop structures added to the end of traces before they are connected to socket pins of the DIMM. These structures are used to balance out the effect of socket capacitance to gain higher performance and also required plane voiding in surrounding ground or power supply reference planes.

 The L Comp structure is available only when High-Speed product option is enabled.

### Related Topics

- [Creating a Structure](#)
- [Defining a Standard Via Structure](#)
- [Defining a High-Speed Via Structure](#)
- [Generating L Comp Structure](#)

## Create Structure Dialog Box

### Access Using

- Menu Path: *Route – Structures – Create*


This dialog box provides options to create via structures and L comp structures. Refer to Description tab of the dialog box to understand how to create various structures.

<i>Select Structure to Create</i>	Select type of structure to create	
	<i>Description</i>	Explains structure definition with visual examples and instructions to create the selected structure
	<i>Define/Generate</i>	Provide options to create and export the selected structure.

### Define/Generate High Speed Via Structure

<i>New Via Structure</i>	<i>Name</i>	Lets you enter a unique name for a new via structure, or accept the default identifier displayed in the name field. The default name is HVS_1.
	<i>Auto Export eXML</i>	Lets you save a defined via structure to an eXML-formatted text file, which you can then reuse in the current or other designs. It is recommended to keep this option enabled.
	<i>Existing Via Structure</i>	
<i>Options</i>	<i>Names</i>	Displays the names of the via structures defined in your design.
	<i>Layer stackup for selected via structure</i>	Lists the layers on which the via structure resides. This is for informational purposes only.
	<i>Cut Clines by Rectangle</i>	If enabled, cut clines and segments inside the drawn rectangle for easy re-use in design.
	<i>Snap origin pick to structure routing element</i>	If enabled, this option allows you to select an item on the routing path of a high-speed structure as the reference origin point for the structure. By default, the option is enabled.  When creating a structure of only route keepout objects, or one designed to be placed relative to a neighboring object, disable this option.
	<i>Preserve Properties</i>	If enabled, preserve properties attached to the selected objects. <ul style="list-style-type: none"> <li>Vias</li> <li>Clines</li> <li>Static Shapes</li> <li>Route Keepouts</li> </ul> By default, the option is disabled.
<i>Description</i>	Hover the cursor over a UI element to display its description.	
<i>OK</i>	Closes the dialog box and commits changes to the database.	
<i>Cancel</i>	Exits the command without saving any changes.	

### Define/Generate L Comp Structure

<i>Padstack</i>	Enable to interactively select a padstack in the design canvas for creating L Comp structures. This option is enabled by default. <div>  You can only select padstack with circular geometry. </div>
-----------------	---

## C Commands

### C Commands--create structure

<b>Select</b>	Click to choose a padstack from the canvas using LMB click. <i>Find</i> filter changes and enables <i>Pins</i> and <i>Vias</i> only. By default, only <i>Pins</i> are selected in the filter.	
<b>Pad diameter</b>	<p>Alternatively, enable to specify pad diameter value instead of selecting padstack in canvas.</p> <p>⚠ When a padstack is selected the <i>Pad diameter</i> displays its diameter on the selected trace layer.</p>	
<b>Trace layer</b>	Choose the trace layer for L Comp structure from the pull down menu that displays list of all the conductor and the plane layers. By default, no trace layer is selected.	
<b>Trace width (WL)</b>	Specifies width of the trace. The default value is set to 3.5 mils.	
<b>Pad to trace/keepout gap (Lpad_gap)</b>	Specifies the gap between the pad and the trace or route keepout. The default value is set to 4 mils.	
<b>Trace to keepout gap (LVss_gap)</b>	Specifies the gap between the trace and the route keepout. The default value is set to 4 mils.	
<b>Loop angle</b>	Defines the angle between the start and the end of the loop structure. By default, the loop angle is set to 270 degrees. Valid values for the loop angle lies between 220 to 320 degrees.	
<b>Ground Void Shape</b>	Choose to select <i>Circular</i> or <i>Rectangular</i> ground void shape. By default, this option is set to <i>Rectangular</i> . Use this option to specify shape of route keepouts created for the selected layers.	
<b>Rectangular Void Height (Rect_H)</b>	Specifies a positive value for the height of rectangular voids. If pad diameter is specified, the height must be equal to or greater than the pad diameter. By default, this value is set to 44 mils. This option is not available when Ground Void Shape is set to Circular.	
<b>Void Adjacent Planes</b>	<p>Displays list of all the conductor and the plane layers defined in the stackup of the design. You can either select all or few layers to create structure with route keepout for voiding adjacent planes.</p> <p>⚠ The L Comp structures can be created without keepout.</p>	
<b>New L Comp Structure</b>		
	<b>Name</b>	Specifies the name of the L Comp structure. The default name is LCOMP_1, if LCOMP_1 already exists, it displays LCOMP_2, so on and so forth. The structure files are, by default, saved in the current working directory. However, the output directory can be changed by using browse button.
	<b>With trace to next dimm</b>	Specifies a suffix that is appended to the name of the L Comp structure to identify structure with trace to next dimm stub. The default is *_STUB.
	<b>Without trace to next dimm</b>	Specifies a suffix that is appended to the name of the L Comp structure to identify structure without trace to next dimm stub. The default is *_NOSTUB.
	<b>Generate</b>	<p>Generates L Comp structures for the selected circular padstack based on the parameters defined in the dialog box. Two structure files are created in eXML format and loaded in the current design. The button is disabled if following options have invalid values:</p> <ul style="list-style-type: none"> <li>• Padstack name (Pad with circular geometry must exist on the selected trace layer)</li> <li>• Pad diameter</li> <li>• Trace layer</li> <li>• Trace width</li> <li>• LPad_gap value</li> <li>• LVss_gap value</li> <li>• Via structure name</li> <li>• With trace to next dimm name</li> <li>• Without trace to next dimm name</li> </ul>

## C Commands

### C Commands--create structure

---

<i>Place L Comp Structure after generation (Structures – Place)</i>	Invokes <i>Place Structure</i> dialog box. The *_STUB that was just created is automatically selected and attached to the cursor for placement in the design. This option is enabled by default.
<i>Reset</i>	Restores the default settings in the dialog box.
<i>Close</i>	Closes the dialog box without saving any changes.

### Define/Generate Standard Via Structure

<i>New Via Structure</i>		
	<i>Name</i>	Lets you enter a unique name for a new via structure, or accept the default identifier displayed in the name field. The default name is SVS_1.
	<i>Auto Export XML</i>	Lets you save a defined via structure to an XML-formatted text file, which you can then reuse in the current or other designs. It is recommended to keep this option enabled.
<i>Existing Via Structure</i>		
	<i>Names</i>	Displays the names of the via structures defined in your design.
	<i>Layer stackup for selected via structure</i>	Lists the layers on which the via structure resides. This is for informational purposes only.
<i>Options</i>		
	<i>Cut Clines by Rectangle</i>	If enabled, cut clines and segments inside the drawn rectangle for easy re-use in design.
	<i>Preserve Properties</i>	If enabled, preserve properties attached to vias and clines. By default, the option is disabled.
<i>OK</i>	Closes the dialog box and commits changes to the database.	
<i>Cancel</i>	Exits the command without saving any changes.	

### Related Topics

- [Defining a Standard Via Structure](#)
- [Defining a High-Speed Via Structure](#)
- [Generating L Comp Structure](#)

## Creating a Structure

You can create a structure in various ways. In some instances, you can choose a combination of vias and connect lines that already exist in a design. In other cases, you want to create new elements or add them from other designs, using one of the following methods.

- `add connect`

You can create the elements of new structures with the standard editing command `add connect`.

- `clpcopy` and `clppaste`

You can copy elements to be used in structures (or structures themselves) from other designs using [clpcopy](#), then add them to your current design with [clppaste](#).

### ***Using Auto Export XML***

You can define a new structure definition from vias and clines, and save them to an XML-formatted text file using the *Auto Export XML* option in the Create Structure dialog box. You also can save structures using the *Export Selected Structure* option in the Place Structure dialog box. You can then reuse these structure files from your current database or from a library of similar files.

With design elements defined as a structure in place (or with a structure brought in from another design), you are ready to run the place structure command.

## Related Topics

- [create structure](#)
- [Defining a High-Speed Via Structure](#)
- [Generating L Comp Structure](#)



## Defining a High-Speed Via Structure

To define a high-speed via structure, perform the following steps:

1. Choose *Setup – Application Mode – Etch Edit* to access the Etch edit application mode.
2. Choose *Route – Structures – Create*.  
Alternately, type `create structure` in the Command line.  
The Create Structure dialog box appears.
3. Select *HighSpeed Via Structure* option and open *Define/Generate* tab.
4. In the *Define/Generate* tab, set the following:
  - a. Specify the name of a new via structure, or accept the default identifier displayed in the name field.  
Every via structure must have a unique name. The default identifiers for defined via structures are HVS\_1, HVS\_2, HVS\_3... and so on.
  - b. Enable *Auto Export XML* to save the via structure you are defining as a text file.  
You save the elements you define as a via structure to an XML-formatted text file, which you can then reuse or import into other designs.

⚠ The padstack files for vias defined in a via structure must exist in your \$PAD\_PATH in order to be written out as part of the file.

- c. Enable *Preserve Properties* checkbox to maintain the properties attached to the selected vias, clines, and shapes to the via structure.
  - d. Enable *Cut Clines by Rectangle* checkbox to include clines and segments enclosed within the drawn rectangle.
5. Ensure visibility of clines, vias, and shapes are set on all the layers.  
Only visible objects are included in via structure.
6. In the *Find* filter, enable the element types from which you want to choose. Valid object types are *Vias* and *Clines* (connect lines), and *Shapes*.
7. Choose the elements of the via structure by selecting vias, clines, static shapes, and route keepouts. To select multiple objects, use window drag or right-click and choose *Temp Group* from the pop-up menu.

⚠ Ensure that at least one via or cline is selected.

⚠ Voids in static shapes are not supported and will be removed.

8. If *Cut by Rectangle* option is enabled, draw a rectangle using mouse clicks.
9. Select a center of a via or an end of a cline segment as the origin of the via structure.  
When you place via structures, the origin is used to snap to pads or clines.
10. If the defined via structure has multiple connections, a message appears asking if return path connects need to be defined.
  - a. If there are return path vias, click *Yes* to identify the return path vias.
  - b. Select a via for each return path connection.
  - c. Right-click and choose *Complete Return Path*.
11. If the Via Structure Start Layer Selection form appears, select the starting layer for the via structure and click *OK*.  
A file browser opens to save the definition.
12. Click *Save* to save the eXML file with default name and to the default location, if *Auto Export* is selected. You can change the name and location.  
The *Existing Via Structures* shows the structure name.
13. Click *OK* to save the via structures to the database.

## Related Topics

- [create structure](#)
- [Create Structure Dialog Box](#)
- [Creating a Structure](#)


## Defining a Standard Via Structure

Follow these steps to define a standard via structure:

1. Choose *Setup – Application Mode – Etch Edit* to access the Etch edit application mode.
2. Choose *Route – Structures – Create*.  
Alternately, type `create structure` in the Command line.  
The *Create Structure* dialog box appears.
3. Select *Standard Via Structure* option and open *Define/Generate* tab.
4. In the *Define/Generate* tab, set the following:
  - a. Specify the name of a new via structure, or accept the default identifier displayed in the name field.  
Every via structure must have a unique name. The default identifiers for defined via structures are SVS\_1, SVS\_2, SVS\_3... and so on.
  - b. Enable *Auto Export XML* to save the via structure you are defining as a text file.  
You save the elements you define as a via structure to an XML-formatted text file, which you can then reuse or import into other designs.

 The padstack files for vias defined in a via structure must exist in your \$PAD\_PATH in order to be written out as part of the file.

- c. Enable *Preserve Properties* checkbox to maintain the properties attached to the selected vias and clines to the via structure.
  - d. Enable *Cut Clines by Rectangle* checkbox to include clines and segments enclosed within the drawn rectangle.
5. Ensure visibility of clines and vias are set on all the layers.  
Only visible objects are included in via structure.
6. In the *Find* filter, enable the element types from which you want to choose. Valid object types are *Vias* and *Clines* (connect lines).
7. Choose the elements of the via structure by selecting vias and clines that are connected to each other. To select multiple objects, use window drag or right-click and choose *Temp Group* from the pop-up menu.

 Ensure that at least one via or cline is selected.

8. If *Cut by Rectangle* option is enabled, draw a rectangle using mouse clicks.
9. Click on the element that will be the origination point of the via structure (the point where the via structure attaches to another design element when added to your design). Select center of via or dangling cline end point as origin.
10. Select a start layer in the Via Structure Start Layer Selection, if displayed.
11. If *Auto Export* button is checked, you are prompted to save the name of the via structure text file you are creating. By default, this is the name of the new via structure, followed by a `.xml` extension, located in your current working directory.  
The name of the new structure appears in the *Existing Structures* list in the dialog box.
12. Click *OK* to add the newly created via structure to your design.

## Related Topics

- [create structure](#)
- [Create Structure Dialog Box](#)
- [Generating L Comp Structure](#)

## Generating L Comp Structure

To generate an L Comp structure, follow these steps:

1. Choose *Setup – Application Mode – Etch Edit* to access the Etch edit application mode.
2. Choose *Route – Structures – Create*.  
Alternately, type `create structure` in the Command line.  
The Create Structure dialog box appears.
3. Select *L Comp* option and open *Define/Generate* tab.
4. In the *Define/Generate* tab, set the following:
  - a. With *Padstack* radio button enabled, click *Select* to choose a padstack from the canvas.
  - b. In the *Find* filter, check the element types from which you want to choose. Valid object types are *Pins* and *Vias*.  
A message is also displayed in the command window.  
  
Select Pin in canvas.
  - c. Click to select a pin (or via) to create L Comp structure geometry.
  - d. Alternatively, select *Pad diameter* and specify a value.
  - e. Select a trace layer for the L Comp structure.
  - f. Specify the value of the following parameters:
    - Trace width
    - Pad to trace/keepout gap
    - Trace to keepout gap
    - Loop angle
  - g. Optionally, choose *Ground Void Shape* to *Circular*.
  - h. If *Ground Void Shape* is set to *Rectangular*, specify the value for *Rectangular Void Height*. It should be greater than the pad diameter.
5. In the *Void Adjacent Planes* section, select layers to create route keepouts for plane voiding.
6. In the *New L Comp Structure* section, enter the name of a new L Comp structure, or accept the default identifier displayed in the name field.

Every L Comp structure will be created in a pair; with stub and without stub. The default identifiers for defined via structures are LCOMP\_1, LCOMP\_2, LCOMP\_3... and so on.

1. Ensure that the *Place L Comp Structure after generation* option is enabled.
2. Click *Generate*, to create the L comp structure.  
The L comp structures are saved to eXML-formatted text files, which you can reuse into other designs.  
The *Create Structure* form closes and *Place Structure* dialog box automatically appears. The structure is selected in the list and attached to the cursor.
3. Optionally, change the selection to LCOMP1\_NOSTUB.
4. Enable element types in the *Find* filter. By default, Pins, vias, and fingers are checked as valid object types.
5. Right-click and choose *Rotate* or *Mirror Geometry* if required.
6. Use LMB click to select an element in the design.  
The L Comp Structure is placed on the selected object.
7. Right-click and choose *Done* to exit the command.

## Related Topics

- [create structure](#)
- [Create Structure Dialog Box](#)
- [Creating a Structure](#)
- [Defining a Standard Via Structure](#)

## create symbol

The `create symbol` command creates a symbol definition from the contents of the active symbol drawing. This command compiles a `.dra` file into the appropriate binary file, based on the drawing parameters of the drawing. You can create one of the following symbols:

Symbol Type	Description	Binary File Extension
Package/Part	Creates a new component symbol such as an IC or a discrete.	<code>.psm</code>
Mechanical	Creates a drawing symbol such as a card edge connector or a board/design outline.	<code>.bsm</code>
Format	Creates a drawing symbol such as a legend or a company logo.	<code>.osm</code>
Shape	Creates a drawing symbol such as a special shape for a padstack.	<code>.ssm</code>
Flash	Creates a flash symbol such as a thermal pad for Rastar formats	<code>.fsm</code>

### Access Using

- Menu Path: *File – Create Symbol*

### Related Topics

- [Defining a Mechanical Symbol](#)
- [Creating a Board Outline](#)
- [Defining a Format Symbol](#)
- [Defining a Flash Symbol](#)
- [Defining Custom Pad Shapes](#)

## Creating a Board Outline

Before creating a board outline, refer to guidelines for creating mechanical symbols in *Defining and Developing Libraries* in the user guide.

Follow these steps to create a board outline:

1. Run the `new` command.

The *New Drawing* dialog box appears.

1. Enter a file name in the *Drawing Name* field.

The symbol file name can consist of any alphanumeric characters, and can include the - (dash) and \_ (underscore) characters.

1. Choose *Mechanical symbol* from the *Drawing Type* list box, and choose *OK*.

The layout editor saves this file with a `.dra` extension to a symbol library.

1. If necessary, choose *Setup – Design Parameters* (`prmed` command) and the *Design* tab of the Design Parameter Editor to change settings for the symbol.
2. Choose *Add – Line* (`add line` command).
  - a. In the *Options* panel, specify the values you need.
  - b. Draw the board by clicking in the design window to choose the corners of the outline.
  - c. After closing the outline, choose *Done* from the pop-up menu.
3. Add tooling corners to the layout outline using the `add line` command with a thick *Line width*.
4. Save the file by choosing *File – Save As* (`save_as` command). Choose the directory where you are saving the symbol drawing file, change the file name if necessary, and choose *Save*.
5. Choose *File – Create Symbol* (`create_symbol` command) to save the drawing as a mechanical symbol. Enter the directory where you are saving the board outline, change the file name if necessary, and choose *Save*.

## Related Topics

- [create symbol](#)
- [Creating a Package Symbol Manually](#)
- [Defining a Flash Symbol](#)
- [Defining Custom Pad Shapes](#)
- [new](#)
- [prmed](#)
- [add line](#)
- [save\\_as](#)

## Creating a Package Symbol Manually

Before creating a package/part symbol, refer to *Developing and Defining Libraries* in the user guide.

1. Choose *File – New*.

The New Drawing dialog box appears.

1. Enter a file name in the *Drawing Name* field.

The symbol file name can consist of any alphanumeric characters, and can include the - (dash) and \_ (underscore) characters.

1. Choose *Package/Part symbol* from the *Drawing Type* list box, and choose *OK*.

The layout editor saves this file with a `.dra` extension to a symbol library.

1. If necessary, choose *Setup – Design Parameters* (`prmed` command) and the *Design* tab of the Design Parameter Editor to change settings for the symbol.
2. Continue the package/part symbol creation process by doing the following:

Add pins to the package/part symbol, described in [Adding Pins](#).

- Add a package/part outline by choosing *Add – Line* (`add line` command). Set *Active Class and Subclass* to *Package Geometry/Part Geometry* and *Assembly\_top*.
- Label the package/part using these menu items or commands:
  - *Layout – Labels – Device* (`label device` command)
  - *Layout – Labels – Part Number* (`label part` command)
  - *Layout – Labels – Refdes* (`label refdes` command)
  - *Layout – Labels – Tolerance* (`label tolerance` command)
  - *Layout – Labels – Value* (`label value` command)

⚠ Labels (the `label <label type>` commands) move with the part they label, differing from *Pin Numbers* (the `add pin` command), which only identify pins, and free-form annotation (the `add text` command), which you use to title the design and provide notes about the package/part.

- Define package/part symbol areas (route keepout, via keepout, package/part boundary, or symbol height text), described in [Adding Areas](#) and *Defining and Developing Libraries* in the user guide.

After you complete each package/part symbol element, choose *Done* from the pop-up menu.

1. Save the file by choosing *File – Save As*. Choose the directory where you are saving the symbol drawing file, change the file name if necessary, and choose *Save*.
2. Choose *File – Create Symbol* (`create symbol` command) to save the drawing as a package/part symbol. Enter the directory where you are saving the symbol, change the file name if necessary, and choose *Save*.

The layout editor checks the symbol to make sure that it contains:

- At least one pin
- A reference designator
- A component outline

## Related Topics

- [Creating a Board Outline](#)
- [Defining a Format Symbol](#)
- [Defining a Flash Symbol](#)
- [Defining Custom Pad Shapes](#)
- [new](#)
- [prmed](#)
- [Adding Pins](#)
- [add line](#)
- [label device](#)
- [label part](#)
- [label refdes](#)
- [label tolerance](#)
- [label value](#)
- [Adding Areas](#)

## Defining a Flash Symbol

To define a flash symbol, perform the following steps:

1. Choose *File – New* ([new](#) command).

The *New Drawing* dialog box appears.

1. Enter a file name in the *Drawing Name* field.

The symbol file name can consist of any alphanumeric characters, and can include the - (dash) and \_ (underscore) characters.

1. Choose *Flash symbol* from the *Drawing Type* list box, and choose *OK*.

The layout editor saves this file with a `.dra` extension to a symbol library.

1. If necessary, choose *Setup – Design Parameters* ([prmed](#) command) and the *Design* tab of the Design Parameter Editor to change settings for the symbol.
2. Choose drawing primitive commands from the Add menu.

Because flash symbols are always defined as `.fsm` files, only filled shapes are supported.

1. Save the file by choosing *File – Save As* ([save\\_as](#) command). Choose the directory where you are saving the symbol drawing file, change the file name if necessary, and choose *Save*.
2. Choose *File – Create Symbol* ([create symbol](#) command) to save the drawing as a package/part symbol. Enter the directory where you are saving the symbol, change the file name if necessary, and choose *Save*.

## Related Topics

- [create symbol](#)
- [Creating a Package Symbol Manually](#)
- [Defining a Mechanical Symbol](#)
- [Creating a Board Outline](#)
- [new](#)
- [prmed](#)
- [save\\_as](#)



## Defining a Format Symbol

Before creating a format symbol, refer to the guidelines for creating format symbols in *Defining and Developing Libraries* in the user guide.

Follow these steps to define a format symbol:

1. Run the `new` command.

The *New Drawing* dialog box appears.

1. Enter a filename in the *Drawing Name* field.
2. Choose *Format symbol* from the *Drawing Type* list box, and choose *OK*.
3. If necessary, choose *Setup – Design Parameters* (`prmed` command) and the *Design* tab of the Design Parameter Editor to change settings for the symbol.
4. Choose drawing primitive commands from the Add and Edit menus.

Ensure that you choose the *Drawing Format* class and the appropriate subclass from the *Options* panel:

- OUTLINE
- TITLE\_BLOCK
- TITLE\_DATA
- REVISION\_BLOCK
- REVISION\_DATA

After you complete each format symbol element, choose Done from the pop-up menu.

1. Save the file by choosing *File – Save As* (`save_as` command). Choose the directory where you are saving the symbol drawing file, change the file name if necessary, and choose *Save*.
2. Choose *File – Create Symbol* (`create symbol` command) to save the drawing as a format symbol. Enter the directory where you are saving the symbol, change the file name if necessary, and choose *Save*.

## Related Topics

- [create symbol](#)
- [Creating a Package Symbol Manually](#)
- [Defining a Mechanical Symbol](#)
- [Defining Custom Pad Shapes](#)
- [prmed](#)
- [save\\_as](#)

## Defining a Mechanical Symbol

Mechanical symbols are created for stiffeners, screws, connectors, shields, and related components that provide mechanical support. Mechanical symbols are not logical components with pins.

If you want to define a mechanical symbol, perform these steps:

1. Open a new file in the Mechanical Symbol mode.
2. Set up drawing parameters.
3. Choose drawing primitive commands from the *Add*, *Layout*, and *Edit* menus.
4. Create keepins and keepouts. See [Adding Areas](#).
5. Create pins without pin numbers (for example, mounting holes) with the `add pin` command. See [Adding Pins](#) for details.
6. Convert the drawing to a mechanical symbol with the `create symbol` command.

## Related Topics

- [create symbol](#)
- [Defining a Format Symbol](#)
- [Defining a Flash Symbol](#)
- [Defining Custom Pad Shapes](#)
- [Adding Areas](#)
- [Adding Pins](#)

## Defining Custom Pad Shapes

Perform the following steps to define custom pad shapes:

1. Choose *File – New* (`new` command) to create the symbol file.

Choose *Shape symbol* from the *Drawing Type* field.

The layout editor saves this file (with a `.dra` extension) to a symbol library.

1. If necessary, choose *Setup – Design Parameters* (`prmed` command) and the *Design* tab of the Design Parameter Editor to change settings for a custom shape.
2. Choose *Add – Circle* (`add circle` command), *Shape – Add Polygon* (`shape add` command), or *Shape – Add Rectangle* (`shape add rect` command) to draw the shape.  
If you define the shape fill type as dynamic in the *Options* panel, the shape fills automatically every time you edit. If you define the shape fill type as static, there is no fill.
3. If you want to further edit the shape, choose *Shape – Select Shape or Void* (`shape select` command).
4. Save the file using *File – Save As* (`save_as` command).
5. Choose *File – Create Symbol* (`create symbol` command) to save the drawing as a shape symbol.

## Related Topics

- [create symbol](#)
- [Creating a Package Symbol Manually](#)
- [Defining a Mechanical Symbol](#)
- [Creating a Board Outline](#)
- [Defining a Format Symbol](#)
- [new](#)
- [prmed](#)
- [add circle](#)
- [shape add](#)
- [shape add rect](#)
- [shape select](#)
- [save as](#)
- [Adding Pins](#)
- [create symbol](#)

## create via structure

The create via structure command allows you to define a standard via structure.

### Related Topics

- [create structure](#)
- [Create Structure Dialog Box](#)

### Define Via Structure Dialog Box

New Via Structure		
	Name	Lets you enter a unique name for a new via structure, or accept the default identifier displayed in the name field. The default name is SVS_1.
	Auto Export XML	Lets you save a defined via structure to an XML-formatted text file, which you can then reuse in the current or other designs. It is recommended to keep this option enabled.
Existing Via Structure		
	Names	Displays the names of the via structures defined in your design.
	Layer stackup for selected via structure	Lists the layers on which the via structure resides. This is for informational purposes only.
Options		
	Cut Clines by Rectangle	If enabled, cut clines and segments inside the drawn rectangle for easy re-use in design.
	Preserve Properties	If enabled, preserve properties attached to vias and clines. By default, the option is disabled.
OK	Closes the dialog box and commits changes to the database.	
Cancel	Exits the command without saving any changes.	

## csvpin in

The `csvpin in` command, available in symbol editor mode, lets you import the symbol pin data from a comma delimited (.csv) file format.

You can import pin number, padstack name, padstack location, pin rotation, text offset values, text rotation and text mirror status from the .csv file.

### Related Topics

- [Importing Symbol Pin Data from a CSV File](#)

## CSV Pin In Dialog Box

### Access Using

- Menu Path: *File – Import – CSV Pin File (in the Symbol Editor)*

<i>CSV file Displays</i>	Specifies the name of the .csv file from which the data is to be imported.
<i>Text block</i>	Specifies the text block size for displaying pin number.
<i>Delete existing pins</i>	Choose if existing pin locations are updated.
<i>Import</i>	Click to import symbol pin data in the .csv file.
<i>Close</i>	Click to close the CSV Pin In dialog box.

## Importing Symbol Pin Data from a CSV File

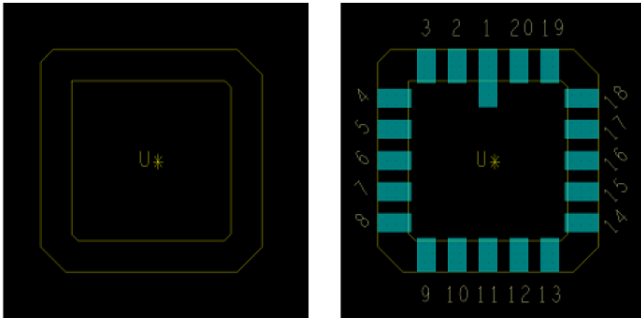
Perform the following steps to import symbol pin data from a CSV file:

1. Run the `csvpin in` command.  
The *CSV Pin In* dialog box appears.
2. Browse to choose the `.csv` file.
3. Specify the text block size.
4. Choose *Delete existing pins* if any existing pin location is altered.
5. Choose *Import* to start the import process.

⚠ If the padstacks specified in the `.csv` file do not exist in the symbol file (`.dra`) or in the library an error message is displayed.

### Example

The following figure shows the symbol before and after import.



### Related Topics

- [csvpin in](#)

## csvpin out

The `csvpin out` command is available in the symbol editor mode. This command lets you export symbol pin data to a comma delimited (`.csv`) file format. You can export pin number, padstack name, padstack location, pin rotation, text offset values, text rotation and text mirror status to the `.csv` file.

### Related Topics

- [Exporting Symbol Pin Data to a CSV File](#)

## CSV Pin Out Dialog Box

### Access Using

- Menu Path: *File – Export – CSV Pin File (in the Symbol Editor)*

<i>CSV file</i>	Specify the name of the output <code>.csv</code> file. By default, the <code>.csv</code> file name is same as the <code>.dra</code> file. You can also enter a new name.
<i>Include pin text location</i>	Select this option to include pin number text X-Y location, rotation and mirror status in the output file.
<i>Export</i>	Click to export symbol pin data in the <code>.csv</code> file.
<i>Close</i>	Click to close the CSV Pin Out dialog box.



## Exporting Symbol Pin Data to a CSV File

If you want to export symbol pin data to a CSV file, perform the following steps:

1. Run the `csvpin out` command.  
The *CSV Pin Out* dialog box appears with the file name similar to symbol name.
2. Alternatively, browse to choose the output `.csv` file or enter a new name.
3. To include pin text location, choose *Include pin text location*.
4. Click *Export* to export the symbol pin data.

A `.csv` file of the same name as the `.dra` design is written to your current working directory.

### Example

The following figure shows an the `.csv` file generated after export.

	A	B	C	D	E	F	G	H	I	J
1	# If units not specified use current design units									
2	Units            mils									
3	# Format for pin definition file (comma delineated)									
4	# To Mirror pin text use "m".									
5	#PinNumber	Padstack	x	y	rotation	textOffsetX	textOffsetY	textRotate	textMirror	
6	1	SMD30_94	0	132.5	0	-10	70	0		
7	2	SMD30_55	-50	152	0	-10	50	0		
8	3	SMD30_55	-100	152	0	-10	50	0		
9	4	SMD30_55	-152	100	90	-50	-10	45		
10	5	SMD30_55	-152	50	90	-50	-10	45		
11	6	SMD30_55	-152	0	90	-50	-10	45		
12	7	SMD30_55	-152	-50	90	-50	-10	45		
13	8	SMD30_55	-152	-100	90	-50	-10	45		
14	9	SMD30_55	-100	-152	0	-10	-75	0		
15	10	SMD30_55	-50	-152	0	-50	-75	0		
16	11	SMD30_55	0	-152	0	-50	-75	0		
17	12	SMD30_55	50	-152	0	-50	-75	0		
18	13	SMD30_55	100	-152	0	-50	-75	0		
19	14	SMD30_55	152	-100	90	40	-20	45		
20	15	SMD30_55	152	-50	90	40	-20	45		
21	16	SMD30_55	152	0	90	40	-20	45		
22	17	SMD30_55	152	50	90	40	-20	45		
23	18	SMD30_55	152	100	90	50	-20	45		
24	19	SMD30_55	100	152	0	-15	50	0		
25	20	SMD30_55	50	152	0	-15	50	0		
26										

## Related Topics

- [csvpin out](#)

## **ctab**


An internal Cadence engineering command.

## custom\_route

The `custom_route` command categorizes and writes several individual rules files based on the characteristics of your design. Launches PCB Router and loads a `<design_name>_custom_route.do` file that uses only relevant rules files for your design once routing commences. Presents a banner announcing Custom Route and awaits your input.

In addition to improving routing efficiency, this custom strategy provides you with opportunities within PCB Router to:

- adjust setup parameters before autorouting begins.
- view the design as it is being autorouted.
- intervene in the autoroute process (Pause, Continue, Stop) if necessary.

 As with other automatic routing strategies, the use of the `custom_route` command creates a `<design_name>.dsn` file for use in PCB Router.

### ***Access Using***


- Menu Path: *Route – Route Custom*


## Using the Custom Route Command

1. Choose *Route – Route Custom*

Alternately, type `custom_route` in the Command line.

The PCB Router user interface appears with your design and a banner area announcing Custom Route. This area also displays important messages and prompts. *Continue* and *Stop* buttons display in the lower-left corner of the PCB Router window enabling you to interact with the autorouting process as required.

 *Stop* suspends Custom Route without any further action. You can then either continue to work in Allegro PCB Router on your own, or you can choose *File – Quit* to return to the layout editor without saving the session (.ses) file.


 At this point, you may wish to check a few setup parameters such as layer selection and direction, grid settings, and so on. You may also want to change the zoom scale of the design window to enhance your view of the design as it routes.

2. Choose *Continue*.

Your design is analyzed for high-speed rules. A length and delay rule report is generated and displayed in a text window. Examine this report carefully before proceeding.

3. Choose *Continue*.

Autorouting commences and progress messages appear in the banner area. A *Pause* button displays in the lower-left corner.

 When displayed, you can choose *Pause* to temporarily halt the progress of Custom Route, examine the results, and when ready, choose *Continue* to resume autorouting.

4. Continue to monitor the banner area for messages and prompts, clicking *Stop*, *Continue*, or *Pause* as appropriate.

A green *Idle* button displays in the lower-left corner of the PCB Router window when autorouting is complete.

5. Choose *File – Quit – Save And Quit*, to save the routing results and return to the layout editor.

- or -

Choose *File – Quit – Quit (No Save)*, to return to the layout editor without saving the routing results.

## custom datatips

The `custom datatips` command lets you customize a context-sensitive datatip that identifies an element when the cursor hovers over it.

The default datatip configuration file `custdatatips.cdt` is located at `<installation_directory>/share/pcb/text`. If you've customized any settings, they are stored in a local `custdatatips.cdt` located in the `pcbenv` directory.

When you launch PCB Editor, the local `pcbenv` directory is checked for `custdatatips.cdt` file. If it exists, the customized datatip settings are loaded from it, else they are loaded from the default file.

### Related Topics

- [Customizing a Datatip](#)

## Customizing a Datatip

Follow these steps to customize context-sensitive datatips:

1. Choose *Setup – Datatip Customization*.  
Alternately, type `custom datatips` in the Command line.
2. Open a `.cdt` file containing datatips customization required, or use the default `.cdt` that loads automatically.
3. Choose an element in *Object type*; all information related to the element displays.
4. Choose the *General* or *Advanced* tab for defining data tops format.
5. Choose the information and values to display in the datatips as required.
6. Specify the data tips format.
7. Click *OK* to apply.

## Related Topics

- [custom datatips](#)

## DataTips Customization Dialog Box

### Access Using

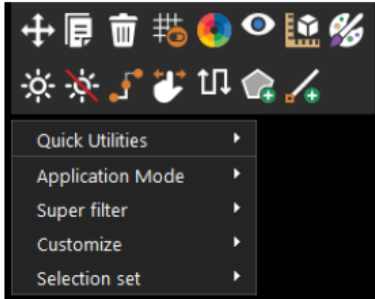
- Menu Path: *Setup – Datatip Customization*

<b>Object type</b>	Customizes datatips for bundle, cline, DRC, figure, flow line, flow via, flow segment, net, net group, pin, plain line, plan line, plan via, port group, segment, shape, symbol instance, and text.
<b>General tab</b>	Lists information to display in a datatip for the selected element in the <i>Object Type</i> . Select the <i>Name</i> checkbox to include it in the datatip; the <i>Value</i> checkbox is checked automatically, indicating its inclusion in the datatip as well. Select the <i>Value</i> checkbox to only include the alphanumeric character string associated with the information in the datatip. Choose <i>All</i> to display both <i>Name</i> and <i>Value</i> for the selected element in the datatip.
<b>Advanced tab</b>	Displays all properties applicable to the selected <i>Object Type</i> in the datatip. Select the <i>Name</i> checkbox to include it in the datatip; the <i>Value</i> checkbox is checked automatically, indicating its inclusion in the datatip as well. Select the <i>Value</i> checkbox to only include the alphanumeric character string associated with the information in the datatip. Select the <i>Save</i> checkbox, appears next to the user-defined attributes, to include these properties in the <i>.cdt</i> file on saving it.  Select <i>All</i> to check the check boxes in the column to display all information available for the chosen element in the datatip.  <div style="border: 1px solid #ccc; padding: 5px; background-color: #fff9c4;">⚠ For Net objects, <i>Path length</i> and <i>Manhattan length</i> are included in the Advanced tab.</div>
<b>Property filter</b>	Enter whole words or character strings to locate a subset of the properties available for the selected element. To specify a character string, use asterisk (*) as a wildcard character. Displays only when the you choose the <i>Advanced tab</i> .
<b>Specify Datatips Format</b>	Displays datatip contents for a selected Object type. The red labels indicate that only the <i>Value</i> will be displayed in the datatip and the blue labels will display both <i>Name and value</i> for the selected Object type.  You can drag and drop content in this field. In addition, you can also change the color for <i>Name</i> and <i>Name and value</i> by left mouse button click on the color swatch.
<b>OK</b>	Saves settings to the <i>.cdt</i> file currently loaded and closes the dialog box.
<b>Cancel</b>	Closes the dialog box without saving any changes.
<b>Reset to Default</b>	Removes all datatips customization and restores original settings as specified in the <i>custdatatips.cdt</i> located in the <i>&lt;installation_directory&gt;/share/pcb/text</i> .
<b>Load – Load default CDT file</b>	Loads settings from the default <i>custdatatips.cdt</i> file.
<b>Save – Save default CDT file</b>	Saves modifications to the default settings in the <i>custdatatips.cdt</i> file.
<b>Load – Load Custom CDT File</b>	Imports your customized datatip settings from an external <i>.cdt</i> file and applies them to the current design. A file browser appears with the filter set to <i>*.cdt</i> and a list of all <i>.cdt</i> files available in the current local working directory. You can manually browse to other directories to open a <i>.cdt</i> file.
<b>Save – Save Custom CDT File</b>	Exports the current design's customized datatip settings to an external <i>.cdt</i> file. A file browser appears with the filter set to <i>*.cdt</i> and a list of all <i>.cdt</i> files in the current local working directory. You can manually browse to other directories to specify an alternate save location.

## Customize Toolbar

Use the Customize dialog box to control the look of the toolbar on your user interface. You can also add custom icons to toolbar commands. The path of custom icon directory is set by the *iconpath* environment variable and the default custom icon directory is available at `<installation_directory>/share/pcb/text/toolbar_icons/icons`

Apart from standard task-specific toolbars, a *ContextMenu* toolbar is also available that contains frequently-used commands.



If enabled, the *ContextMenu* toolbar starts appearing on the right mouse button pop-up menu.

### Related Topics

- [Hiding Toolbar Categories](#)
- [Rearranging Buttons on Toolbars](#)
- [Creating Your Own Toolbar](#)
- [Deleting a Custom Toolbar](#)
- [Assigning or Modifying Icon to a Command](#)



## Assigning or Modifying Icon to a Command

1. Choose *View – Customize Toolbar*.  
The *Customize* dialog box appears.
2. Choose the *Commands* tab.
3. Select the toolbar category from the pull-down menu.
4. Select the command that contains the button you want to change or add.
5. Click *Change Icon*.  
A file browser opens to choose the icon file from the custom icon directory set by the *iconpath* environment variable
6. Choose the image file.  
The icon of the selected command changes or added with the selected command in the toolbar.
7. To restore the default icon or to remove the custom icon, choose *Reset Icon*.

## Related Topics

- [Customize Toolbar](#)
- [Customize Dialog Box](#)
- [Hiding Toolbar Categories](#)
- [Rearranging Buttons on Toolbars](#)
- [Creating Your Own Toolbar](#)

## Creating Your Own Toolbar

1. Choose *View – Customize Toolbar*.  
The *Customize* dialog box appears.
2. Choose the *Toolbars* tab.
3. Choose *New*.  
The *New Toolbar* dialog box appears.
4. Enter the name for the new toolbar in the name field.
5. Choose *OK* to close the *New Toolbar* dialog box.  
The new toolbar is added to the list in the *Toolbars* window and a new toolbar appears. Choose the *Commands* tab and then add buttons you want to add, to your new toolbar.  
The new toolbar expands to hold as many buttons as you want.
6. Drag your toolbar to the location you want to use it from.  
You can use it as a floating vertical or a horizontal toolbar anywhere on your desktop, however, as soon as you drop it into the toolbar area, it becomes fixed.

## Related Topics

- [Customize Toolbar](#)
- [Customize Dialog Box](#)
- [Hiding Toolbar Categories](#)
- [Assigning or Modifying Icon to a Command](#)

## Customize Dialog Box

### Access Using

- Menu Path: *View – Customize Toolbar*

Toolbars Tab	
<i>Toolbars</i>	Lets you display or hide categories of toolbar buttons. Checking off all the toolbars extends the size of the working area in the user interface.
<i>Toolbar Name</i>	Displays the name of the selected toolbar.
<i>New</i>	Opens the <i>New Toolbar</i> dialog box that lets you add a user-defined toolbar category to the list. You can drag existing toolbar buttons into this new category and add them to the toolbar in the user interface.
<i>Delete</i>	Deletes the user-defined toolbar selected in the <i>Toolbars</i> section.

Commands Tab	
<i>Select a toolbar to rearrange</i>	Select a toolbar category from the pull-down menu to modify.
<i>Add Command</i>	Opens the <i>Add Command</i> dialog box that lets you add commands to the selected toolbar category. In the <i>Add Command</i> dialog box, commands are listed in order of menu and submenus. A list of <i>All Commands</i> is also available, which is sorted by command name.
Change Icon	Lets you browse the custom icon directory to change the selected icon. To set the custom icon directory, set the environment variable <i>iconpath</i> in User Preferences Editor. The valid image formats are *.bmp, *.png, *.jpg. This option gets enabled when a command is selected.
Reset Icon	Resets the selected icon to Cadence default icon. This option gets enabled when a command is selected.
<i>Buttons</i>	Displays the buttons that appear in the selected toolbar category.
<i>Move Up</i>	Moves up the selected button in the <i>Buttons</i> window. You can also drag the button to change its position in the toolbar
<i>Move Down</i>	Moves down the selected button in the <i>Buttons</i> window. You can also drag the button to change its position in the toolbar.
<i>Delete</i>	Deletes the selected button from the <i>Buttons</i> window.
<i>Reset</i>	Returns the toolbars to their default configurations.

### Related Topics

- [Rearranging Buttons on Toolbars](#)
- [Creating Your Own Toolbar](#)
- [Deleting a Custom Toolbar](#)
- [Assigning or Modifying Icon to a Command](#)

## Deleting a Custom Toolbar

1. Choose *View – Customize Toolbar*.  
The *Customize* dialog box appears.
2. Choose the *Toolbars* tab.
3. Highlight the toolbar you want to delete, and then choose *Delete*.  
The *Delete* button is not available unless you have chosen a toolbar you previously defined.


## Related Topics

- [Customize Toolbar](#)
- [Customize Dialog Box](#)
- [Hiding Toolbar Categories](#)
- [Rearranging Buttons on Toolbars](#)

## Hiding Toolbar Categories

You can hide all toolbar categories, or only those that you do not use.

1. Choose *View – Customize Toolbar*.  
The *Customize* dialog box appears.
2. Choose the *Toolbars* tab.  
A listing of all the available Toolbars appears.
3. Deselect the boxes next to the toolbar categories in the Toolbars window that you want to hide.  
The buttons disappear. If you deselect all the categories, the toolbar disappears, giving you more design workspace.

 You can see the different toolbar categories by clicking the *Commands* tab on the *Customize* dialog box and highlighting a category. The corresponding buttons display in the *Buttons* window.


## Related Topics

- [Customize Toolbar](#)
- [Creating Your Own Toolbar](#)
- [Deleting a Custom Toolbar](#)
- [Assigning or Modifying Icon to a Command](#)

## Rearranging Buttons on Toolbars

You can rearrange buttons for any toolbar category. This is useful if you only use one button from a category. You can add the button to another category and remove the toolbar category that you do not use from the design window.

1. Choose *View – Customize Toolbar*.  
The *Customize* dialog box appears.
2. Choose the *Commands* tab.
3. Select the toolbar category to rearrange from the pull-down menu.
4. Click *Add Command*. Select the toolbar that contains the button you want to add. Choose the commands and add them by clicking *Add*. Close *Add Command* dialog box once all the commands are added.
5. Choose and drag the buttons to rearrange the positions of the buttons.


 To return the toolbars to their default configuration, choose *Reset*.

## Related Topics

- [Customize Toolbar](#)
- [Customize Dialog Box](#)
- [Deleting a Custom Toolbar](#)
- [Assigning or Modifying Icon to a Command](#)

## custom smooth

The `custom smooth` command optimizes chosen clines or cline segments according to parameters set in the *Options* panel. Smoothing the angles of clines or cline segments can minimize the distance to pad connections. Use this command as you manually edit etch.

 This feature does not operate on clines with DRC errors, so you may need to update DRCs and clean up your design before using this command.

### Related Topics

- [Running Custom Smoothing](#)

## Custom Smooth Command: Options Panel

### Access Using

- Menu Path: *Route – Custom Smooth*



- Toolbar Icon:

<i>Corner type</i>	Choose the type of corner to be generated when smoothing. The default is <i>45</i> .
<i>Restrict seg entry for pads of type</i>	Choose a pad type. The existing angle of the cline segment that connects to this pad type is not altered by smoothing. Choices are <i>Rectangular</i> , <i>All</i> , and <i>None</i> . The default is <i>Rectangular</i> , which includes all non-circular pads.
<i>Minimum pad entry length</i>	Enter the minimum length that a cline segment connected to a restricted pad (chosen in the previous field) can be shortened to. For example, if the cline segment is 250, and you enter 100 in this field, it means that the cline segment shortens only to 100 units. A negative value indicates unlimited length. The values are in user units, and the default is <i>-1</i> . If you enter a value of <i>-1</i> , the segment is not broken. See <a href="#">example</a> .
<i>Max iterations</i>	Enter the number of times a set of clines or cline segments can be smoothed. This feature allows you to perform a number of smoothing passes for instances when smoothing one cline clears room for a previously smoothed cline to be further smoothed. The default is <i>10</i> .



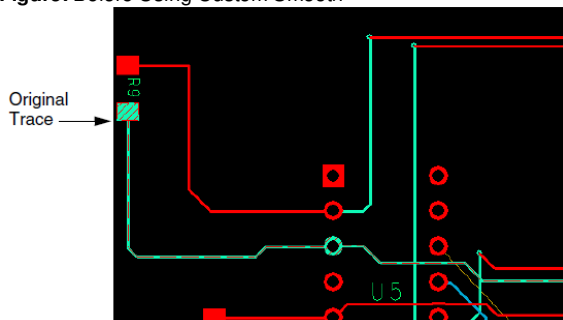
## Running Custom Smoothing

1. Run `custom smooth`.  
Alternately, choose *Route – Custom Smooth*.
2. Set parameters in the *Options* panel. For details, see [Custom Smooth Command: Options Panel](#).
3. Choose *Nets*, *Clines*, and *Segments* from the Find filter.
4. Choose one object. –or– From the pop-up menu, choose Temp Group, choose a number of objects, and choose *Complete* from the pop-up menu.
5. Choose *Done*.

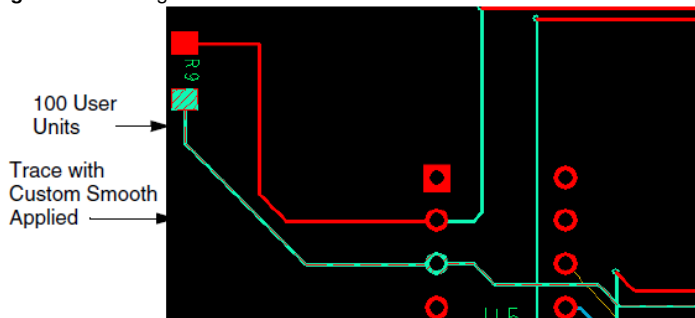
### Example

The following figures show a board example before and after using the custom smooth command. For the highlighted trace in [the second figure](#), *Corner Type* is 45, *Restrict seg entry for pads of type* is Rectangular, and Minimum pad entry length is 100 user units.

**Figure:** Before Using Custom Smooth



**Figure:** After Using Custom Smooth




### Related Topics

- [custom smooth](#)

## cut marks

The `cut marks` command displays the *Cut Mark Options* dialog box where you define and add cut marks at each corner of a board outline. When you use this command in a design that previously did not have them, the tool automatically creates a CUT\_MARKS subclass.

 Cut marks can be applied only to closed shapes.

### Related Topics

- [Adding Cut Marks to a Board Outline](#)

## Adding Cut Marks to a Board Outline

1. Run the `cut marks` command.

Alternately, choose *Manufacture – Cut Marks*.

2. If necessary, run the `color` command to set color and visibility options for the CUT\_MARKS subclass in the Geometry Group in the Color dialog box.

The CUT\_MARKS subclass is created the first time you run the cut marks command on this design.

1. In the Cut Mark Options dialog box, set the options you need. For details, see [Cut Marks Options Dialog Box](#).
2. Choose *Apply*.

Any existing cut marks are replaced by new cut marks.

1. Choose *OK*.

## Related Topics

- [cut marks](#)

## Cut Marks Options Dialog Box

### Access Using

- Menu Path: *Manufacture – Cut Marks*

Use this dialog box to specify cut marks for your board outline.

<i>Line options</i>	The values for these settings are in the units you defined using <i>Setup – Design Parameters</i> ( <a href="#">prmed</a> command) and the <i>Design</i> tab of the Design Parameter Editor. You can also use the Drawing Parameters dialog box ( <a href="#">drawing param</a> command).
<i>Offset</i>	Sets the cut marks at a specific distance from the board outline. The default is 50.
<i>Length</i>	Defines the line length of each cut mark.
<i>Line width</i>	Specifies the thickness of the cut mark.
<div style="border: 1px solid #ccc; background-color: #fff9c4; padding: 10px; margin: 10px 0;"> <p>⚠ For the next two options, the default method (no option chosen) creates cut marks for arcs in the same manner as they are generated for lines.</p> </div>	
<i>Arcs as corners</i>	Creates cut marks as corners at the endpoints of the arc.
<i>Always complete arcs</i>	Creates cut marks that span the entire length of the arc, ignoring the <i>Length</i> option.
Corner type	<p>Creates cut marks for the inside and/or outside corners of the outline. This board outline has one inside corner and five outside corners:</p> 