

A 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
A Commands	10
a2dxf	12
Creating a DXF file from a Design Using a Batch Command	14
About Dialog Box	15
acroread	16
Reading a PDF file	17
active subclass	18
Changing a subclass	19
add arc	20
Add Arc Command: Options Panel	21
Creating an Arc-Shaped Element	22
Changing the Font of Arc or Circle	23
add circle	24
Add Circle Command: Options Panel	25
Adding a Circle	28
add codesign die	29
Add Co-Design Die Dialog Box	29
Adding an Existing OA Co-Design Die to the Layout Editor	34
Adding a New Co-Design Die Using DEF	35
Adding a New Co-Design Die Using Verilog	36
Adding a New Co-Design Die Using a Die Abstract	37
add codesign pkg	38
add connect	39
Add Connect Command: Options Panel	40
Add Connect Command: Pop-Up Menu Options	44
Add Connect Command: Tasks	48
Adding a Connect Line	49
Adding a Through-Hole Via While Routing a Single Trace	51
Adding a Via Structure While Routing	52
Adding a Jumper While Routing a Single Trace	53
Routing from or to Rat Ts	54
Using Single Trace Mode With Differential Pairs	55

Adding Vias to a Differential Pair	56
Changing Via Patterns	57
Changing Via Spacing Using the Diff Pair Via Space Dialog Box	58
Routing Groups	59
Using Single Trace Mode During Group Routing	60
Changing the Spacing Mode During Group Routing	61
Routing with Layer-Set Constraints	62
Performing a Freestyle Multi-line Route	63
Routing Connections Using the Contour Option	64
Routing Using Route Offset Angle	65
Routing in Channel	66
add d2d vias	67
Add D2D Vias Command: Options Panel	67
Adding D2D Vias to Design	67
add fillet	68
Generating Fillets Interactively	69
add flash	70
Thermal Pad Symbol Defaults Dialog Box	71
Defining Parameters of a Flash Thermal Pad	72
add frect	73
Creating Filled Rectangles	74
add fshape	76
Add Fshape Command: Options Panel	77
Adding Elements to a Design	78
add interposer	80
Add Interposer Dialog Box	82
Adding an Interposer to a Die-Stack	83
add line	84
Add Line Command: Options Panel	85
Creating Non-Etch/Conductor Line Segments between Two Points	86
add parallel line	87
Add Parallel Line Command: Options Panel	88
Adding Parallel Lines to your Design	89
add perp line	90
Adding Perpendicular Lines	91
add pin	92
Add Pin Command: Options Panel	93

Adding Pins	94
add rarc	96
Add RARC Command: Options Panel	97
Adding Arcs by Specifying the Radius	98
add rect	100
Add Rect Command: Options Panel	101
Adding a Rectangle	102
Adding a Room	102
add ruler	104
Creating Static Rulers	105
Controlling Display of Static Rulers	106
add spacer	107
Add Spacer Dialog Box	108
Creating Spacer Symbols	109
add tangent line	110
Adding Tangent Lines to an Arc or a Circle	111
add taper	112
Adding Tapers	112
add testpoint	113
Adding Testpoints	113
add text	114
Add Text Command: Options Panel	114
Adding Text to a Design	115
Assigning a Room Name	115
add vertex	117
Add Vertex Command: Options Panel	117
Adding a Vertex	119
add xshape	120
Add Xshape Command: Options Panel	121
Adding Closed, Cross-hatched Filled Shapes, or Elements To a Design	122
advanced highlight	123
Advanced Highlight Dialog Box	124
Setting Object Color based on Characteristics:	125
Advanced Package Router	126
Advanced Package Router Dialog Box	127
Advanced Selection Filtering Dialog Box	131
Routing Constraint Driven Flip-Chip Designs	132

Filtering Nets and Pins	133
adv nonstandard fillets	133
Adv Nonstandard Fillets Command: Options Panel	134
Updating Fillets in the Design	135
adv thieving	135
Adv Thieving Command: Options Panel	136
Creating a Thieving Pattern	138
aibt deletebreakout	139
Deleting Interconnect between Breakouts	140
aibt routetrunk	141
Connecting the Breakout on Both Ends of a Bundle	142
aibt single	143
Breaking Out one side (end) of a Bundle	144
Breaking Out both sides (ends) of a Bundle	145
aibt trimtobreakout	146
Adjusting Breakouts on a Bundle	147
aidt	148
AiDT Command: Options Panel	148
Using the Auto-interactive Delay Tune (AiDT) Command	150
aif in	151
aif out	151
aipt	151
AIPT Command: Options Panel	152
Using the Auto-interactive Phase Tune (AiPT) Command	153
alias	154
Creating a Command Alias	154
alias_protect	156
Assigning an Alias Read-Only Status	157
align components	158
Align Components Command: Options Panel	159
Aligning Components to Optimize Routing Channels	161
align groups	162
Aligning Groups to Optimize Routing Channels	163
align modules	164
Aligning Modules to Optimize Routing Channels	164
allegro	165
allegro_component	167

Component Options Dialog Box	168
Exporting from a .mcm Design	170
Importing Part Information into Design Entry HDI or System Connectivity Manager	171
allegro_cshrc	174
allegro_downrev_library	175
allegro_plot	177
Allegro Plot Dialog Boxes	178
Running the Allegro Plot Command in Graphical Mode	183
allegro_uprev	184
Updating a Design Database	185
allegro_uprev_overwrite	186
altsubclass	187
anchor 3d view	188
angle	189
Specifying an Angle Value	190
annotation in	191
Importing an ASCII File with Board Information	192
annotation out	193
Exporting Board Information in an ASCII File	194
apd	195
aperture	197
Edit Aperture Wheels Dialog Box	198
Specifying an Aperture Wheel	199
Editing Aperture Data	200
Generating the Aperture Data File	202
apick	203
Highlighting Objects	204
apick_to_grid	205
artwork	206
Generating Artwork Data Files	207
assemrules custom	209
assemrules standard	210
Assembly Design Rule Checks Dialog Box	211
Setting Up and Running the Assembly Rules Checker	213
assign color	214
Assign Color Command: Options Panel	215
Assigning a Custom Color or Highlighting an Element	216

assign multi nets	217
Multi-Net Assignment Dialog Box	218
Assigning Multiple Nets to Pins	222
assign net	223
Assign Net Command: Options Panel	224
Assigning Pins to an Existing Net	225
assign plating layer	226
Assign Plating Layer Command: Options Panel	227
Assigning a Plating Layer to Pins and Nets	228
assign port	229
assign portgroup	230
assign power	231
assign refdes	232
Assign Refdes Command: Options Panel	233
Assigning Reference Designators to Package Symbols	234
assign region	235
Assigning Region to Multiple Shapes	236
assign route layer	237
Assign Route Layer Command: Options Panel	238
Assigning A Routing Layer to Pins and Nets	239
auto_connect	240
auto_pin_renumber	241
Pin Renumbering Options Dialog Box	242
Renumbering Pins Automatically	243
auto_route	243
Automatic Router Dialog Box	243
Routing the Board Automatically	247
auto assign net	247
Automatic Net Assignment Dialog Box	248
Assigning Nets to Design	249
auto assign pinuse	250
Auto Pin Use Assignment Dialog Box	251
Setting Pin Use Codes on Component Pins	252
autobundle	253
Bundling Rats Associated with Selected Objects Automatically	254
auto create net	254
Auto Create Net Dialog Box	255

Creating Nets Automatically	256
auto define bbvia	257
Create bbvia Dialog Box	258
Creating Blind or Buried Vias Automatically	260
awb2therm	261
axlmark	262

A Commands

a2dxf	About Dialog Box	acoread
active subclass	add arc	add circle
add codesign die	add codesign pkg	add connect
add d2d vias	add fillet	add flash
add frect	add fshape	add interposer
add line	add parallel line	add perp line
add pin	add rarc	add rect
add ruler	add spacer	add tangent line
add taper	add testpoint	add text
add vertex	add xshape	advanced highlight
Advanced Package Router	adv nonstandard fillets	adv thieving
aibt deletebreakout	aibt routetrunk	aibt single
aibt trimtobreakout	aidt	aif in
aif out	aipt	alias
alias_protect	align components	align groups
align modules	allegro	allegro_component
allegro_cshrc	allegro_downrev_library	allegro_plot
allegro_uprev	allegro_uprev_overwrite	altsubclass
anchor 3d view	angle	annotation in
annotation out	apd	aperture
apick	apick_to_grid	artwork
assemrules custom	assemrules standard	assign color
assign multi nets	assign net	assign plating layer
assign port	assign portgroup	assign power
assign refdes	assign region	assign route layer

A Commands

A Commands

auto_connect	auto_pin_renumber	auto_route
auto assign net	auto assign pinuse	autobundle
auto create net	auto define bbvia	awb2therm
axlmark		

a2dxf

The `a2dxf` batch command exports mechanical design data from a database design into a DXF file in the ASCII format, using either DXF Revision 12 or 14. You can also use the `a2dxf` command to selectively output certain classes or subclasses that correspond to specific layers in a DXF file.

Syntax

```
a2dxf [-u units] [-a accuracy] [-b] [-d] [-f] [-h] [-l] [-n] [-m] [-p] [-s] [-version] <layer_conversion_filename> <dxfname> <designname>
```

>

-u units	Optional. The unit of measurement for the DXF output. Specify one of the following unit types, using the abbreviation for the unit type or entering the complete unit name in either all lowercase or uppercase letters: <ul style="list-style-type: none"> • ML, mils, or MILS • IN, inches, or INCHES • CM, centimeters, or CENTIMETERS • MM, millimeters, or MILLIMETERS • MC, microns, or MICRONS
-a accuracy	Optional. The number of decimal places that represent the level of accuracy specified for the DXF file. The number must be a positive integer. You can specify the accuracy upto four decimal places. For example, for three decimal places (.000), enter the argument: <pre>-a 3</pre> <p>If you do not specify an accuracy, the interface program uses the accuracy of the design file. If the accuracy specified here is not as precise as the accuracy of the design, some data, such as arcs, may not convert to the DXF file. Data values may also be inaccurate.</p>
-b	Optional. Indicates that symbol definitions and padstacks are exported as blocks. By default it is off. You can specify: S for symbols P for padstacks
-d	Optional. Specifies that the resulting DXF file contains drill figure information corresponding to pin and vias. To use the <code>-d</code> option, the layer conversion file must contain a DXF layer name corresponding to class MANUFACTURING and its subclass NCLEGEND-<L1>-<L2>, where <L1> and <L2> are the layer numbers of the drilled layers. Otherwise, the drill information is output in a layer name MANUFACTURING_NCLEGEND-<L1>-<L2> in the DXF file.
-f	Optional. Indicates to the layout editor which revision of the DXF file format to write. If you do not include this argument, the layout editor defaults to Revision 12 to maintain backwards compatibility. Legal values are 12 and 14.
-h	Optional. Specifies a default value for symbol package height. The value must be consistent with the specified DXF units. For example, to have the default package height be one-quarter inch, and the DXF units are INCHES, enter the argument: <pre>-h .25</pre> <p>The value is used for all packages that do not have a specified package height. Valid only when combined with the <code>-b</code> argument.</p>
-l	Optional. Indicates to the layout editor that when exporting a Revision 14 format file, it should create shapes representing the lines, which are either filled or unfilled, based on the <code>-s</code> argument specified.
-n	Produces a DXF file without multi-segment polylines. Exports each line segment or cline as a separate DXF polyline.
-c (val)	Specifies the color. Choose one of the three vales: <pre>m - monochrome</pre> <pre>l - color by layer, assign design's layer color to the objects</pre> <pre>s - sequential</pre>
-m	Choose to export the entities in the drawing as white to ensure that if you convert the DXF file to a PDF format, the white entities become black lines and therefore more readable when you print the PDF drawing. Otherwise, entities retain their colors, but are difficult to read against the white background of the printed PDF drawing.

A Commands

A Commands--a2dxf

-s	<p>Specifies fill solid-fill shapes. If you do not define the variable, these shapes are exported unfilled.</p> <p>If writing Revision 12, shapes are filled with solid lines of the specified line width. The value must be consistent with specified DXF units. You are responsible for setting a value legal for all shapes. A legal value is any decimal number greater than 0. The default is off, which means that the shapes are exported unfilled.</p> <p>If writing a Revision 14 version (see the <code>-f</code> argument), then the value for the line width is ignored, and the shape is filled as a HATCH block in the resulting file.</p>
-p	<p>Specifies fill solid-fill pads. If you do not define the variable, these pads are exported unfilled.</p> <p>If writing Revision 12, the pads are filled with solid lines of the specified line width. The value must be consistent with specified DXF units. You are responsible for setting a value legal for all pads. A legal value is any decimal number greater than 0. The default is off, which means that the pads are exported unfilled.</p> <p>If writing a Revision 14 version (see the <code>-f</code> argument), then the value for the line width is ignored, and the pad is filled as a HATCH block in the resulting file.</p>
-version	<p>Prints the version.</p>
layer_conversion_file	<p>Required. The name of ASCII text file that specifies the mapping of DXF layer to design class/subclass.</p> <p>If the layer conversion file that you created is not in the directory from which you run the <code>a2dxf</code> command, specify the complete directory path for the layer conversion file.</p>
dxfname	<p>Required. The name of the file in which the design data is output in DXF format. You do not need to enter the <code>.dxf</code> extension.</p> <p>If the DXF output file is not to be placed in the directory in which you run the <code>a2dxf</code> command, specify the complete directory path for the DXF file.</p>
designname	<p>Required. The name of the database design from which the data is extracted. You do not need to enter the <code>.brd</code> extension.</p> <p>If the design is not in the directory from which you run the <code>a2dxf</code> command, specify the complete directory path for the design.</p>

Related Topics

- [dxf out](#)
- [DXF Bi-directional Interface](#)

Creating a DXF file from a Design Using a Batch Command

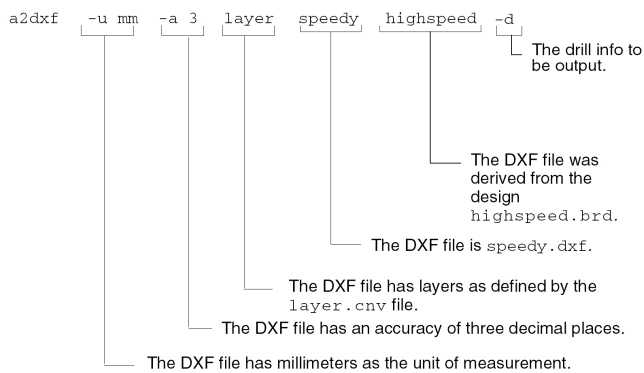
Use the following procedure to run the `a2dxf` command:

1. Enter `a2dxf` and appropriate arguments at your operating system command prompt.
The command prompts for the name of a layer conversion file if you do not specify one. If the specified file does not exist, a default layer conversion file is created with the given name.
The command prompts for the DXF file and a layout file if you do not specify in the argument list.
The interface program creates the DXF file(.dxf) using the database design specified.
A log file (`a2dxf.log`) is also generated describing the process and any errors or warning messages.

Example

The command shown in Figure 1-1 creates a DXF file called `speedy.dxf` :

Figure 1.1: Example Showing the `a2dxf` Command



Related Topics

- [DXF Bi-Directional Interface](#)
- [Specifying an Aperture Wheel](#)

About Dialog Box

The About dialog box shows release information about the version of the Cadence product you are using. This information may be useful if you need to call Cadence Design Systems.

Access using

- Menu path: *Help – About*

acroread

The `acroread` command lets you read a PDF file by opening the Adobe® Acrobat® software installed on your machine.

The command does not run if Acrobat is not installed.

```
acroread <filename>
```


Reading a PDF file

To read a PDF file, in the command console of the product you are running, do the following step:

1. Type `acroread` and the name of the file to open.
The file opens in Acrobat.

active subclass

The `active subclass` command changes the subclass that is active.

Use the pop-up command *Quick Utilities – Change Active Subclass* to run the command.

Changing a subclass

1. Hover your cursor anywhere in the design canvas.
2. Right-click and choose *Quick Utilities – Change Active Subclass* from the pop-up menu.

add arc

The add arc command creates an arc-shaped element using mouse button clicks. Run the `add arc` command when the end points of the arc are known. `add arc` requires three points: a point to start the arc, an end point, and a third point to determine the radius of the arc. To create an arc, specify three points either by mouse click or by typing cursor coordinates at the command line.

Related Topics






- [add rarc](#)
- [Creating an Arc-Shaped Element](#)
- [Changing the Font of Arc or Circle](#)

Add Arc Command: Options Panel

Access using

- Menu path: *Add – 3pt Arc*

The following table describes the various fields available in the *Options* panel for the `add arc` command:

<i>Line width</i>	Defines the width of the Solid lines used in creating the arc in user units. All other line fonts remain at 0 width.
<i>Line font</i>	<p>Defines the line pattern used in creating the arc. The choices are Solid , Hidden , Phantom , Dotted , and Center . The default is Solid .</p> <p>The line pattern types are:</p> <div style="margin-left: 40px;"> <p>Solid </p> <p>Hidden </p> <p>Phantom </p> <p>Dotted </p> <p>Center </p> </div> <p>Line fonts, other than Solid, are allowed on the following Class/Subclasses:</p> <ul style="list-style-type: none"> ◦ DRAWING FORMAT/All user defined subclasses ◦ MANUFACTURING/NCDRILL_LEGEND ◦ MANUFACTURING/All user defined subclasses ◦ PACKAGE GEOMETRY/ASSEMBLY_TOP ◦ PACKAGE GEOMETRY/ASSEMBLY_BOTTOM ◦ PACKAGE GEOMETRY/All user defined subclasses ◦ BOARD GEOMETRY/OUTLINE ◦ BOARD GEOMETRY/ASSEMBLY_NOTES ◦ BOARD GEOMETRY/ DIMENSIONS ◦ BOARD GEOMETRY/ASSEMBLY_DETAIL ◦ BOARD GEOMETRY/All user defined subclasses

Related Topics

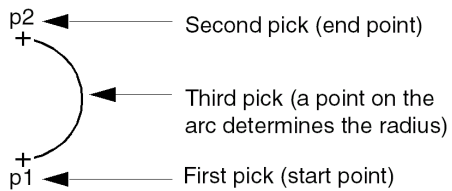
- [add arc](#)
- [Changing the Font of Arc or Circle](#)

Creating an Arc-Shaped Element

To create an arc-shaped element, do the following steps:

1. Choose *Add – 3pt Arc*.
2. Verify the values for *Class*, *Subclass*, *Line Width* and *Line Font* for the arc.
3. Choose the start point of the arc.
4. Choose the end point of the arc.
5. Complete the arc.
You can enter more arcs as required by picking another starting point.
6. When you have entered all arcs required, click right and choose *Done* from the pop-up menu.
7. Pick the start point of the arc, the end point, and a third point that dynamically establishes the radius of the arc, as shown in the example:
8. Click right to display the pop-up and choose *Done* to make the arc permanent, or pick another three points for the next arc.

Example



Related Topics

- [add arc](#)

Changing the Font of Arc or Circle

You can change the line pattern used in creating an arc or a circle.

1. Select the arc or circle and right-click to choose *Change Line Font* command.
2. Choose a new pattern from the list appears.

Related Topics

- [Changing line fonts of Elements](#)
- [add arc](#)
- [Add Arc Command: Options Panel](#)

add circle

The `add circle` command adds a circular element to your design.

Related Topics

- [Adding a Circle](#)

Add Circle Command: Options Panel

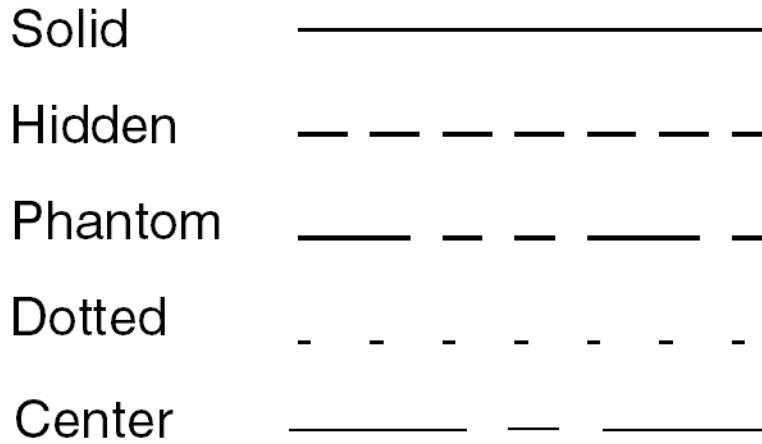
Access using

- Menu path: *Add – Circle*

The following table describes the various fields available in the *Options* panel for the `add circle` command:

Active Class and Subclass	You can add circles to your drawings in the following classes: <ul style="list-style-type: none">• BOARD/SUBSTRATE GEOMETRY• ETCH/CONDUCTOR• PACKAGE/PART GEOMETRY
Line width	Defines the width of the Solid lines used in creating the circle in user units. All other line fonts remain at 0 width.

<i>Line font</i>	Defines the line pattern used in creating the circle. The choices are Solid , Hidden , Phantom , Dotted , and Center . The default is Solid . The line pattern types are:
------------------	--



Line fonts, other than Solid, are allowed on the following Class/Subclasses:

- DRAWING FORMAT/All user defined subclasses
- MANUFACTURING/NCDRILL_LEGEND
- MANUFACTURING/All user defined subclasses
- PACKAGE GEOMETRY/ASSEMBLY_TOP
- PACKAGE GEOMETRY/ASSEMBLY_BOTTOM
- PACKAGE GEOMETRY/All user defined subclasses
- BOARD GEOMETRY/OUTLINE
- BOARD GEOMETRY/ASSEMBLY_NOTES
- BOARD GEOMETRY/ DIMENSIONS
- BOARD GEOMETRY/ASSEMBLY_DETAIL
- BOARD GEOMETRY/All user-defined subclasses

A Commands

A Commands--add circle

<i>Circle Creation</i>	<p>Choose to set circle creation mode.</p> <p><i>Draw Circle</i>: Choose to draw circle.This option is selected by default.</p> <p><i>Place Circle</i>: Choose to place the circle of known radius by setting the <i>Radius</i> value.</p> <p><i>Center/Radius</i>: Choose to create the circle with known center and radius by setting the <i>Radius</i> and <i>Center</i> values in the field below.</p> <p><i>Create</i>: Click to create the circle.</p> <p><i>Radius</i>: Set radius of the circle.</p> <p><i>Center</i>: Set the origin of the circle.</p>
------------------------	--

Adding a Circle

Do the following steps to add a circle:

1. Choose *Add – Circle*.
The following message appears:

Pick center point of circle
2. Verify the Class and Subclass, the Line Width, and Line Font of the circle in the *Options* panel.
3. Choose options in the *Circle Creation* to create the circle.
4. If you select *Draw Circle*, click to specify the center and then click again to specify the perimeter.
The values for the
5. Draw the circle by doing the following steps:
 - a. Specify the center of circle by clicking where you want the circle center.
The coordinates of the center are updated in the *Options* panel.
 - b. Specify the radius of circle by clicking at the needed location.
The value of the radius of the circle is updated in the *Options* panel.
6. Place the circle by doing the following steps:
 - a. Specify the radius of circle in the *Radius* field in the *Options* panel.
 - b. The circle is attached to the cursor.
 - c. Click to place the circle.
The coordinates of the center are updated in the *Options* panel.
 - d. Choose *Create* to add the circle with specified radius.
7. Repeat steps 3 and 4 for each circle.
8. When all circles are complete, right click and choose Done from the pop-up menu.

Related Topics

- [Changing the Font of Arc or Circle](#)
- [add circle](#)


add codesign die

The `add codesign die` command lets you create and add co-design dies using the Add Co-Design Die dialog box.

This command is available only in Allegro X Advanced Package Designer (APD).

You can work in a concurrent or dynamic environment or in a distributed environment.

With the `add codesign die` command, you can also apply scribe lines and an optical shrink to the imported die. For additional information, see *Placing the Elements* in the user guide. You can also view the values for scribe lines and optical shrink on an existing design using the `die properties` command.

 If you have a wire bond co-design, the I/O cells have the bond pad built into them; therefore you cannot use standard flip-chip commands to assign power and ground connections to the bond pads. Since the power and ground pads are not defined in the Verilog netlist, you may add I/O instances for the power and ground with the IOP command `addNewIoInst`, and then you need to run the `globalNetconnect` command to connect the power and ground nets to I/O pads of the newly added instances. For additional information on these commands, see the Encounter documentation.

Related Topics

- [Placing the Elements](#)
- [die properties](#)
- [About Die Shrink](#)
- [Adding an Existing OA Co-Design Die to the Layout Editor](#)
- [Adding a New Co-Design Die Using DEF](#)
- [Adding a New Co-Design Die Using Verilog](#)
- [Adding a New Co-Design Die Using a Die Abstract](#)

Add Co-Design Die Dialog Box

Access Using

- Menu path: *Add – Co-Design Die*

Different tabs appear on the dialog box based on the platform on which you are running.

Existing OA Design

Lets you add an existing co-design die into a database without invoking IOP.

Control	Description
<i>.defs file</i>	<p>Lets you specify an OA library definition file (<code>lib.defs</code>). The OA database for the co-design file must be located or will be created in a library of the selected library definition file. If you type in a library definition file name, then you must specify the full path to the file. Otherwise, it is understood that the file location is relative to the current working directory. The default setting for this field is <code>lib.defs</code>. The layout editor looks for a <code>lib.defs</code> file in the current working directory.</p> <p>The <i>OK</i> button is disabled until you specify an existing library definition file and the layout editor is able to read it successfully. You must have write permission to the library definition file and the directory containing it.</p>
<i>Browse</i>	Lets you browse the file system for an existing OA library definition file. The full path to the selected file is automatically populated in the <i>.defs file</i> field.


A Commands

A Commands--add codesign die

<i>Library name</i>	<p>Lets you select the OA library from the currently selected library definition file. This is the location from which an existing co-design die will be read.</p> <p>Only those libraries in the current library definition file that contain layout cell views written by IOP are listed in this pull-down menu. If there are no libraries listed, then the <i>Library</i>, <i>Cell Name</i>, and <i>View Name</i> fields, as well the <i>OK</i> button, are disabled.</p> <p>The default setting is the first library listed in the library definition file.</p>
<i>Cell name</i>	<p>Lets you specify the name of an OA cell that is read from the currently selected library.</p> <p>The pull-down menu contains only cells with layout views written by IOP. If these cell views are not in the current library, then the <i>Cell name</i> and <i>View name</i> fields, as well as the <i>OK</i> button, are disabled. Otherwise, the field defaults to the first cell of type layout in the currently selected library.</p>
<i>View name</i>	<p>Lets you specify the name of the OA view that is read in the specified cell and from which the co-design die will be read.</p> <p>The pull-down menu contains all views of the currently selected cell that are of the type layout and that are written by IOP. The default value is the first view of this type. If there are no views of this type, the field, as well as the <i>OK</i> button, are disabled.</p>

New design from DEF

Lets you add a co-design die by specifying a DEF file.

 This tab does not appear on the Windows platform.


Control	Description
<i>.DEF file to load</i>	<p><i>Specifies</i> the name of a DEF file to open in IOP. When IOP starts up, it loads the DEF file, which creates the netlist for the design and then loads the place and route information.</p> <p>If you specify the absolute path to the DEF file, the layout editor searches that path to locate the file. If you do not specify an absolute path, then the layout editor searches for the DEF file relative to the current working directory.</p> <p>If you do not specify a DEF file to load, IOP starts up empty with no logic or floor plan information loaded.</p> <p>The default setting for this field is empty,</p>
<i>Browse</i>	Lets you browse the file system for an existing DEF file. The full path to the selected file is automatically populated in the <i>.DEF file to load</i> field.
<i>.defs file</i>	<p>Lets you specify an OA library definition file (<i>lib.defs</i>). The OA database for the co-design file must be located or will be created in a library of the selected library definition file. If you type in a library definition file name, then you must specify the full path to the file. Otherwise, it is understood that the file location is relative to the current working directory. The default setting for this field is <i>lib.defs</i>. The layout editor looks for a <i>lib.defs</i> file in the current working directory.</p> <p>The <i>OK</i> button is disabled until you specify an existing library definition file and the tool is able to read it successfully. You must have write permission to the library definition file and the directory containing it.</p>
<i>Browse</i>	Lets you browse the file system for an existing OA library definition file. The full path to the selected file is automatically populated in the <i>.defs file</i> field.
<i>Library name</i>	<p>Lets you select the OA library from the currently selected library definition file. This is the location into which a new co-design database is written.</p> <p>Only those libraries in the current library definition file that contain layout cell views written by IOP are listed in this pull-down menu. If there are no libraries listed, then the <i>Library</i>, <i>Cell Name</i>, and <i>View Name</i> fields, as well the <i>OK</i> button, are disabled.</p> <p>The default setting is the first library listed in the library definition file.</p>
<i>Cell name</i>	<p>Lets you specify the name of an OA cell that is created in the currently selected library.</p> <p>The pull-down menu contains only cells with layout views written by IOP. If these cell views are not in the current library, then the <i>Cell name</i> and <i>View name</i> fields, as well as the <i>OK</i> button, are disabled. Otherwise, the field defaults to the first cell of type layout in the currently selected library.</p>
<i>View name</i>	<p>Lets you specify the name of the OA view that is created in the specified cell and from which the co-design die will be read.</p> <p>The pull-down menu contains all views of the currently selected cell that are of the type layout and that are written by IOP. The default value is the first view of this type. If there are no views of this type, the field, as well as the <i>OK</i> button, are disabled.</p>

New design from Verilog

A Commands

A Commands--add codesign die

Lets you add a co-design die by specifying a Verilog file.

 This tab does not appear on the Windows platform.

Control	Description
New design from Verilog	
<i>.v file to load</i>	Lets you add a co-design die by specifying a Verilog file, which contains netlist information.
<i>.defs file</i>	<p>Lets you specify an OA library definition file (<code>lib.defs</code>). The OA database for the co-design file must be located or will be created in a library of the selected library definition file. If you type in a library definition file name, then you must specify the full path to the file. Otherwise, it is understood that the file location is relative to the current working directory. The default setting for this field is <code>lib.defs</code>. The layout editor looks for a <code>lib.defs</code> file in the current working directory.</p> <p>The OK button is disabled until you specify an existing library definition file and the tool is able to read it successfully. You must have write permission to the library definition file and the directory containing it.</p>
<i>Browse</i>	Lets you browse the file system for an existing OA library definition file. The full path to the selected file is automatically populated in the <i>.defs file</i> field.
<i>Library name</i>	<p>Lets you select the OA library from the currently selected library definition file. This is the location into which a new co-design database is written.</p> <p>Only those libraries in the current library definition file that contain layout cell views written by IOP are listed in this pull-down menu. If there are no libraries listed, then the <i>Library</i>, <i>Cell Name</i>, and <i>View Name</i> fields, as well the OK button, are disabled.</p> <p>The default setting is the first library listed in the library definition file.</p>
<i>Cell name</i>	<p>Lets you specify the name of an OA cell that is created in the currently selected library.</p> <p>The pull-down menu contains only cells with layout views written by IOP. If these cell views are not in the current library, then the <i>Cell name</i> and <i>View name</i> fields, as well as the OK button, are disabled. Otherwise, the field defaults to the first cell of type layout in the currently selected library.</p>
<i>View name</i>	<p>Lets you specify the name of the OA view that is created in the specified cell and from which the co-design die will be read.</p> <p>The pull-down menu contains all views of the currently selected cell that are of the type layout and that are written by IOP. The default value is the first view of this type. If there are no views of this type, the field, as well as the OK button, are disabled.</p>
<i>Power nets</i>	Lets you type in a list of power nets. These power net names are user-generated names, separated by spaces and are available for assignment when editing the die in IOP.
<i>Ground nets</i>	Lets you type in a list of ground nets. These ground net names are user-generated names, separated by spaces and are available for assignment when editing the die in IOP.
New design from Abstract	Completing the information in this tab lets you use a die abstract file to generate your co-design.
<i>Die abstract file to load</i>	Specifies the path and name of the die abstract to be used as the source for the co-design die. Using the <i>Browse</i> button will override information in this field.
<i>Design Name</i>	Specifies the name of the component that is read from the die abstract file. This field is read-only.
OK	<p>Clicking this button either displays the IOP window for new dies or imports the existing co-design. This button is disabled until you complete the appropriate settings.</p> <p>The button is also disabled if you have not configured a Library Manager.</p> <p>If you select an OA design that is already used for a co-design die in the package, an error message appears because currently you cannot have multiple instances of a co-design die in a package.</p>
<i>Cancel</i>	Exits the command without making any changes.

A Commands

A Commands--add codesign die

<i>Library Manager</i>	Opens the Library Manager. You can also perform this operation using the <i>Setup – LEF Libraries</i> command.
<i>Help</i>	Displays help for this dialog box.

Cadence I/O Planner Window

The Cadence I/O Planner (IOP) Window, an IC layout editor, automatically appears when you create or edit a co-design die. You can actively plan the die down to the I/O buffer level concurrently with the package design in which it will be placed.

For additional information on the Cadence I/O Planner, see *the First Encounter documentation*.


Place Co-Design Die Dialog Box

The Place Co-Design Die dialog box automatically appears when you add an existing co-design die to the package or after you execute the `updatePackage` command from IOP for the first time. With this dialog box, you can specify the details of the new die component and symbol that you are adding to your package database.

Die Logic	
<i>Ref Des</i>	Specifies the unique reference designator for the new die. The default value is the name of the OA view where the co-design IC design is stored.
<i>Import net assignments</i>	Check this box to have the IC net names assigned to the I/O pads read into the layout design as a starting netlist for the package. Leave the box unchecked to have the die pads brought into the design unassigned (on dummy nets). The default setting is checked, which means IC net names are brought in as a starting netlist.
<i>Die Attachment</i>	Specifies the die attachment type: <i>Wire Bond</i> or <i>Flip Chip</i> .
<i>Wire Bond</i>	Indicates that this die will be attached with wire bonds to the package substrate. Therefore, the connection points for the die are on the die side opposite the package substrate. Choosing this option automatically changes the chip orientation to <i>Chip Up</i> .
<i>Flip Chip</i>	Indicates that this die will be mounted as a flip chip. Therefore, the die pins directly touch (and are soldered to) the package/die below it. Choosing this option automatically changes the chip orientation to <i>Chip Down</i> . <i>Flip Chip</i> is the default attachment type for a co-design die.
<i>Chip Up</i>	Specifies that the die will be placed unmirrored. A wire bond is placed on the top side of the substrate. A flip-chip is placed on the bottom side of the substrate.
<i>Chip Down</i>	Specifies that the chip will be mounted mirrored. A flip-chip is mounted on the top of the package substrate. A wire bond is mounted on the bottom side of the BGA. This is the default setting (Flip Chip, Chip Down).
Die Placement	
<i>Pin Layer</i>	Specifies the layer on which this die pin pads are created. The default setting is the first layer in the design.
<i>Origin X/Y</i>	Specifies the X/Y coordinate value at which to place the die origin in the layout design. The default setting is 0. The origin is the center of the design (0, 0).
<i>Rotation</i>	Specifies the angle of rotation at which to place the die. By default, this is 0 (unrotated). Valid choices are 0, 90, 180, and 270 degrees.
<i>Apply IC Fabrication Optical Shrink</i>	Check this box to shrink the die. The optical shrink is applied with respect to the 0, 0 location of the IC in the IOP design. The origin location is not impacted by applying a shrink. The default setting is <i>Off</i> .
<i>%</i>	Enter a positive value (1 - 100) in the text box to indicate the percentage by which the original die size should be shrunk. For example, a 10% shrink means that the resulting die will only be 90% of the size of the original die. The default setting of this field is 0%, which means that no shrink will be applied and that the die symbol will be the size of the original die.

A Commands

A Commands--add codesign die

<i>Add Scribe Width</i>	<p>Check this box to add scribe line information to the specified die. The default setting is <i>Off</i>.</p> <p>Since the scribe width is applied to the outside of the die, it has no impact on the location of the die origin. If (0, 0) is the lower-left corner of the die, then a scribe width applied to the west and south sides of the die goes to the outside of this, and the origin stays at the original IC (0, 0) location. Thus the lower-left corner of the boundary of the physical die defined by the inclusion of the scribe widths will have negative coordinates.</p> <div> For placement purposes, the extents are fixed per side and rotate when you change the orientation of the die. For example, if you rotate the die ninety degrees clockwise, the north side now exists on the east side of the representation.</div>
<i>North</i>	Enter a value to indicate the amount that the actual physical die is larger than the represented extents on the North side of the die.
<i>South</i>	Enter a value to indicate the amount that the actual physical die is larger than the represented extents on the South side of the die.
<i>East</i>	Enter a value to indicate the amount that the actual physical die is larger than the represented extents on the East side of the die.
<i>West</i>	Enter a value to indicate the amount that the actual physical die is larger than the represented extents on the West side of the die.
<i>OK</i>	Places the die as specified (or as picked using the mouse) and exits the command.
<i>Cancel</i>	Cancels from the command and does not place the co-design die.
<i>Place</i>	Places the die as specified. Using the mouse to pick in the Design Window is the same as clicking <i>Place</i> , except that it takes the X, Y location from the cursor.
<i>Help</i>	Display helps on the command.

Related Topics

- [Adding a New Co-Design Die Using DEF](#)
- [Adding a New Co-Design Die Using Verilog](#)
- [Adding a New Co-Design Die Using a Die Abstract](#)

Adding an Existing OA Co-Design Die to the Layout Editor

To add an existing co-design die to a package do the following steps:

1. Choose *Add – Co-Design Die*.
The Add Co-Design Die dialog box appears.
2. Click the *Existing OA design* tab.
3. Type in the library definition file in the *.defs file* field.
Typically, this is the `lib.defs` file in the current working directory. You must have write permission to this file.
4. From the pull-down menu in the *Library Name* field, select the OpenAccess library from which the IC layout for the co-design will be read.
You must have write permission to this library.
5. From the pull-down menu in the *Cell Name* field, select the cell from the selected OA library for the co-design IC.
6. From the pull-down menu in the *View Name* field, select the view of the selected cell that contains the IC layout for the co-design die, and click *OK*.
If IOP did not write the library/cell/view, the co-design die does not work correctly.
The Place Co-Design Die dialog box appears.
The die footprint also appears on the cursor in the Design Window in preparation for placement.

⚠ If the IC design does not contain any physical die pins or does not yet have a die area, no graphical representation appears on the cursor. Instead, it may be represented as a box outlining the die area. However, you still need to complete the parameters in this dialog box.

7. Specify the reference designator for the co-design die.
8. Specify the orientation, location and rotation for the co-design die.
9. Place the die on the substrate in the Design Window, or type explicit x, y coordinates in the console window.
10. Click *OK* to accept the placement and import the IC data from OA and add an instance of the die to the package as a co-design die.
If the import is successful, the die is added to the package according to the placement parameters specified. IOP is not launched because at this time, the existing die is added to the package from OA. You are not making any changes to the die at this time.
If you select an OA design that already exists in the package, an error message appears because currently you cannot have multiple instances of a co-design die in a package.
11. Save the design in APD.
Although a save is not mandatory at this time, it is a good practice to save the design now.
12. If you are using System Connectivity Manager (SCM) for the logic design, update the SCM design. Choose *File – Export – Logic*, click *Design Entry HDL*, and click *Export Cadence*.
You now can backannotate the addition of the new die to SCM using the resulting output from the Export Logic command.
Test probe pins are imported as pads on the appropriate `Probe_Top` or `Probe_Bottom` subclass.

⚠ The IOP Window does not appear during this process. To edit this die, you can run the `die editor` command after placing the new instance in the package. Then the IOP Window appears.

Related Topics

- [die properties](#)
- [add codesign die](#)
- [Adding a New Co-Design Die Using Verilog](#)
- [Adding a New Co-Design Die Using a Die Abstract](#)

Adding a New Co-Design Die Using DEF

You can add a new co-design die by specifying a DEF file or Verilog file (UNIX) or by importing a die abstract file.

To add a new co-design die using DEF, do the following:

1. Choose *Add – Co-Design Die*.
The Add Co-Design Die dialog box appears.
The number of tabs on the box depends on the platform on which you are running.
2. Click the *New design from DEF* tab.
3. Browse to find the DEF file to load.
IOP opens it to load logic and any existing layout to start the new IC design. If you do not specify a DEF file to load, IOP starts up empty with no logic or floor plan information loaded.
4. Select the library definition file to use, normally `lib.defs` in the current working directory.
If the library definition file specified does not exist, the layout editor creates it. You must have write permission to the library definition file and the directory containing it.
5. In the *Library Name* field, select or type in the name of the OA library into which the IC layout for the co-design will be written.
You must have write permission to this library. If the library does not exist, the layout editor creates a new one.
6. In the *Cell Name* field, type in the name of the new cell for the OA library into which the new co-design IC design will be written. This cell must not already exist.
7. In the *View Name* field, type in the view name (normally "layout") for the new OA cell into which the IC layout design for the co-design die will be written.
8. Click *OK*.
The IOP window opens. Using the capabilities of IOP, load the netlist and create or import the I/O floor plan and die pads/bumps for the IC. For additional information, see the First Encounter documentation.
9. When you have successfully created the die in IOP, use the Cadence I/O Planner *updatePackage* command (or update and exit).
IOP saves the new IC layout to a temporary OpenAccess library/cell/view using its *Save OA Design* capability. Then IOP sends a message to the layout editor to instruct it to import the data from OA and prepare the new die representation for placement in the package.
You are prompted whether to import the nets from the OA design or keep the existing net assignments.
The layout editor automatically reads the temporary database to get the new component and symbol definition information for the die.
The Place Co-Design Die dialog box appears. The die footprint also appears on the cursor in the Design Window in preparation for placement.

⚠ If the IC design does not contain any physical die pins or have a die area yet, there will be no graphical representation on the cursor; it may be represented as a box outlining the die area. However, you still need to complete the parameters in this dialog box.
10. Specify the reference designator for the co-design die, and specify the orientation, location and rotation for the co-design die.
The die footprint appears on the cursor, so you can place it onto the substrate in the Design Window, or type explicit x, y coordinates.
11. Click *OK* to accept the placement and import the IC data from OpenAccess and add an instance of it to the package as a co-design die.
The existing instance of the die in the database is replaced by an instance of the modified component/symbol using the same reference designator as the old instance and placed at the same location, orientation and rotation. Net assignments are propagated from IOP to the layout editor as logical pin name to physical pin number assignment changes. The logical pin name is matched with IOP's OA terminal name and ensures that logical pin name is assigned to the physical pin corresponding to that OA terminal name's assignment in IOP.
12. Once the `add codesign die` command worked successfully, save the layout design using the *File – Save* command.
13. If you are using SCM for the logic design, to update the SCM design, choose *File – Export – Logic*, click *Design Entry HDL*, and click *Export Cadence*.
You now can backannotate the addition of the new die to SCM using the resulting output from the *Export Logic* command.

Related Topics


- [Placing the Elements](#)
- [die properties](#)
- [add codesign die](#)
- [Add Co-Design Die Dialog Box](#)
- [Adding a New Co-Design Die Using a Die Abstract](#)

Adding a New Co-Design Die Using Verilog

You can add a new co-design die by specifying a DEF file or Verilog file (UNIX) or by importing a die abstract file.

To add a new co-design die using a Verilog file, do the following:

1. Choose *Add – Co-Design Die*.
The Add Co-Design Die dialog box appears. The number of tabs on the box depends on the platform on which you are running.
2. Click the *New design from Verilog* tab.
3. Browse to find the Verilog file to load.
4. Select the library definition file to use, normally `lib.defs` in the current working directory.
If the library definition file specified does not exist, the layout editor creates it. You must have write permission to the library definition file and the directory containing it.
5. In the *Library Name* field, select or type in the name of the OA library into which the IC layout for the co-design will be written.
You must have write permission to this library. If the library does not exist, the layout editor creates a new one.
6. In the *Cell Name* field, type in the name of the new cell for the OA library into which the new co-design IC design will be written. This cell must not already exist.
7. In the *View Name* field, type in the view name (normally *layout*) for the new OA cell into which the IC layout design for the co-design die will be written.
8. Click *OK*.
The IOP window opens. Using the capabilities of IOP, create or import the I/O floor plan and die pads/bumps for the IC. For additional information, see the First Encounter documentation.
9. When you have successfully created the die in IOP, use the Cadence I/O Planner *updatePackage* command (or *update* and *exit*).
IOP saves the new IC layout to a temporary OpenAccess library/cell/view using its *Save OA Design* capability. Then IOP sends a message to the layout editor to instruct it to import the data from OA and prepare the new die representation for placement in the package.
You are prompted whether to import the nets from the OA design or keep the existing net assignments.
The layout editor automatically reads the temporary database to get the new component and symbol definition information for the die. The Place Co-Design Die dialog box appears. The die footprint also appears on the cursor in the Design Window in preparation for placement.

 If the IC design does not contain any physical die pins or have a die area yet, there will be no graphical representation on the cursor; it may be represented as a box outlining the die area. However, you still need to complete the parameters in this dialog box.
10. Specify the reference designator for the co-design die, and specify the orientation, location and rotation for the co-design die.
The die footprint appears on the cursor, so you can place it onto the substrate in the Design Window, or type explicit x, y coordinates.
11. Click *OK* to accept the placement and import the IC data from OpenAccess and add an instance of it to the package as a co-design die.
The existing instance of the die in the database is replaced by an instance of the modified component/symbol using the same reference designator as the old instance and placed at the same location, orientation and rotation. Net assignments are propagated from IOP to the layout editor as logical pin name to physical pin number assignment changes. The logical pin name is matched with IOP's OA terminal name and ensures that logical pin name is assigned to the physical pin corresponding to that OA terminal name's assignment in IOP.
12. Once the `add codesign die` command worked successfully, save the layout design using the *File – Save* command.
13. If you are using SCM for the logic design, to update the SCM design, choose *File – Export – Logic*, click *Design Entry HDL*, and click *Export Cadence*.
You now can backannotate the addition of the new die to SCM using the resulting output from the *Export Logic* command.

Related Topics

- [die properties](#)
- [add codesign die](#)
- [Add Co-Design Die Dialog Box](#)
- [Adding an Existing OA Co-Design Die to the Layout Editor](#)

Adding a New Co-Design Die Using a Die Abstract

You can add a new co-design die by specifying a DEF file or Verilog file (UNIX) or by importing a die abstract file.

To add a new co-design die using a die abstract file, do the following:

1. Choose *Add – Co-Design Die*.
The Add Co-Design Die dialog box appears.
The Add Co-Design Die dialog box appears. The number of tabs on the box depends on the platform on which you are running.
2. Click the *New design from Abstract* tab.
3. Click *Library Manager* to set up your LEF files.
The LEF Library Manager dialog box appears.
4. When finished setting up the LEF Library Manager, click *OK* in the LEF Library Manager dialog box.
5. Type the name of the file and path in the *Die abstract file to load* field or click *Browse* to point to a die abstract file.
The design name (read from the die abstract file becomes the component name) appears in the *Design Name* field.
6. Click *OK* to add the co-design die to the design.
An image of the die appears on the cursor and the Place Co-Design Die dialog box appears. You can change parameters in this box.
7. Click *Place* to add a new co-design die to the design, then click *OK* in the dialog box, or double-click in the Design Window, and then click *OK* in the dialog box.
The layout editor adds the co-design die to the design only if the die design name (equivalent to component name) does not already exist in the current database.
When using the `show element` command, you can see if the die is a co-design die and also whether it is distributed or concurrent.

Related Topics

- [die properties](#)
- [add codesign die](#)
- [Add Co-Design Die Dialog Box](#)
- [Adding an Existing OA Co-Design Die to the Layout Editor](#)
- [Adding a New Co-Design Die Using DEF](#)
- [lef lib](#)

add codesign pkg

Internal command.

add connect

The `add connect` command lets you interactively route a single connection as well as differential pairs. When you window select a group of elements, the command can also be used as an interactive group router. Push and shove controls aggressively shift adjacent traces to clear a path during command execution. The `add connect` command is aligned with the DRC system. On high-speed designs, graphical timing/length feedback is provided on nets with electrical rules.

This command functions in a pre-selection use model, in which you choose an element first, then right-click and execute the command. In Etch Edit application mode, single clicking on an element launches the command by default. When you execute the command from the right mouse button pop-up menu, the point at which the connection begins is from the location at which you right-clicked.

Elements ineligible for use with the command generate a warning and are ignored. Valid elements for which you may initiate a connection are:

- Arc segments
- Filled Rectangles
- Cline segments
- Vias
- Pins
- Shapes
- Rat Ts
- Ratsnests

For Ratsnest, the closest point determines the source. When you choose multiple elements and no Ratsnest exists between them, a connection occurs from each of them.

If you choose an element, execute `add connect`, and then choose *Oops*, the command terminates, returning to the location of the last click.

In addition to setting parameters relevant for this command on the *Options* panel, you may also set them by right-clicking to display the pop-up menu from which you may choose:

- Design Parameters to access the Design Parameter Editor
- Options

Changing a parameter using either of these pop-up menu choices automatically updates the *Options* panel as well.


Before adding connections, you should familiarize yourself with the various aspects of interactive routing as described in the Routing the Design user guide in your documentation set.

Related Topics

- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Add Connect Command: Options Panel

Access Using

- Menu path:
 - Layout edit mode: *Route – Connect*
 - Symbol edit mode: *Layout – Connections*
- Toolbar icon: 


✓ The Design Parameter Editor is also available for editing the parameters listed on the *Options* panel. Use *Setup – Design Parameters* (prmed command) to access the Design Parameter Editor or right-click whenever you are working in the pre-selection use model and choose Design Parameters.

The following table describes the fields in the panel:

<i>Act</i>	<p>The <i>Act</i> (active subclass) drop-down list box displays the current value and provides choices for modifying the value. While actively routing a net, you can change the value, if you are routing from a via or multi-layer pin.</p> <p>If layer-set constraints exist for the net, the layers (also referred to as subclasses) in the legal layer sets appear in bold-faced type. For more information about interactive routing with layer-set constraints, see <i>Routing the Design</i> in the user guide.</p>
<i>Alt</i>	<p>The <i>Alt</i> (alternate subclass) drop-down list box displays the current value and provides choices for modifying the value.</p> <p>The alternate layer becomes the active layer when you right-click within the design and choose <i>Layers</i> or <i>Add Via</i> from the pop-up menu when an element is active. If no element is active or the active element also exists on an alternate subclass, <i>Layers</i> lets you alternate active and alternate layers.</p> <p>If layer-set constraints exist, the layer set appear in bold-faced type.</p>
<i>WL</i>	<p>The <i>Working Layers</i> mode is designed to accommodate HDI designs (though you can use it on any layout). When you choose this selection, the <i>Working Layers</i> dialog box displays all the etch/conductor layers in the current design and is used to control the layers that appear in the <i>Via Popup</i> GUI. Instead of being confined to routing from an active layer to a single alternate layer, a double-click in this mode launches a pop-up GUI with all working layers available for selection. A single pick on any of the layers resumes routing on that respective layer. When routing to HDI Rules, you can automatically add stacked vias or semi-automatically add staggered vias across multiple layers. Additionally, you do not need to continually navigate from your design to the <i>Options</i> panel in order to select individual vias for each layer.</p>
<i>Via</i>	<p>Lists the available via padstacks between the active and alternate subclasses. You must have already defined the padstacks in the constraint set via list.</p>
<i>VS</i>	<p>Lists the available via structures between the active and alternate subclasses. You must have already defined the via structures in the Electrical constraint set via structures list.</p>
<i>Net</i>	<p>Identifies the net assigned to the element you select. If no net is assigned, the value is NULL NET.</p> <p>To the left of the field name is an indicator for the nets. When you are routing a single net, the indicator shows one net. When you are performing differential pair routing, the indicator shows two nets. If you are performing group routing, the indicator shows multiple nets. If you are in single trace mode in differential pair routing or group routing, the indicator shows only the control trace highlighted. The field always shows the net name of the control trace (for both differential pairs or single traces) and never the differential pair name.</p>
<i>Line lock</i>	<p>Defines the connection as a line or an arc, and specifies the connect lines' corner angle when they change direction. The values default from the <i>Design</i> tab of the Design Parameter Editor, available by choosing <i>Setup – Design Parameters</i>. Changing the values here changes them in the Design Parameter Editor as well.</p> <p><i>Off</i> implies that any-angle routing is allowed. Setting this field to <i>Arc</i> disables <i>Optimize in channel</i>.</p>
<i>Route offset</i>	<p>Choose to enable the route offset angle. By default this option is <i>Off</i>. You can specify the route offset angle in the value field. The default angle is <i>10 degrees</i>.</p> <p>This option allows routing at angles that are offset from multiple of 45 degrees (0, 45, 90, 135...). Routes are snapped to an angle, that is +- the offset angle, from the 45 degree incremental angle. This option works only when <i>Line lock</i> value is <i>Line/45</i>.</p> <p>If set, the <i>Optimize in channel</i> option is disabled.</p>

A Commands

A Commands--add connect

<i>Miter</i>	<p>Defines the value for the Miter size. In the editor, the miter size is the amount that is cut away from the pre-cornered segments (when cornering orthogonal segments, this is the x or y offset between the endpoints of the new segment). This field appears only if the Line lock setting is <i>Line</i> and the angle setting is <i>45</i>.</p> <p>The <i>Miter</i> field also accepts values for a corner size that is relative to the current line width. In addition to typing in a number, you can also type in a value in the format <i><n>x</i> to get n times the line width. For example, to obtain a corner size of 3 times the line width, type in <i>3x</i>. With this setting, the corner size varies with the line width.</p> <p>Note: If you type in a value of 5, it results in a corner segment length of 7.1. In general, the segment length is about the square root of 2 times the miter value, for example 1.414 times the miter width.</p> <p>To the right of the Miter field is a drop-down list with these choices:</p> <p>Min: When you set the value to <i>Min</i>, the value you entered in the Miter field is the minimum that the editor allows.</p> <p>Fixed: When you set the value to <i>Fixed</i>, the editor uses the value you entered in the <i>Miter</i> field.</p> <p>For concurrent routing of differential pairs, this <i>Min</i> or <i>Fixed</i> value applies to the inside (smaller) corner of the trace.</p>
<i>Radius</i>	<p>Defines the value for the radius size. This field appears only if the Line lock setting is <i>Arc</i> with an angle of <i>45</i> or <i>90</i> degrees. The <i>Radius</i> field also accepts values that are relative to the current line width. In addition to typing in a number, you can also type in a value in the format <i><n>x</i> to get n times the line width. For example, to obtain a radius size of 3 times the line width, type in <i>3x</i>. With this setting, the radius size varies with the line width.</p> <p>To the right of the Radius field is a drop-down list with these choices:</p> <p>Min: When you set the value to <i>Min</i>, the value you entered in the Radius field is the minimum that the editor allows.</p> <p>Fixed: When you set the value to <i>Fixed</i>, the editor uses the value you entered in the Radius field.</p> <p>For concurrent routing of differential pairs, this <i>Min</i> or <i>Fixed</i> value applies to the inside (smaller) corner of the trace.</p>
<i>Line width</i>	<p>Defines the width of the line in units. The value defaults from the Physical (Lines/Vias) Rule Set. DRC uses this value to compare against the design rule set for the net and flags any violations in the design. For more information about defining the line width, see <i>Routing the Design</i> in the user guide.</p> <p>This field shows up to 16 previous values that were set in the drop-down menu. The most recently used line widths top the list. If the current value in the field is not the default value (the minimum line width) the drop-down list shows an item called <i>Default</i>. Choosing this item resets the line width so that the layout editor uses the minimum line width from the applicable physical constraint set. This feature replaces the <i>Reset</i> button. For information about overriding the minimum line width, see <i>Routing the Design</i> in the user guide.</p> <div style="border: 1px solid #f0e68c; padding: 5px; margin-top: 10px;"> <p> To delete the line width value in the drop-down list, select the line width value, delete and press <i>Enter</i>. Click <i>Yes</i> to confirm the deletion. You can only delete user added values.</p> </div>
<i>Bubble</i>	<p>Controls any automatic bubbling (moving of existing connections) to resolve DRC errors. Enabling either of the hug modes or shove-preferred bubble mode sets the <i>Line lock</i> field to <i>Line</i> to prevent you from adding arcs while in shove- or hug-preferred mode. Bubble mode does not support arcs.</p> <p>These are the choices:</p> <p>Off: The clines you route start at the location you indicate and no bubbling occurs. DRC flags all clearance violations with error markers. Disables <i>Optimize in channel</i> option.</p> <p>Hug Only: Where possible, the routed cline contours around other etch/conductor objects to avoid spacing DRCs. Other etch/conductor remains unchanged.</p> <p>Hug Preferred: Where possible, the routed cline contours around other etch/conductor objects to avoid spacing DRCs. If not possible, the layout editor tries shoving other etch/conductor objects to open routing paths.</p> <p>Note: This method is more aggressive than <i>Hug Only</i>.</p> <p>Shove Preferred: Where possible, the routed cline pushes and shoves other etch/conductor objects to avoid spacing DRCs. If not possible, the layout editor tries hugging other etch/conductor objects.</p>
<i>Shove vias</i>	<p>Allows the bubble functionality in shove mode to move vias when you are editing etch/conductor. It is only active when <i>Bubble</i> is enabled.</p> <p>These are the choices:</p> <p>Full: Vias are shoved in a shove-preferred manner. Any new or edited etch/conductor always shoves vias out of the way.</p> <p>Minimal: Vias are shoved in a hug-preferred manner. Vias are not moved unless there is no way to draw a connect line around them.</p> <p>Off: Vias are not shoved</p>

A Commands

A Commands--add connect

<i>Gridless</i>	<p>Specifies that the etch/conductor can go off the routing grid. Gridless routing lets the layout editor add connections at maximum density while accommodating varying design rules and line widths. The DRC minimum space separates objects.</p> <p>When <i>Bubble</i> is disabled, the <i>Gridless</i> field controls the removal of a small segment at the end of the new route when in <code>add connect</code> mode. Normally, if the last segment is small, the editor does not add it (to avoid adding a little jog). If <i>Gridless</i> is off, the editor adds the segment.</p>
<i>Clip dangling clines</i>	<p>Active for shove-preferred mode and controls whether the layout editor clips back dangling clines to fix DRC errors. When disabled, the dangling cline endpoints remain unchanged, and the layout editor corrects the DRC errors, if possible, by bubbling the new cline around the dangling endpoints (similar to hug-preferred mode).</p>
<i>Smooth</i>	<p>Active when you set the <i>Bubble</i> field to hug- or shove-preferred mode and controls whether smoothing occurs on the cline to minimize segments between the start and finish points. Smoothing occurs dynamically as you move the mouse on cline segments close to the segment you selected.</p> <p>Performance with the <i>Smooth</i> option active may be somewhat slower than when it is inactive.</p> <p>These are the choices:</p> <p><i>Minimal</i>: Executes dynamic smoothing to minimize unnecessary segments.</p> <p><i>Full</i>: Executes more extensive smoothing to remove any unnecessary jogs.</p> <p><i>Super</i>: For APD only. Removes unnecessary vertexes for the entire cline, as well as the shoved traces, during routing or sliding.</p> <p>⚠ With any of the <i>Smooth</i> options, if you attach a FIXED property to a cline, there is no smoothing on the specified cline.</p> <p><i>Off</i>: Disables smoothing.</p> <p>⚠ Full smoothing does not smooth the cline you are adding back to its source. Rather, it smooths the newly created etch/conductor back to your last pick. Additionally, parts of other clines that are shoved during this procedure may also be smoothed.</p>
<i>Snap to connect point</i>	<p>Specifies whether the connection snaps to the connect point if it is close to a target element.</p>
<i>Replace etch</i>	<p>Changes the path of an existing trace without extra delete and add steps. Add a loop into an existing trace and <code>add connect</code> recognizes the older portion of the loop and automatically deletes it.</p>
Auto-blank other rats	<p>Hides all ratsnest during interactive routing.</p>
Optimize in channel	<p>Centers new and existing clines within a channel formed between two pads (pins or vias). The size of this channel is the maximum distance between the two pad edges.</p> <p>Clines are centered based on the value of channel size (air gap) that can be set through the <i>Options</i> button. By default, the value of <i>Channel Air Gap</i> is set to ten times of the minimum line width for that layer.</p> <p>⚠ Optimization in channels does not effect differential pairs gathering and coupling.</p>
Clearance View (use Ctrl-Tab to toggle)	<p>Provides a visual clue by generating polygons around objects in a channel to show the amount of space available for routing in channels. The space is calculated using the spacing and the line width constraints and depends on the modes of operation:</p> <ul style="list-style-type: none"> • <i>Spacing</i>: The space is determined by the spacing constraint between the cline and the object. • <i>Channel</i>: The space is determined by the spacing constraint between the cline and the object plus half the width of the cline being routed. <p>⚠ For differential pairs, spacing in <i>Channel</i> mode is the sum of three values: the spacing constraint between cline and the other objects, line width of differential pair, and half the air gap between differential pair being routed.</p> <p>⚠ When in <i>Channel</i> mode and adjacent polygons are not touching routing is possible without creating a DRC.</p> <p>The Ctrl-Tab keys toggles between on and off state.</p> <p>This option is enabled only for single trace mode and grayed out if Multi-Line Route or Group Routing option is selected.</p>

Related Topics

- [prmed](#)
- [Adding Vias Using the Working Layers Mode](#)
- [Add Connect Command: Tasks](#)

Add Connect Command: Pop-Up Menu Options

When you are in `add connect`, right-click in your design canvas to display the pop-up menu. The pop-up menu and items appearing in it differ slightly if you are routing differential pairs or groups, or working in an application mode with the pre-selection use model.

The following table describes the menu items and differences:

Item	Description
<i>Done</i>	Commits the current route and returns the editor to the idle state.
<i>Oops</i>	Reverses the action of the last pick.
<i>Cancel</i>	Reverses results of the current route and returns the editor to the idle state.
<i>Next</i>	Commits the current route and lets you choose another element to begin a new route. Only available in the verb-noun use model.
Persistent select	Specifies the selection mode (<i>Select by Polygon</i> , <i>Select by Lasso</i> , and <i>Select on Path</i>) for selecting multiple objects. By default, the <i>Persistent select</i> option is off. The active persistent select mode remains unchanged and applies to all the commands that access selection modes using right-click pop-up menus.
Select by Polygon	Lets you select the multiple items to route at one time by creating a polygon.
Select by Lasso	Lets you select the multiple items to route at one time by creating a free-form polygon. Objects that are partially or completely contained within that boundary that matches the find filter settings is selected.
Select on Path	Lets you select the multiple items to route at one time by creating a free-form line. Any object touching the line is selected, also matching find filter settings.
Temp Group	Allows you to create a window around multiple items for selection. Only available in the verb-noun use model.
<i>Reject</i>	Reverses the current selection and lets you choose another element from those that are near the selection pick location. Only available in the verb-noun use model.
<i>Add Jumper</i>	Adds a jumper on the currently chosen alternate layer and provides a list of all the jumpers. The jumpers that are defined in the PSMPATH are displayed in bold lettering. The jumpers that are not defined in the PSMPATH are disabled.
<i>Add Via</i>	Locates the via currently chosen in the <i>Via</i> field on the <i>Options</i> panel on the currently chosen alternate layer.
<i>Add Via Structure</i>	Locates the via structure currently chosen in the <i>VS</i> field on the <i>Options</i> panel on the currently chosen alternate layer.
<i>Via Pattern</i>	Only available when you are routing differential pairs. It lets you change the pattern and spacing of the via. Choices for via pattern are: <i>Next</i> , <i>Horizontal</i> , <i>Vertical</i> , <i>Diagonal Up</i> , <i>Diagonal Down</i> , and <i>Spacing</i> . You can also use the <code>pop viapattern</code> command.
<i>Change Active Layer</i>	Displays the current active layers and provides choices for modifying it. Only layers that you made visible in the Visibility panel. While actively routing a net, you can change the value, if you are routing from a via or multi-layer pin.
<i>Change Alternate Layer</i>	Displays the current alternate subclass and provides choices for modifying it.
<i>Swap Layers</i>	Switches the active and alternate layers in the <i>Options</i> panel.
<i>Single trace mode</i>	Only available when routing differential pairs or during group routing. It allows you to switch from routing one or more nets to routing one net. A check appears before the item when single trace mode is active. You can also use the <code>pop routespace</code> command.

A Commands

A Commands--add connect

Change Control Trace	Only available when group routing or routing differential pairs. During group routing, it allows you to change the control trace that the editor selected. The control trace routes to the cursor location and the other traces follow along with it. During routing of differential pairs, you can switch from the active trace to the other trace.
Neck mode	<p>Changes the line width for the next segment to the neck width specified in the physical rule set. A check appears before the item when neck mode is active. The editor remains in neck mode until you choose the <i>Neck mode</i> menu item again. If you are routing a differential pair, necking the traces may also result in a change in the line-to-line spacing.</p> <p>In differential pair routing, if the <i>Min Neck Width</i> is the same as the <i>Min Line Width</i>, but the <i>DiffPair Neck Gap</i> value is less than the <i>DiffPair Primary Gap</i>, the layout editor recognizes neck mode for both DRC and line width checking. When the <i>Primary Line Width</i> and <i>Neck Width</i> are equal, DRC does not check for <i>Max Neck Length</i>.</p> <p>For information on how the layout editor uses constraint values in routing and checking differential pairs, see the <i>Routing the Design</i> user guide in your documentation set.</p>
Toggle	Enables you to flip the orientation of the rubber band line when <i>Line Lock</i> is set in the <i>Options</i> panel. The following example shows the result of using <i>Toggle</i> with <i>Line Lock</i> set to <i>Line 45</i> .
Multi-Line Route	<p>Lets you perform a freestyle bus route comprised of one or more connect lines. The route is initiated independent of any existing elements in the design. This command option recognizes a pre-existing route keepin and provides graphic feedback whenever the route exceeds its boundary.</p> <p>Your first mouse pick determines the start location of the route and also displays the <i>Multi-Line Route</i> dialog box enabling you to specify the route parameters and control trace.</p>
Enhanced Pad Entry	<p>Improves transitioning of clines that enter and exit a pad. The Enhanced Pad Entry mode works on circular, rectangular, and oblong pads, placed at any angle. It allows a cline to exit perpendicularly to the pad edge or at an angle to the pad edge that does not create an acute angle.</p> <p>By default, this option is ON. You can toggle from the pop-up menu when the command is active.</p>
Target	<p>Provides access to the following items in a submenu:</p> <ul style="list-style-type: none"> ◦ <i>New Target</i> enables you to change the destination of the connection (redirect the ratsnest line to a different pin on the same net). Choose this option and then choose the new target. ◦ <i>No Target</i> eliminates the rubber band line from the cursor to the destination. This is useful in congested areas that require cleanup before you can complete a new connection. ◦ <i>Route from Target</i> s the route-from and route-to elements. In both single net and group routing applications, previous existing etch/conductor remains while the view shifts to the new route-from element. If necessary, the routing subclass changes to be compatible with the new route-from element, which could be a pin, via, or cline segment. ◦ <i>Snap Rat T</i> moves a Rat T to the last pick location if the destination is a Rat T.
Finish	Completes the connection. This option only routes on a single layer and is not available for differential pair routing.
Via Structure Rotation	<p>Only available when you selected <i>Add Via structure</i> command.</p> <p>It lets you mirror or orthogonally rotate the via structures. Choices for rotation are: <i>0</i>, <i>90</i>, <i>180</i> and <i>270</i> degrees.</p>
Via Structure Return Net	<p>Lets you select a net to assign to the return path vias. Available only if the selected via structure has return path vias.</p> <p>This option is available only when you selected <i>Add Via structure</i> command.</p>
Scribble Mode	<p>Lets you generate a complex route path between two points using controlled shove and push techniques.</p> <p>If this option is enabled, then <i>Clip dangling clines</i> and <i>Route offset</i> are disabled in the <i>Options</i> panel.</p>
Snake Mode	<p>Lets you generate arc routing in a channel of pin/via hex field pattern.</p> <p>If this option is enabled, then <i>Bubble</i>, and <i>Optimize in channel</i> are disabled in the <i>Options</i> panel.</p>

A Commands

A Commands--add connect

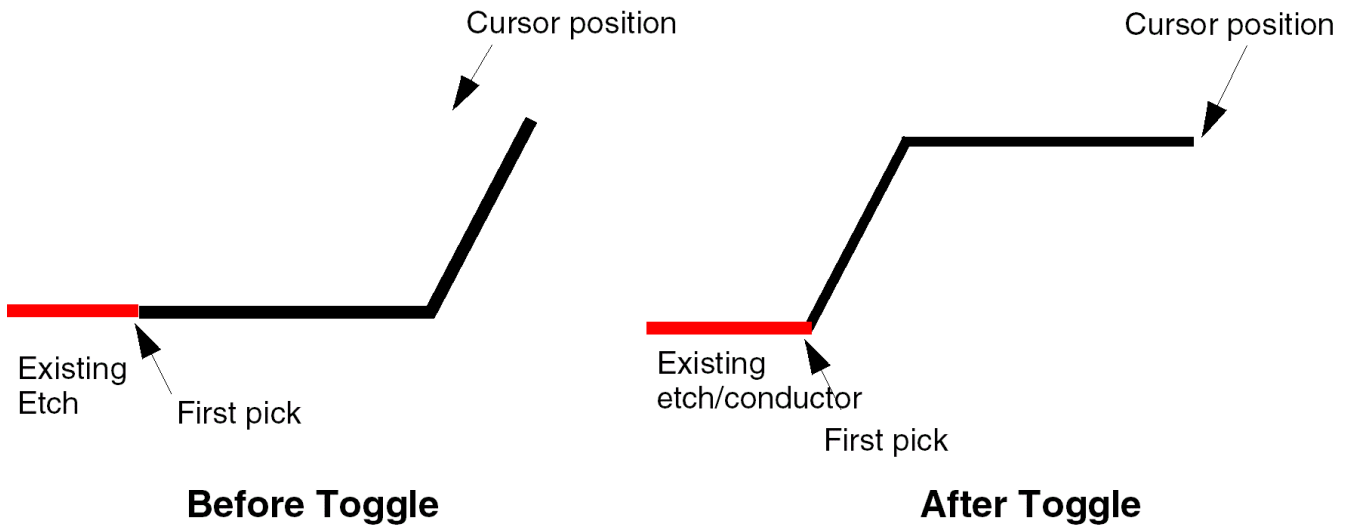
Snake Options	<p>Available when <i>Snake Mode</i> is active.</p> <p>Lets you create snake routing for a single trace. Two options</p> <p>are available:</p> <ul style="list-style-type: none">• Center Single Traces in Channel• Switch Single Trace to other Lane
Contour Mode	<p>Lets you route one or more connect lines while allowing you to hug the contour of the boundary line. The boundary can be either a route keepin or a connect line.</p> <p>To set the optional gap (boundary offset) for the route, select <i>Contour Options</i>.</p> <p>If this option is enabled, then <i>Optimize in channel</i> is disabled in the <i>Options</i> panel.</p>
Contour Options	<p>Provides options to control spacing between the etch being routed and the object being contoured. (Objects within a group route and differential pairs remain at their current spacing within the group).</p> <ul style="list-style-type: none">• <i>Current Space</i>: Select to start the contouring from any user-selected location• <i>Minimum DRC</i>: Select to use minimum DRC value of the control trace to the contour object• <i>User-defined</i>: Select to specify a user-defined value <p><i>Contour Space</i>: Only enabled to specify <i>User-defined</i> spacing.</p>
Design Parameters	<p>Displays the <i>Etch Edit</i> tab of the <i>Design Parameter Editor</i> when you use the pre-selection use model and you need to change several common parameters that apply to etch edit mode (see the <code>prmed</code> command). Changing a parameter here automatically updates its value on the <i>Options</i> panel as well.</p>
Options	<p>Displays all parameters relevant to the command when you use the pre-selection use model and you need to quickly change one parameter. Changing a parameter here automatically updates its value on the <i>Etch Edit</i> tab of the <i>Design Parameter Editor</i> as well.</p>
Snap Pick to	<p>Enables you to snap your next mouse pick to the closest design element you choose from the option sub-menu.</p>
Complete	<p>Finishes the <i>Temp Group</i> selection. Only available in the verb-noun use model.</p>
Layers	<p>Switches the active and alternate layers in the <i>Options</i> panel.</p>
Route Spacing	<p>Lets you change the spacing mode during group routing.</p> <ul style="list-style-type: none">• <i>Spacing Mode</i>:<ul style="list-style-type: none">◦ <i>Current Space</i>: Select to use the current spacing of the selected objects◦ <i>Minimum DRC</i>: Select to use the minimum DRC value for all the objects in the group◦ <i>User-defined</i>: Select to use a specific user-defined value• <i>Space</i>: Only enabled to specify <i>User-defined</i> spacing.• <i>Alignment</i>: Can be set to <i>Default</i> or <i>Control Trace</i>. This option is available only for <i>Minimum DRC</i> and <i>User-defined</i> spacing modes.

Related Topics

- [pop routespace](#)
- [Interactive Freestyle Multi-Line Routing](#)
- [Routing with Enhanced Pad Entry](#)
- [About Scribble Mode](#)
- [About Snake Mode](#)
- [Using Contour to Route Rigid-Flex Designs](#)
- [add connect](#)

A Commands

A Commands--add connect



Add Connect Command: Tasks

You can perform the following tasks using the `add connect` command:

- [Adding a Connect Line](#)
- [Adding a Through-Hole Via While Routing a Single Trace](#)
- [Adding a Via Structure While Routing](#)
- [Adding a Jumper While Routing a Single Trace](#)
- [Routing from or to Rat Ts](#)
- [Using Single Trace Mode With Differential Pairs](#)
- [Adding Vias to a Differential Pair](#)
- [Changing Via Patterns](#)
- [Changing Via Spacing Using the Diff Pair Via Space Dialog Box](#)
- [Routing Groups](#)
- [Using Single Trace Mode During Group Routing](#)
- [Changing the Spacing Mode During Group Routing](#)
- [Routing with Layer-Set Constraints](#)
- [Performing a Freestyle Multi-line Route](#)
- [Routing Connections Using the Contour Option](#)
- [Routing Using Route Offset Angle](#)
- [Routing in Channel](#)

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)

Adding a Connect Line

To add connect lines, do the following:

1. Hover your cursor over an element (pin, via, or etch/conductor segment) from which you want to start adding etch/conductor. The layout editor highlights the net on which you are routing and a data tip identifies its name.
2. Right-click and choose *Add Connect* from the pop-up menu to automatically launch the command. The layout editor identifies the element name in the *Options* panel, and the net name and active subclass also appear in the two panes of the status bar, to the left of the current mouse coordinates. A rubber band line appears from the element to the cursor and from the cursor to the target element.

⚠ The point at which the connection begins is from the location at which you right mouse clicked or snaps to the center of the pin or vertex.

The color of the rubber band is the same as the *Etch/Conductor* active subclass if the target element is on the active subclass. Otherwise, the color is that of the *Etch/Conductor* subclass that the target element is on. It follows the cursor while maintaining the angle specified in the *Line lock* field in the *Options* panel. In the following figure, the line lock angle is set to *Off*.

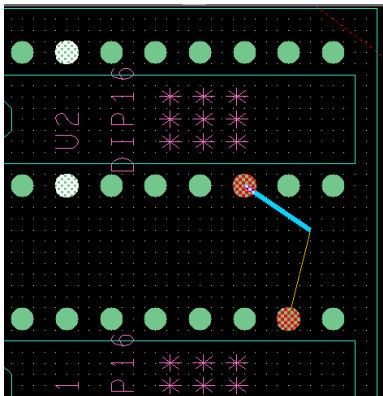
If the net has a timing constraint, the layout editor provides you with feedback. For additional information on displaying timing feedback, see Routing the Design in the user guide.

3. Right-click and choose *Change Active Layer* to choose an Etch/Conductor subclass from the list that displays.

⚠ If the first (starting) element is not on the active subclass, the active subclass is automatically changed to the subclass of the picked object. The action typically applies when you connect to clines, shapes, filled rectangles, surface-mount pins, and blind/buried vias. If the automatically changed subclass is the same as the current alternate subclass, the subclasses are simply picked. Otherwise, the alternate subclass remains unchanged.

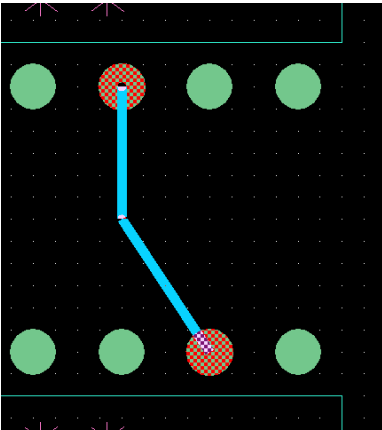
4. Configure the other options by right-clicking and choosing *Design Parameters* from the pop-up menu when you need to change several parameters or by entering values on the *Options* panel to quickly change one parameter. Changing a parameter in either place automatically updates its value in the other.

Starting a Connect Line



1. Move the cursor to the location at which you want the first etch/conductor segment to end.
The segments are shown at the specified width, in the color for etch/conductor, as shown in the following figure.

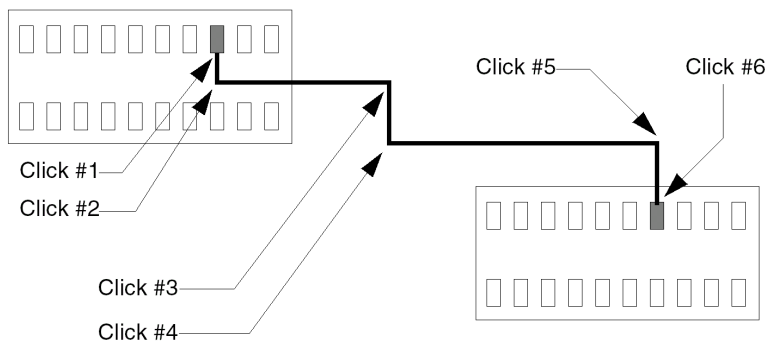
Segments of a Connect Line



You can override the default line width by changing it in the *Options* panel of the Control Panel or by right-clicking and choosing *Design Parameters* from the pop-up menu. Changing a parameter in either place automatically updates its value in the other.

1. Click to add the first segments, or right-click to use the pop-up menu options.
2. Continue clicking to add etch/conductor segments until you reach the destination element as shown in the following figure.

Routing Segments of a Connection



Click to create a start and an end point for each segment in a connection.

You are automatically set up to begin a new connection when you reach the destination element.

1. To end the connection at any time, right-click and choose *Done* from the pop-up menu.
You can also choose *Cancel* from the pop-up menu to reverse the connection back to the point where you started routing.
An online DRC check occurs after each pick.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Adding a Through-Hole Via While Routing a Single Trace

To add a through-hole via while routing a single trace, follow these steps:

1. Hover your cursor over an element (pin, via, or etch/conductor segment) from which you want to start adding etch/conductor. A data tip identifies the element name.
2. Right-click and choose *Add Connect* from the pop-up menu to automatically launch the command.
3. Set parameters as needed in the *Options* panel, for example, *Via*, or right-click to display the pop-up menu from which you may choose either *Options* panel to quickly change one parameter or *Design Parameters*, to access the Design Parameters Editor when you need to change several parameters.
4. Move the cursor to the location where you want to place the through-hole via.
5. Choose one of these ways to add the via:
 - Double click to automatically add a via while adding conductor segments.
 - Click to add the segment and then choose *Add Via* from the pop-up menu.

The via appears at the location where you clicked last.

6. Continue clicking to add etch/conductor segments until you reach the destination element.
Any connections the layout editor creates are added on the active layer. When you choose any pop-up options such as *Add Via* or *Layer* that move the connection to another layer, the layout editor switches to the alternate layer (active and alternate layers reverse in the *Options* panel).
7. To end the connection, choose *Done* from the pop-up menu.
You can check the current routing status by choosing *Tools – Reports* ([reports](#) command). For additional information about generating reports on interactive routing, see [Routing the Design](#) in the user guide.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Adding a Via Structure While Routing

To add a via structure while routing, follow these steps:

1. Hover your cursor over an element (pin, via, or etch/conductor segment) from which you want to start adding etch/conductor. A data tip identifies the element name.
2. Right-click and choose *Add connect* from the pop-up menu to automatically launch the command.
3. Set parameters as needed in the *Options* panel.
4. Select a via structure from the *Via/VS* drop-down list.
5. Move the cursor to the location where you want to place the via structure.
6. Choose one of these ways to add the via structure:
 - Double-click to automatically add a via structure while adding conductor segments.
 - Click to add the segment and then choose *Add Via Structure* from the pop-up menu.
The via structure is attached to the mouse cursor.
7. To mirror or rotate the via structure, right-click and choose *Via Structure Rotation*.
8. If the selected via structure has return path vias, the *Select Return Path Net* dialog box appears.
9. Select a net to assign to the return path via nets and click OK in the *Select Return Path Net* dialog box.
10. Click to add the via structure.
The via structure appears at the location where you clicked.
11. Continue clicking to add etch/conductor segments until you reach the destination element.
12. To end the connection, choose *Done* from the pop-up menu.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Adding a Jumper While Routing a Single Trace

To add a jumper while routing a single trace, follow these steps:

1. Hover your cursor over an element (pin, via, or etch/conductor segment) from which you want to start adding etch/conductor. A data tip identifies the element name.
2. Right-click and choose *Add Connect* from the pop-up menu to automatically launch the command.
3. Set parameters as needed in the *Options* panel.
4. Move the cursor to the location where you want to place the jumper.
5. Right-click and choose *Add Jumper* from the pop-up menu.
6. Choose jumper footprint name from the list.
The jumpers that are defined in the PSMPATH are displayed in bold. The jumpers that are not defined in the PSMPATH are disabled.
The jumper pin 1 appears at the location where you clicked last.
7. Alternatively, choose to mirror or rotate the jumper.
8. Click in the design to complete the jumper placement.
Ratsnest does not display across jumper pins.
9. Continue to add etch/conductor segments until you reach the destination element.
10. To end the connection, choose *Done* from the pop-up menu.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Routing from or to Rat Ts

The following procedure describes the `add connect` behavior when you interactively route nets containing Rat Ts.

1. Hover your cursor over a pin, Rat T, or other etch/conductor object as the starting point for the trace. The layout editor highlights the object and a data tip identifies its name.
2. Right-click and choose *Add Connect* from the pop-up menu to automatically launch the command.
3. Choose the active Etch/Conductor subclass in the *Options* panel or right-click to display the pop-up menu from which you may choose *Change Active Layer* to choose an Etch/Conductor subclass from the list that displays. The net name and active subclass also appear in the two panes of the status bar, to the left of the current mouse coordinates.
4. Click to add traces that you want to route.
If your pick completes the connection to the destination:
 - The rubber band lines and ratsnest line disappear.
 - `add connect` terminates.
If your destination is a Rat T and your pick does not complete the connection, you can choose Snap Rat T from the pop-up menu to move the Rat T to your last pick location, completing the connection to the destination.
5. When the connection is complete, the command terminates.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Using Single Trace Mode With Differential Pairs

While you usually route both traces of a differential pair, you can use single trace mode to route one trace at a time. For additional information about single trace mode, see the Routing the Design user guide in your documentation set.

To use single trace mode during the routing or editing of differential pairs:

1. In your design, right-click and choose *Single trace mode* from the pop-up menu.
The companion net is immediately dropped.
While in single trace mode, if you route the selected net to its destination, routing automatically switches to the companion net. The route-from point for the companion net is the same one that was in effect when you turned on single trace mode.
2. Choose *Change Control Trace* from the pop-up menu if you want to switch to the other net in the differential pair before completing the active route.
The companion net becomes active, and the net that was previously active becomes the companion net.
If the cursor is positioned close to a cline segment of the companion net, the route-to point snaps to a point that is spaced from the companion net segment by the applicable differential pair gap. The snapping trap distance is the differential pair gap.
When snapping to a companion net cline segment, if the new route also begins at a point that is the differential pair space from another segment of the same companion cline, the editor routes the new trace along the companion cline, spacing the new route by the differential pair gap. The new route ends at the snapped route-to point.
3. To exit single trace mode, choose *Single trace mode* again from the pop-up menu.
If you turn off single trace mode after having added single trace routes, routing is controlled by the net that was last active in single trace mode. The companion trace either snuggles up to the route on the control net or trims back to it, depending on which routes further.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Adding Vias to a Differential Pair

To add vias while routing a differential pair:

1. Double click or right-click in the design to display the pop-up menu (see [Pop-up Menu Options](#) for additional information).
2. Choose *Add Via*.
The vias with connecting Etch/Conductor appear and move with the cursor.
You can also change the via pattern and space. For information, see [Changing Via Patterns](#) and [Changing Via Spacing Using the Diff Pair Via Space Dialog Box](#).
3. Position the cursor so that the vias are in the specified location and pick to place the vias.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Changing Via Patterns

You can change the via pattern. The editor remembers the values and uses them the next time you add vias.

To change the via pattern:

1. In the design, right-click to display the pop-up menu.
You can also use the Pop via Pattern command.
2. Choose Via Pattern and then choose one of the patterns that appear in the submenu: *Next Pattern*, *Horizontal*, *Vertical*, *Diagonal Up*, or *Diagonal Down*.
The via pattern shown to the left of each menu item corresponds to the via pattern type listed.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Changing Via Spacing Using the Diff Pair Via Space Dialog Box

From the pop-up menu:


1. Choose *Via Pattern* and then choose *Spacing* from the submenu.
The Diff Pair Via Space dialog box appears.
2. Choose a spacing mode from the choices described below:
 - *Automatic*: The editor uses a spacing value that allows room to best meet the spacing, pad entry, length tuning, and uncoupled length requirements.
 - *Minimum*: The editor considers these values when spacing: *Via To Via*, *Primary Gap*, and *Line To Line* or *Via To Line*.
 - *User-defined*: The editor uses the value that you define by entering a value in the Space field.
3. Click *OK* to set the value and dismiss the dialog box.


For additional information on routing with vias, see *Routing the Design* in the user guide.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Routing Groups

 Group routing does not support routing from shapes.

 You can only add vias when you are in single trace mode.

For routing groups, follow these steps:

1. Window select multiple nets for group routing. The layout editor highlights the objects and a data tip identifies its name.
2. Right-click and choose *Add Connect* from the pop-up menu to automatically launch the command.
3. To change the control trace, right-click and choose *Change Control Trace from the pop-up menu*; then pick on the net to specify the control trace. The control trace changes to the specified trace. If there are only two traces, the editor automatically selects the other trace as the control trace. Then, routing resumes with the new control trace.
4. Continue routing to the destination.
 - To change group routing to single trace mode, see [Using Single Trace Mode During Group Routing](#).
 - To change the spacing mode, see [Changing the Spacing Mode During Group Routing](#).
5. When the routing is complete, the command terminates.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Using Single Trace Mode During Group Routing

To use single trace mode during group routing, follow these steps:

1. To switch to single trace mode during group routing, right-click and choose *Single Trace Mode* from the pop-up menu.
The control trace is active and the companion nets are dropped.
2. You can do either of the following:
 - a. Route to the destination. When the active trace's connection is completed, another trace becomes active.
The route-from point for the new active trace is the same one that was in effect when you switched to single trace mode.
 - b. Change the control trace by choosing *Change Control Trace* from the pop-up menu.
3. Disable single trace mode in the pop-up menu, thereby switching to group routing.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)


Changing the Spacing Mode During Group Routing

To change the spacing mode while grouprouting, follow these steps:

1. When you are in the `add connect` mode for group routing, right-click and choose *Route Spacing* from the pop-up menu.
The Route Spacing dialog box appears.
2. Click one of the radio buttons to select a spacing mode:
 - *Current*
 - *Minimum DRC*
 - *User-defined*

If you choose *User-defined* as your spacing mode, make sure that you specify a value in the *Space* field.
For additional information about spacing mode during routing, see *Routing the Design* in the user guide.

3. Click *OK* to apply the setting and dismiss the dialog box.

 The spacing mode reverts to *Current* when you initiate the `add connect` command. The user-defined values are saved with the database and restored from the saved values.


Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Routing with Layer-Set Constraints

Before you can route a net with layer-set constraints, you must define the layer sets and assign nets to them.

1. Hover your cursor over a net to route. The layout editor highlights the net, and a data tip identifies its name.
2. Right-click and choose *Add Connect* from the pop-up menu to automatically launch the command.
The legal routing layers (also referred to as subclasses) display in bold-faced type in the *Act* (active subclass) and *Alt* (alternate subclass) drop-down list boxes. If necessary and possible, the active subclass field automatically changes to the subclass closest to the current active subclass.

 Adding routes on a layer set subclass locks the net to that layer set. Routing on layers from more than one layer set results in DRC violations.

3. When routing is complete, the command terminates.

For additional information on interactive routing with layer-set constraints, see the *Routing the Design* user guide in your documentation set.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Performing a Freestyle Multi-line Route

To perform a freestyle multi-line route, follow these steps:

1. Choose *Route – Connect*.
2. Place your cursor in the canvas, right-click and choose *Multi-Line Route* from the pop-up menu, then click a starting location in free space where you intend to begin the route.
The Multi-Line Route dialog box appears.
3. In the dialog box, enter the route parameters, choose a control trace for the connect line group, then click *Ok*.
4. Move your cursor in the desired direction of the route staying within the bounds of a previously defined route keepin area. Click to insert vertices in the route path as necessary and continue on towards the destination.
As you route, the command provides graphic feedback by changing the color of the connect lines as well as the display of the control cursor when the bounds of the route keepin are exceeded. Additional message feedback is provided in the Allegro Console window.

⚠ If the control trace becomes inconvenient as you route, right-click and choose *Change Control Trace* from the pop-up menu. Click on an alternate trace in the group to provide control, then continue with the route.

5. When the multi-line route is complete, right-click and choose *Done* from the menu.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Routing Connections Using the Contour Option

To route connections using the contour option, follow these steps:

1. Select an object to route.

 A net must be assigned to the selected object.

The layout editor highlights the object and a data tip identifies its name.

2. Right-click and choose *Contour Mode* from the pop-up menu.
3. Right-click and select *Contour Options* from the pop-up menu
The *Contour Options* dialog box appears.
4. Optionally, select a *Spacing Mode* to designate a hug offset distance from the boundary to the route.
5. Click *OK* to dismiss the dialog box.
6. Move the cursor to select a curved section of a Route Keepin or Connect Line you intend to hug.
The curved boundary (of the type specified) closest to your cursor highlights and the following message is displayed in the command window:
Contour Unlocked: Click to lock onto highlighted object
7. Click to select the boundary to designate a starting location for contour routing.
The route begins to hug the boundary line at a location perpendicular to the start point.
8. Move the cursor along the curved boundary section and continue on with the route hugging the boundary contour.
9. Click when you reach the point where you wish to suspend contour routing and resume straight-line routing.
The route ceases to hug the boundary and continues on a straight path. The following message is displayed in the command window:
Contour Locked: Click to unlock from contour routing
10. Right-click and choose *Next* to select a single line segment or window select to select multiple line segments.
11. Repeat steps 1 to 9 to contour routing again within the same route as necessary.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Routing Using Route Offset Angle

To perform routing using route offset angle, follow these steps:

1. Hover your cursor over an element (pin, via, or etch/conductor segment) from which you want to start adding etch/conductor. The layout editor highlights the net on which you are routing and a data tip identifies its name.
2. Right-click and choose *Add Connect* from the pop-up menu to automatically launch the command. The layout editor identifies the element name in the *Options* panel, and the net name and active subclass also appear in the two panes of the status bars. A rubber band line appears from the element to the cursor and from the cursor to the target element.
3. Select *Route offset* and set the offset angle in the *Options* panel.
4. Move the cursor to the location at which you want the first etch/conductor segment to end. The route angle is 10 degrees.
5. Click to add the segments.
6. Continue clicking to add etch/conductor segments until you reach the destination.
7. To end the connection, right-click and choose *Done* from the pop-up menu.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

Routing in Channel

To route in channel, follow these steps:


1. Hover your cursor over an element (pin, via, or etch/conductor segment) from which you want to start adding etch/conductor. The layout editor highlights the net on which you are routing and a data tip identifies its name.
2. Right-click and choose *Add Connect* from the pop-up menu to automatically launch the command. The layout editor identifies the element name in the *Options* panel, and the net name and active subclass also appear in the two panes of the status bars. A rubber band line appears from the element to the cursor and from the cursor to the target element.
3. Enable *Optimize in channel* and set the *Channel Air Gap* using *Options* button.
4. Move the cursor to the location at which you want the first etch/conductor segment to end.
5. Click to add the segments.
Clines are automatically centered between the pads either dynamically, or after the pick in *Scribble* mode.
6. Continue clicking to add etch/conductor segments until you reach the destination.
7. To end the connection, right-click and choose *Done* from the pop-up menu.

Related Topics

- [add connect](#)
- [Add Connect Command: Options Panel](#)
- [Add Connect Command: Pop-Up Menu Options](#)
- [Add Connect Command: Tasks](#)

add d2d vias

The `add d2d vias` command lets you add die-to-die vias to pins in a design and also create the padstack definitions for the vias being added. This command provides an efficient workflow to create multiple die-to-die vias simultaneously, rather than creating them individually using Padstack Editor.

 The `add d2d vias` command is only available in Allegro X Advanced Package Designer.

A *Through Via* type padstack cannot be defined in the diestack region of the cross-section because die-to-die vias, or any vias with at least one pad on a diestack layer, have a maximum span of one layer transition.

Related Topics

- [Padstack Editor](#)
- [Adding D2D Vias to Design](#)

Add D2D Vias Command: Options Panel

Access using

- Menu path: *Route – Die to Die Via Generator*

<i>Start Layer</i>		Defines the name of the diestack layer where padstack definitions begin.
<i>End Layer</i>		Defines the name of the diestack layer where padstack definitions end.
<i>Padstack Name</i>		Defines the base name for the generated padstack. Default padstack name is <i>D2D</i> . Padstack names adhere to the following pattern: <i>D2D</i> , <i>D2D_1</i> , <i>D2D_2</i>
<i>Add layer names to base pad name</i>		Specifies that layer names are to be included in the base pad name. Padstack names adhere to the following pattern: <i>D2D_START_END</i>
<i>Pad Shape</i>		Defines the shape of the generated pads. The pad shape options are: <i>Square</i> , <i>Rectangle</i> , <i>Circle</i> , and <i>Octagon</i> . The default padstack shape is <i>Square</i> .
<i>Pad Width</i>		Defines the width of the generated pads. The default pad width is set to 1.00 UM.
<i>Pad Height</i>		Defines the height of the generated pads. The default pad height is set to 1.00 UM. This field is visible only if an asymmetric pad shape is selected.
<i>Generate Padstacks</i>		Select this option to generate the padstacks.
<i>Add Vias to Pins</i>		Specifies that vias are to be added to pins.
	<i>To layers above</i>	Specifies that the vias are to be added to pins in a specified number of layers above the Start layer. The value is set to 1 by default.
	<i>To layers below</i>	Specifies that the vias are to be added to pins in a specified number of layers below the Start layer. The value is set to 1 by default.

Adding D2D Vias to Design

Die-to-die vias can have a maximum of one layer transition because these vias have at least one pad on a diestack layer. You can also create padstacks while adding the die-to-die vias.

To add D2D vias to your design, do the following:

1. Choose *Route – Die to Die Via Generator*. Alternatively, run the `add d2d vias` command from the command line.
2. Configure the *Options* panel.
3. Optionally, create padstacks by clicking *Generate Padstacks* after specifying the start and end layers, padstack name, and pad shape and width. The padstacks are added to the database.


4. Repeat steps 2 and 3 to create the required padstacks. This step may be required if the required padstacks are of different sizes and shapes for different layer pairs.
5. Update the number of layers you want to transition the selected pin by setting the controls under *Add vias to pins*.
6. Select the pins where die-to-die vias have to be added.

Related Topics


- [add d2d vias](#)
- [Add D2D Vias Command: Options Panel](#)

add fillet

The `add fillet` command generates fillets interactively on conductive elements in your design; that is, on individual nets, clines, pins, and vias. Fillets may be created between a trace and a pin, a trace, and a via, or two traces (a T). Before executing the command, set the parameters that govern filleting in the *Fillet and Tapered Trace* dialog box, available by choosing *Route – Teardrop/Tapered Trace – Parameters* ([gloss param fillet command](#)).

 Fillets are always generated on the largest pad at a location or layer and not necessarily the pad a cline is connected to in the database. In addition, if a T point lies inside a pad, fillet for the cline creating the T point will be generated to the pad.

You cannot run this command if the *Dynamic* option is enabled on the *Fillet and Taper Trace Fillet* dialog box. Or if you have specified NO_GLOSS areas, no fillets generate in those areas.

 The NO_FILLET property, attached to a net, pin, or via, prevents the creation or regeneration of fillets on these elements, even if the Dynamic Fillets option is enabled on the Pad and T Connection Fillet dialog box.

Access Using

- Menu Path: *Route – Teardrop/Tapered Trace – Add Teardrops*

The only configurable options for this command are the active class and subclass.

Generating Fillets Interactively

To interactively generate fillets, perform three steps:

1. Choose *Route – Teardrop/Tapered Trace – Add Teardrops* (add fillet command).
The *Options* panel displays the active class and subclass and the *Find* panel defaults to *Nets* as the active design object.
2. Choose one or more traces for filleting. If you are performing the operation on multiple traces, you can use the right-button menu to choose *Temp Group* or *Window Select*.
3. Click on the right mouse button and choose *Done* or *Complete* from the pop-up menu.

add flash

The `add flash` command, available in the Symbol editor, displays the Thermal Pad Symbol Definition dialog box that lets you define the parameters of a flash thermal pad and add it to the `.dra` file.

Related Topics

- [Defining Parameters of a Flash Thermal Pad](#)

Thermal Pad Symbol Defaults Dialog Box

Access Using

- Menu Path: *Add – Flash*

<i>Thermal pad definition</i>	Defines the inner and outer diameters of the thermal shape in design units.
<i>Spoke definition</i>	Defines the width, number, and angle of the pad spokes.
<i>Center dot option</i>	When checked and the diameter defined, creates a dot void of Etch/Conductor in the center of the thermal pad.
<i>OK</i>	Closes the dialog box and adds the thermal shape to the <code>.dra</code> file.
<i>Cancel</i>	Closes the dialog box without saving changes.

Defining Parameters of a Flash Thermal Pad

To define the parameters of a flash thermal pad, perform the following steps:

1. Open a new or existing flash symbol drawing.
2. Run `drawing param` to open the Drawing Parameters dialog box.
 - a. Enter left *x* and lower *y* coordinates to accommodate a shape centered around 0, 0.
 - b. Choose *Flash* from the Type drop-down.
3. Run `add flash` to display the Thermal Pad Symbol Definition dialog box.
4. Define the parameters of the thermal pad by entering the appropriate information in the dialog box fields, as described above.
5. Click *OK* to close the dialog box and to add the thermal pad to the `.dra` file.
6. Run `create symbol` to save the thermal pad as a flash symbol (`.fsm`).


Related Topics

- [add flash](#)

add frect

The `add frect` command creates filled rectangles (frectangles). You can add filled rectangles in your drawings that you can define as

- Etch/Conductor rectangles (with associated net name for voltage distribution)
- Route keepouts
- Package/Part keepouts
- Via keepouts

 Keepouts can be any shape.

- Package/Part placement boundaries
- Masks

Filled rectangles added to the Etch/Conductor class represent etch/conductor on the design. The `plot` command writes line-plot commands to the photoplot file to fill that area on that layer. Since a major use of filled etch/conductor frectangles is to distribute a voltage over an area on a layer, a net name (voltage) is associated with each such filled rectangle.

When you add a filled rectangle as etch/conductor, a dialog box prompts you for the name of the net with which the filled rectangle is to be associated. Thereafter, you can attach connect lines to the frectangle so it is physically attached to its net. The `connect` command lets you make the connection because the frectangle is logically on that net.

 You can verify the net of any etch/conductor frectangle by running `show element` on the frectangle.

Access Using

- Menu Path: *Add – Frectangle*

The *Options* panel for `add frect` is configured only for class and subclass.

Creating Filled Rectangles

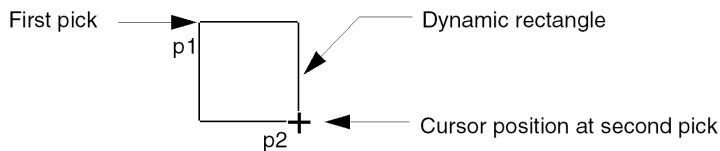
Follow these steps to create filled rectangles:

1. Run `add frect`.
2. Set/verify the Class and Subclass in the *Options* panel.
3. Draw the filled rectangle.
4. Click to complete the drawing.
If you drew the element on class ETCH/CONDUCTOR, a data browser displays a list of nets from which to associate the filled rectangle.
 - a. Choose a net name from the list and click *OK*.
5. Click right; choose *Done* from the pop-up menu to exit from the command.

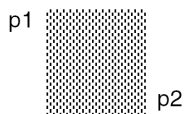
Examples

Filled Rectangle on Class ETCH/CONDUCTOR

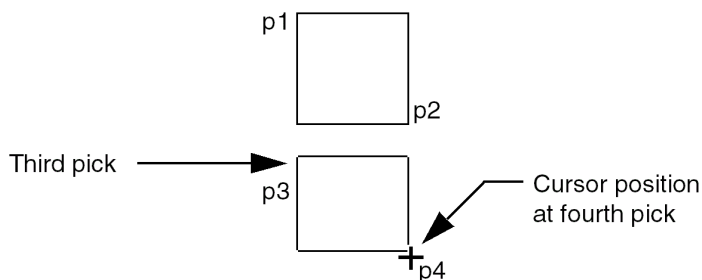
1. The following illustration shows picks at points p1 and p2. The dynamic rectangle displays unfilled before you click at p2.



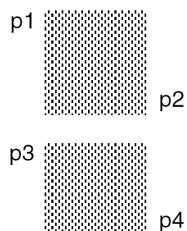
2. When you click at p2, a data browser displays a list of net names. When you choose a net and click *OK*, the filled rectangle displays as shown in the illustration.



3. Click at p3 and p4 to display two highlighted filled rectangles:

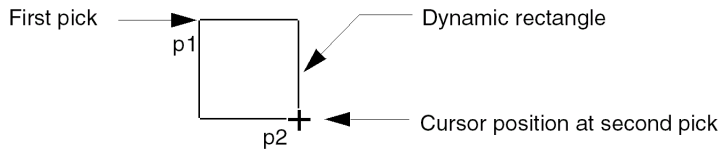


4. When you click right and choose *Done* from the pop-up menu, the two highlighted rectangle outlines appear filled in the color you selected for the etch/conductor layer:

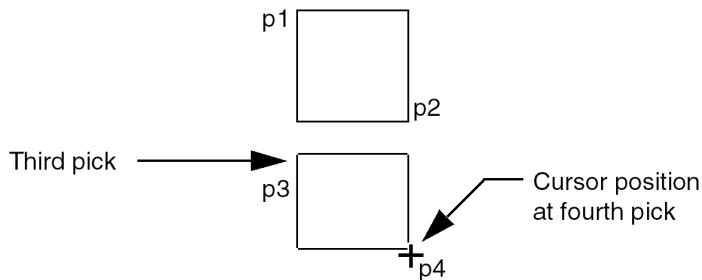


Filled Rectangle on Non-ETCH/CONDUCTOR Classes

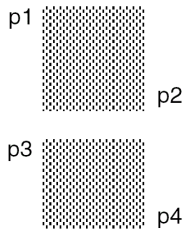
1. The following illustration shows picks at points p1 and p2. The editor displays the dynamic rectangle after you click at p1.



2. When you click at p2, the editor creates a rectangle between p1 and p2 and highlights it.
3. Click at p3 and p4 and the editor displays two highlighted filled rectangles:



4. When you click right and choose *Done* from the pop-up menu, the two highlighted rectangle outlines appear filled in the color you selected for the etch/conductor layer:



Changing font of Rectangle

You can change the line pattern used in creating the rectangle. Select the rectangle and right-click to choose *Change Line Font* command. Choose a new pattern from the list appears.

For more information, see [Changing line fonts of Elements](#) in *Allegro User Guide: Preparing the Layout*.

add fshape

The `add fshape` command adds closed, solid filled shapes or elements only on ETCH/CONDUCTOR subclasses of your design. The shapes are made from a continuous series of line/arc segments and filled with a solid field of copper. You create line segments by continuous mouse clicks or by entering coordinates at the command line. The shape between the first and last points is completed when you choose *Done* from the pop-up menu. At that point, the UI changes to the Shape editor.

Related Topics

- [Adding Elements to a Design](#)

Add Fshape Command: Options Panel

Access Using


- Menu Path: *Shape – Select Shape or Void*

<i>Line lock</i>	Specifies whether the shape is drawn with straight lines or with arcs and defines the angle of the corner when a line segment changes direction. The choices are <i>Line</i> , <i>Arc</i> and <i>Off</i> , <i>45</i> , and <i>90</i> .
<i>Line width</i>	Specifies the width of the line in user units.
<i>Line font</i>	Is not active in <code>add fshape</code> mode.


Adding Elements to a Design

To add elements to your design, follow these steps:

1. Run the `add fshape` command.
2. Configure the *Options* panel in the Control Panel.

 You can create filled shapes only on ETCH/CONDUCTOR subclasses.

3. Ensure that the subclass you are drawing the shape on is visible.
4. Left click at the vertices of the shape outline that you want to create.
5. When you are ready to complete the shape, do one of the following:
 - Close the shape by picking the starting point again (closing the shape outline), and then click right and choose *Done* from the pop-up menu.
 - Click right and choose *Done* from the pop-up menu.

 Note: When the shape outline is complete, the design area changes from layout editor to the shape editor. You can only edit one shape at a time while in the shape editor. The active shape is the last shape selected in your layout before you entered the shape editor.


6. Attach the shape to a net using one of the following techniques
 - Choose *Edit – Change Net (Pick)* and pick any object already associated with the net you require, such as a pin, connect line, via, or another shape.

-or-

- Choose *Edit – Change Net (Pick)* and enter the net name, at the fill-in, with which to associate the shape. Then click *Close*.

This makes the shape part of the net you select. Until you do this step, an etch/conductor shape is on a dummy net (which means no net). Non-etch/conductor shapes are never on a net.

7. Continue to define the shape, if need be.
8. When the shape meets your requirements, run `shape fill`.
The shape fills and you return to the layout editor. DRC is performed on the shape during the shape fill process.

 Note: Choosing `shape fill` is the only method to exit the Shape editor.

Setting Shape Parameters

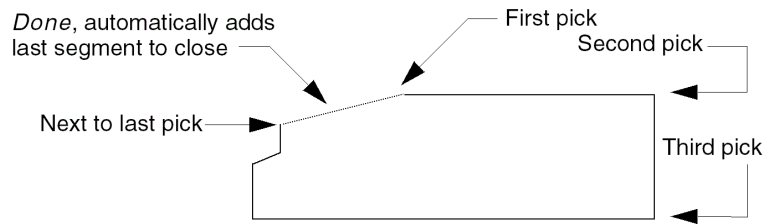
After you create a shape outline, you must specify the shape parameters. The parameters determine the following:

- The type of shape fill
- How voids are generated
- Void clearances
- How thermal-relief connect lines are generated

For details on how to specify these parameters, see [shape param](#).

Example

This example shows how to create a shape using mouse picks.



Related Topics

- [add fshape](#)

add interposer

The `add interposer` command lets you add interposers to a die stack. Interposers facilitate the wire bonding of dies where lateral wire-bond spans challenge the physical limits of a wire bond or the equipment used for attaching wire bonds. Interposers are rectangular in the die-stack editor graphics. You can place them at orthogonal rotations of 0, 90, 180, or 270 degrees.

The layout editor automatically attaches the LOCKED property to an interposer so that you cannot accidentally edit (move, delete, rotate) the symbol children, for example, place-bounds, assembly-rectangles, vias, etch, and so on. Although you can edit this property, it is recommended that you do not as corruption can occur if symbol children are edited. Whenever you update the interposer, the layout editor automatically adds the property to the spacer if you have removed it.

Building the Package Symbol

Before adding an interposer to a die stack, build it as a package symbol (.psm) with the following:

- A filled rectangle on PART_GEOMETRY/PLACE_BOUND_TOP
- Ref ID text on REF_DES/ASSEMBLY_TOP
- Clines, vias, and shapes on CONDUCTOR/TOP
- A rectangle on the PART_GEOMETRY/ASSEMBLY_TOP class and subclass (optional)
A corner mark helps viewing rotations.

The interconnect that you use for interposer symbols is limited to clines, vias, and shapes (no pins). Place the symbol on the TOP_COND layer in the Symbol Editor.

Specifying Properties

You can also add the properties for an interposer's thickness, material, and part number in the Symbol Editor. APD passes these properties to each interposer symbol instance in a package design. If APD does not find a given property on the symbol, you need to enter a value before the layout editor can place the symbol. The property names are:

- PART_NUMBER, an alphanumeric string, for example, 1ZX-256X-CL4 (optional)
- CONDUCTOR_THICKNESS, a number, for example, 30.48
- CONDUCTOR_MATERIAL, a material existing in the material file, `mcm_mat.data`, for example, COPPER
You can invoke the Material Browser by clicking the ... button that follows the *Name* text box in the Add Interposer dialog box.
- DIELECTRIC_THICKNESS, a number, for example, 100.00
- DIELECTRIC_MATERIAL, a material existing in the material file, `mcm_mat.dat`, for example, CERAMIC
You can invoke the Material Browser by clicking the ... button that follows the *Name* text box in the Add Interposer dialog box.

Adding the Symbol to the Package Design

When you add the symbol to the package design with the `add interposer` command, APD moves all CONDUCTOR symbol geometry to the non-substrate DIESTACK class layer where you choose to place the interposer symbol.

The `add interposer` command does not generate a log file.

Preconditions for Wire Bonding Interposers

The following describes preconditions for wire bonding interposers. For additional information, refer to the Wire Bonding Tools in the *Routing User Guide* of your documentation set, as well as the `wirebond select` command.

- You must add an interposer to your design using the *Add – Interposer* (`add interposer`) command. Otherwise, it does not appear in the die stack editor, nor are its pads (bond fingers) available as start positions for bond wires.
- All bond wire connection points of an interposer must be bond fingers (via type objects with a BOND_PAD property).
- Any conductor traces of the interposer symbol must *not* have the BOND_WIRE property on them. These are two-dimensional connection lines on a single-layer substrate above the regular package substrate's surface. As a result, they appear on a DIESTACK layer when added to a design. They should not have the BOND_WIRE property so that they do not get confused with actual, 3D bond wire objects.
- You can place an interposer only on a DIESTACK layer, similar to a wire bond die. Position it on a layer between the wire bond die that connects to it and the substrate layer to which it connects.

Related Topics

- [Adding an Interposer to a Die-Stack](#)

Add Interposer Dialog Box

Access Using

- Menu Path: *Add – Interposer*

<i>Ref ID</i>	Specifies the reference designator of the interposer. If you place multiple instances of the interposer, this value increments to a unique value after you place each interposer symbol. You can edit this value before placing the next symbol instance.
<i>Symbol Name</i>	Specifies the symbol name of the interposer. Click ... to find the directory where the library of symbol names is located.
<i>Part Number</i>	Specifies the alphanumeric part number of the interposer used in the Bill of Materials.
<i>Conductor Material</i>	
<i>Name</i>	Specifies the name of the conductor material used on the interposer. The electrical properties of an interposer are derived from its conductor material. Click ...to launch the Materials Editor for a selection of a conductor material. You can edit material properties by using the define materials command.
<i>Thickness</i>	Specifies the thickness of the interposer conductor material.
<i>Dielectric Material</i>	
<i>Name</i>	Specifies the name of the dielectric material that makes up the interposer. The thermal conductivity of a spacer is a property of its material.
Thickness	Specifies the thickness of the interposer dielectric material.
Placement	
Layer	Specifies the name of the non-substrate CONDUCTOR layer on which you are placing the interposer interconnect (vias, clines, and shapes).
<i>Rotation</i>	Specifies the angular rotation of the interposer. Use the drop-down list to specify the orientation of the spacer: <i>0, 90, 180, or 270</i> degrees.
OK	Saves the placements and dismisses the dialog box.
Place	Places the instance of the interposer in the design if you completed all the fields in the dialog box (<i>Part Number</i> is optional). APD places the symbol on the cursor and places multiple instances of the interposer when you pick an X, Y location in the Design Window or type X, Y coordinates at the console window prompt.
Cancel	Removes the placements and dismisses the dialog box.
Help	Displays the Help Window.

Adding an Interposer to a Die-Stack

Perform these steps to add an interposer to a die-stack:

1. Create the interposer in the symbol editor.
2. Run the `add interposer` command.
3. Complete the fields in the Add Interposer dialog box.
4. Click *Place* to place an instance of the interposer in the design.
The symbol appears on the cursor.
5. To place one instance of the interposer, either pick an X, Y location in the plan view or type an X, Y coordinate at the console window prompt, for example, x 2500, 3000.
The dialog box remains open.
6. To add another instance of the same interposer, edit the value of *Ref ID* (or use the default provided) and change the layer as required, then click *Place* again.
7. Click *OK* to save the placements and close the dialog box
or
Click *Cancel* to remove the placements and close the dialog box.

Related Topics

- [add interposer](#)

add line


The `add line` command creates non-etch/conductor line segments between two points. Use this command to create outlines, irregular shapes, and other figures in your design. When you create a line, the editor displays a rubber band from the point you selected to the cursor. The rubber band line adheres to the 90- or 45-degree constraints specified in the *Line Lock Direction* field of the *Options* panel, and draws arcs or line segments as specified in the *Line Lock Mode* field.

Related Topics

- [Creating Non-Etch/Conductor Line Segments between Two Points](#)

Add Line Command: Options Panel

Access Using

- Menu Path: *Add – Line*
- Toolbar Icon: 

<i>Line lock</i>	D efines whether the editor lays in the segments as lines or arcs. Defines the angle of the corner when a line segment changes direction. The choices are <i>Off</i> , <i>45</i> , and <i>90</i> .
<i>Line width</i>	Defines the width of the <i>Solid segment</i> in user units. All other line fonts remain at 0 width.
<i>Line font</i>	Defines the line pattern used in creating the segment. The choices are <i>Solid</i> , <i>Hidden</i> , <i>Phantom</i> , <i>Dotted</i> , and <i>Center</i> . The default is <i>Solid</i> .

The line pattern types are:

Solid	_____
Hidden	- - - - -
Phantom	_____
Dotted	- . - . - . - .
Center	_____

Line fonts, other than Solid, are allowed on the following Class/Subclasses:

- DRAWING FORMAT/All user defined subclasses
- MANUFACTURING/NCDRILL_LEGEND
- MANUFACTURING/All user defined subclasses
- PACKAGE GEOMETRY/ASSEMBLY_TOP
- PACKAGE GEOMETRY/ASSEMBLY_BOTTOM
- PACKAGE GEOMETRY/All user defined subclasses
- BOARD GEOMETRY/OUTLINE
- BOARD GEOMETRY/ASSEMBLY_NOTES
- BOARD GEOMETRY/ DIMENSIONS
- BOARD GEOMETRY/ASSEMBLY_DETAIL
- BOARD GEOMETRY/All user defined subclasses

You can change settings on *Options* panel before selecting points for each new segment.

Creating Non-Etch/Conductor Line Segments between Two Points

Perform the following steps to create a non-etch conductor line segment in your design:

1. Run the `add line` command.
2. Specify the values in the *Options* panel.
3. Choose the start and end points that define each line segment. You can use the mouse or enter coordinates at the command line.
4. When all lines are complete, choose *Done* from the pop-up menu or choose *Next* to create another series of lines. You can also flip the line using *Toggle*.

Changing Class and Subclass of Line

You can however, change the class and subclass of the line after addition. Select the line segment and right-click to choose *Change class/subclass* command. Choose a new class and subclass from the list appears.

For more information, see [Moving Elements to other classes](#) in *Allegro User Guide: Preparing the Layout*.

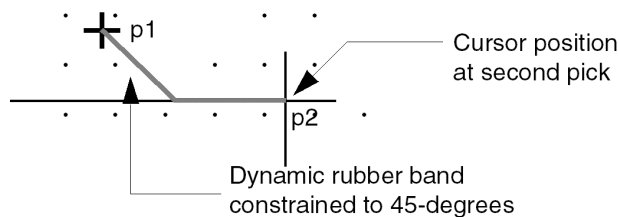
Changing font of Line

You can change the line pattern used in creating the line. Select the line segment and right-click to choose *Change Line Font* command. Choose a new pattern from the list appears.

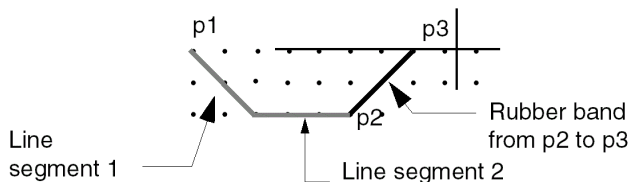
For more information, see [Changing line fonts of Elements](#) in *Allegro User Guide: Preparing the Layout*.

Example

1. In the first illustration, you have made the first pick at point p1 and are about to make the second pick at point p2.



2. You make pick 2.
The editor creates two line segments and continues the rubber band from p2.
3. You make pick 3.
The editor adds a line segment to p3, and rubber bands from that point.



4. To end the current line and start a new one, you click right and choose *Next* from the pop-up menu.
The current line completes and the cursor appears, letting you pick the starting point of the next line.

Related Topics

- [add line](#)

add parallel line

The `add parallel line` command creates lines parallel to existing lines. You can set the distance between the lines and the number of occurrences in the *Options* panel.

Related Topics

- [Adding Parallel Lines to your Design](#)

Add Parallel Line Command: Options Panel

Access Using

- Menu Path: *Manufacture – Drafting – Add Parallel Line*

Offset	Specifies the distance between the original and new lines. By default, it is set to 100.
Repetitions	Specify the number of lines to be added. By default, it is set to 1.

Adding Parallel Lines to your Design

Follow these steps to create lines parallel to existing lines in your design:

1. Choose *Manufacture – Drafting – Add Parallel Line* or run the `add parallel line` command.

OR

1. Set *General Edit* application mode and select a line segment. Right-click and choose *Drafting – Add Parallel Line*.
2. Select one or more lines.
The selected lines are highlighted.
3. Specify *Offset* in the *Options* panel.
4. Specify *Repetitions* in the *Options* panel.
5. Specify the direction for adding lines.
The lines are added with defined offset.
6. Right-click and choose *Next* to continue or *Done* to complete the operation.

Related Topics

- [add parallel line](#)

add perp line

The `add perp line` command adds a new line that is perpendicular to an existing line. To add this perpendicular you can choose either a point or an existing line as a starting point.

Access Using

- Menu Path: *Manufacture – Drafting – Add Perpendicular Line*

Adding Perpendicular Lines

Perform these step to add a new line which is perpendicular to an existing line in the design:

1. Choose *Manufacture – Drafting – Add Perpendicular Line* or run the `add perp line` command.

OR

1. Set *General Edit* application mode and select a line segment. Right-click and choose *Drafting – Add Perpendicular Line*.
2. Select an existing line or a start point.
A rubber band line is attached to the cursor with the point of selection as first end point.
3. Specify either the second end point or an existing line.
A perpendicular line is added to the selected line.
4. Right-click and choose *Next* to continue or *Done* to complete the operation.

add pin

The `add pin` command, available only in the Symbol Editor, lets you to place pins and create horizontal or vertical rows of sequentially numbered pins with a single click using both Rectangular and Polar coordinates. Use the *Options* panel to define which type of pin is active; for example, rectangular or polar.

Related Topics

- [Adding Pins](#)

Add Pin Command: Options Panel

Access Using

- Menu Path: *Layout – Pins*

<i>Connect</i>	Adds automatically numbered pins to a package/part symbol.
<i>Mechanical</i>	Adds pins to a mechanical symbol. These pins do not have numbers.

The following options on the *Options* panel reflect whether you choose the Connect or Mechanical option.

<i>Padstack</i>	Specifies the padstack to be associated with the pin. You must associate a padstack with the pin before you can place pins on the board/design. The button to the right of the padstack field displays a data browser containing a list of available database/library padstacks.
<i>Copy mode</i>	Specifies rectangular or polar for placement of the pins. Rectangular copies pins in a straight line, horizontally or vertically. Polar copies pins in a circle using the point you pick as the center of the circle.
<i>Qty x</i>	Specifies the number of pins you want placed in a horizontal direction.
<i>Qty y</i>	Specifies the number of pins you want placed in a vertical direction.
<i>Spacing</i>	Defines the spacing of each pin contained in a row. These two fields define the spacing of each pin. The first fields define the row spacing. The direction of the pins is determined by the entry in the box to the right of this field. The direction options are <i>Right</i> or <i>Left</i> . The default spacing is 100 mils and the direction default is <i>Right</i> . The second fields define column spacing. The direction of the pins is determined by the entry in the box to the right of this field. The direction options are <i>Up</i> or <i>Down</i> . The default spacing is 100 mils and the direction default is <i>Down</i> .
<i>Order</i>	Specifies the placement of pins. In <i>x</i> , you can choose to the left or right of the starting point. In <i>y</i> you can choose up or down from the starting point.
<i>Copies</i>	Specifies the number of pins you want to place.
<i>Angle inc</i>	Specifies the angle to add multiple pins along an arc, defined by origin and a start location for the first pin. Type a number between 0 and 360 or choose an option from the pop-up menu. Choose from 0, 45, 90, 135, 180, 215, 270, and 315.
<i>Ccw</i>	Specifies the direction of rotation as Counter-clockwise
<i>Cw</i>	Specifies the direction of rotation as Clockwise
<i>Rot mode</i>	Specifies the mode of rotation. <i>Absolute</i> : All pins are placed with selected padstack rotation. <i>Incremental</i> : All pins are placed with an incremental padstack rotation.
<i>Rotation</i>	After defining the angle at which the pins will be added, the pin can be rotated before defining the radius with the second click. The angles provided on the pop-up menu are: 0, 45, 90, 135, 180, 225, 270, and 315. You can also type a number between 0 and 360. The default value is 0.000.
<i>Pin #</i>	Specifies the pin number to be added.
<i>Inc</i>	Specifies the increment at which the automatically generated pin numbers are added.
<i>Text block</i>	Indicates the text block you want to use for the pin number text.
	A text block defines the size and spacing of the text you add to the design. Using the define text command, you can define up to 16 text blocks. The default is <i>1</i> .
<i>Text name</i>	Indicates the name of the text block.
<i>Offset X, Y</i>	Specifies the X and Y offset of the pin number text point from the origin of its associated pin. The default values are x-100 and y0.

Adding Pins

To add pins to your design, follow these steps:

1. From an open symbol drawing, run the `add pin` command.
2. Configure the controls in the *Options* panel.
3. Place the specified number of elements into your design.
4. When you have finished adding elements, choose *Done* from the pop-up menu.
You can use the `undo` command to step back and recover a recent changes(s). To reverse an undo operation, you can use the `redo` command immediately after using undo.

Example

Follow these steps to lay out pins and pin numbers for a 14-pin DIP. The first pin is square to distinguish it from the remaining, circular pins.

1. From an open symbol drawing, run the `add pin` command.
2. To define the first pin of the symbol, complete the *Options* panel as follows:
Copy Mode = Rectangle
Padstack = p50s32
X Qty = 1
Y Qty = 1
Spacing and Order = 100 Right : 100 Down
Rotation = 0.000
Pin # = 1
Increment = 1
Text Block = 1
Offset X = -100; y offset = 0
3. Move the cursor into the symbol window.
The cursor displays the padstack being added as a dynamic rectangle.
4. Click to anchor the pin, then immediately right-click and choose *Done* from the pop-up menu
Specifying a negative X offset places the pin number to the left of the pin
5. To define the next six pins in the first column of the symbol, complete the *Options* panel as follows:
PIN = Rectangle
Padstack = p50c32
X Qty = 1
Y Qty = 6
Spacing and Order = 100 Right: 100 Down
Rotation = 0.000
Pin # = 2
Increment = 1
Text Block = 1
Offset X = -100; y offset = 0
6. Move the cursor into the symbol window.
7. To anchor the pin 100 mils beneath Pin # 1:
Click to anchor the pin, then immediately click right and choose *Done* from the pop-up menu
8. To define the next seven pins in the second column of the symbol, complete the *Options* panel as follows:
PIN = Rectangle
Padstack = p50c32
Columns = 1
Rows = 7
Spacing = 100 Right: 100 Up
Rotation = 0.000
Next Pin = 8
Increment = 1
Text Size = 1

X offset = 100; y offset = 0

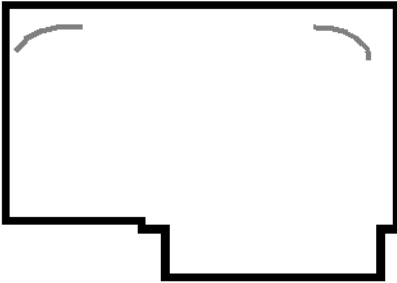
9. Move the cursor into the symbol window.

Related Topics

- [add pin](#)

add rarc

The `add rarc` command creates an arc-shaped element. When you know the radius of the arc you are adding to the design, run `add rarc`. The ability to locate the center point of an arc at a fixed reference is often important for mechanical specification of a design, particularly the outline. For example, to round off edges of an outline, you can create arcs, as shown here:



The `add rarc` command lets you specify the precise center point location or radius of the arc to be created. `add rarc` is typically used for the **OUTLINE** subclass, the default. However you can specify another layer for the arc by picking the subclass field in the *Options* panel and then selecting from the pop-up list of subclasses that appears.

Related Topics

- [Adding Arcs by Specifying the Radius](#)
- [add arc](#)

Add RARC Command: Options Panel



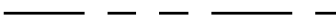
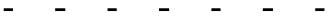

Access Using

- Menu path: *Add – Arc w/ Radius*

The Class and Subclass fields on the *Options* panel control the arc. The `add rarc` command is typically used for the OUTLINE subclass. You can change settings before clicking points for each new arc. The editor follows the parameter definitions in the Status dialog box unless you change them interactively in the *Options* panel.

<i>Line width</i>	D efines the width of the <i>Solid</i> lines used in creating the arc in user units. All other line fonts remain at 0 width.
<i>Lock angle</i>	Defines the angle of the lines used in creating the arc. The default is 90.000.
<i>Line font</i>	D efines the line pattern used in creating the arc. The choices are <i>Solid</i> , <i>Hidden</i> , <i>Phantom</i> , <i>Dotted</i> , and <i>Center</i> . The default is <i>Solid</i> .

The line pattern types are:

Solid	
Hidden	
Phantom	
Dotted	
Center	

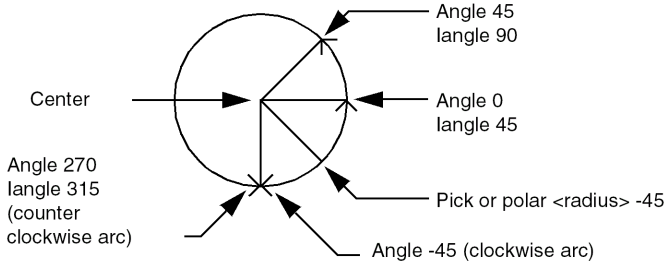
Line fonts, other than Solid, are allowed on the following Class/Subclasses:

- Drawing Format/All user defined subclasses
- MANUFACTURING/NCDRILL_LEGEND
- MANUFACTURING/All user defined subclasses
- PACKAGE GEOMETRY/ASSEMBLY_TOP
- PACKAGE GEOMETRY/ASSEMBLY_BOTTOM
- PACKAGE GEOMETRY/All user defined subclasses
- BOARD GEOMETRY/OUTLINE
- BOARD GEOMETRY/ASSEMBLY_NOTES
- BOARD GEOMETRY/ DIMENSIONS
- BOARD GEOMETRY/ASSEMBLY_DETAIL
- BOARD GEOMETRY/All user defined subclasses

Adding Arcs by Specifying the Radius

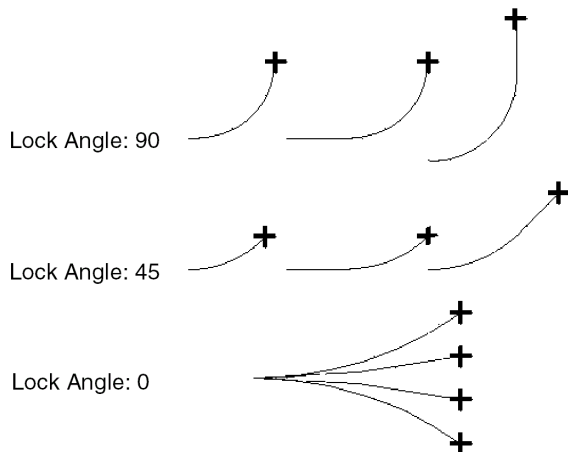
When adding arcs by specifying the radius, you must enter three points:

- Center point
- Start point, or enter a polar coordinate to establish the start point
- End point or enter an `angle` or `iangle` command as illustrated in the following example:



The procedure for adding an arc by specifying the radius is

1. Run the `add racc`.
2. Verify the Options for Class, Subclass, Line Width, and Lock Angle for the arc. Specify the angle of the arc by selecting a value from the pop-up menu (options are 0, 45, 90, 135, 180, 225, 270, or 315) or enter a unique angle with up to three decimal places.
The following illustration shows the `add racc` options and angles drawn using the Line Angle at 90, 45, and 0 degrees.

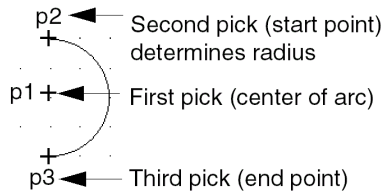


The editor prompts you to pick the center point of the arc.

3. Choose the first pick.
4. Choose the second pick.
5. Choose for the third pick to complete the arc.
6. Execute the completed arc or continue to enter picks for a second arc.
7. To add an arc to the drawing, click right and choose *Done* from the pop-up menu.

⚠ Instead of selecting points with mouse clicks, you can enter the points at the command line. As an alternative, you can enter a polar coordinate at the command line to specify the arc start point as radius and start angle. This method makes it easier to add a fixed radius arc to the drawing and is useful when you know the radius and start angle of a required arc.

The first pick specifies the center point of the arc (point 1). The second and third picks specify the start and end points:



Changing font of Arc

You can change the line pattern used in creating the arc. Select the arc and right-click to choose *Change Line Font* command. Choose a new pattern from the list appears.

For more information, see [Changing line fonts of Elements](#) in *Allegro User Guide: Preparing the Layout*.

Related Topics


- [add rarc](#)

add rect

The `add rect` command creates unfilled rectangles (see also [add frect](#)). The `add rect` command creates unfilled rectangles. Unfilled rectangles are used to represent the route keepin, the package keepin, and all other non-etch/conductor rectangles.

Rectangles added to the Etch/Conductor class represent etch/conductor on the design. The `plot` command writes line-plot commands to the photoplot file to fill that area on that layer. Since a major use of etch/conductor rectangles is to distribute a voltage over an area on a layer, a net name (voltage) is associated with each such rectangle.

When you add a rectangle as etch/conductor, a dialog box prompts you for the name of the net with which the rectangle is to be associated. Thereafter, you can attach connect lines to the rectangle so it is physically attached to its net. The `connect` command lets you make the connection because the rectangle is logically on that net.


 You can verify the net of any etch/conductor rectangle by running `show element` on the rectangle.

Related Topics

- [Adding a Rectangle](#)
- [Adding a Room](#)

Add Rect Command: Options Panel

Access Using

- Menu path: *Add – Rectangle*
- Toolbar Icon: 

<i>Line font</i>	Defines the line pattern used in creating the segment. The choices are Solid , Hidden , Phantom , Dotted , and Center . The default is Solid .
------------------	--

The line pattern types are:

Solid	_____
Hidden	- - - - -
Phantom	_____ - -
Dotted	- . - . - . - .
Center	_____ - - _____

Line fonts, other than Solid, are allowed on the following Class/Subclasses:

- DRAWING FORMAT/All user defined subclasses
- MANUFACTURING/NCDRILL_LEGEND
- MANUFACTURING/All user defined subclasses
- PACKAGE GEOMETRY/ASSEMBLY_TOP
- PACKAGE GEOMETRY/ASSEMBLY_BOTTOM
- PACKAGE GEOMETRY/All user defined subclasses
- BOARD GEOMETRY/OUTLINE
- BOARD GEOMETRY/ASSEMBLY_NOTES
- BOARD GEOMETRY/ DIMENSIONS
- BOARD GEOMETRY/ASSEMBLY_DETAIL
- BOARD GEOMETRY/All user defined subclasses

<i>Draw Rectangle</i>	Set to draw a rectangle by specifying two opposite corners, either by clicks or by typing in the coordinates. Set by default.
<i>Place Rectangle</i>	Set to place a rectangle by specifying the <i>Width</i> and <i>Height</i> of the rectangle and then clicking to place the rectangle.
<i>Width</i>	Specify the width of the rectangle to be placed. Available only if <i>Place Rectangle</i> is set.
<i>Height</i>	Specify the height of the rectangle to be placed. Available only if <i>Place Rectangle</i> is set.

Related Topics

- [Adding a Room](#)

Adding a Rectangle

To add a rectangle to your design, follow these steps:

1. Run `add rect`.
Alternately, choose *Add – Rectangle*.
2. Set or verify the Class and Subclass for the rectangle in the *Options* panel.
3. Specify a corner of the rectangle by typing a value on the command line, such as `x 200 345`, for example, or move the cursor to the position you want as the corner, and left click.
If you drew the rectangle on class ETCH/CONDUCTOR, a data browser appears that displays a list of nets; choose a net to attach to the filled rectangle and click *OK*.
4. Specify the opposite corner that defines the rectangle by typing a coordinate. For example, type `x 75 690`, or left click again to choose the opposite corner.
5. When all rectangles are complete, right click and choose *Done* from the pop-up menu.

Changing Class and Subclass of Rectangle

You can however, change the class and subclass of the rectangle after addition. Select the rectangle and right-click to choose *Change class/subclass* command. Choose a new class and subclass from the list appears.

Changing font of Rectangle

You can change the line pattern used in creating the rectangle. Select the rectangle and right-click to choose *Change Line Font* command. Choose a new pattern from the list appears.

Related Topics

- [add rect](#)
- [Moving Elements to other classes](#)

Adding a Room

Prior to adding a room, turn on the layers that display the room information. Once you have added a room, you then assign it a name.

1. Run `color192`. The Color dialog box appears.
2. Select BOARD GEOMETRY.
3. Locate the TOP_ROOM subclass under the BOARD GEOMETRY column.
4. Toggle the TOP_ROOM layer ON. If you prefer a different color for this subclass, you can also set the color at this time.
5. Click OK to close the Color dialog box.
6. Click the *Options* panel to bring it forward.
7. Run `add rect`.
Alternately, choose *Add – Rectangle*.
The *Options* panel displays two fields for Active Class and Subclass. The top box is the Class and the lower box is the Subclass.
8. Set the *Options* panel as follows:
 - Class = BOARD/SUBSTRATE GEOMETRY
 - Subclass = TOP/SURFACE_ROOM, BOTTOM/BASE_ROOM, or BOTH_ROOMS
9. Click and drag the left mouse button from left to right in a downward direction to draw the rectangle, then right-click and choose *Done* from the pop-up menu that appears.

Examples

See [add rect](#) for examples of how to create rectangles.

Related Topics

- [add rect](#)
- [Add Rect Command: Options Panel](#)

add ruler

The `add ruler` command allows you to create rulers in the design for measuring distances between design objects. When you run the *add ruler* command, you can create static rulers in the design window.

Access Using

- Menu path: *Display — Add Ruler*

Creating Static Rulers

If you want to create static rulers, follow these steps:

1. From the *RF Module* menu, choose *Ruler* or type `add ruler` at the command prompt. You can also click the *Add Ruler* toolbar button.
The *Options* panel changes to show the options for the ruler function.
2. Specify the options you want to apply to the ruler:
Line lock: The *Line lock* parameter controls the angle of the ruler segment(s). Supported values are: `off`, `45 °`, and `90 °`. Setting line lock to `off` creates a single ruler segment from the first pick to the next pick. Setting line lock to `45 °` creates multiple segments leading from the first pick to the next pick, each at multiples of `45 °`. Setting line lock to `90 °` behaves similarly, only in multiples of `90 °` rather than `45 °`. The default value is the value last used in any command that uses the line lock (`add ruler`, `add line`, `add connect`).
The rubber band shown on the cursor automatically reflects the line lock setting. You can only create straight ruler segments; arc ruler segments are not supported.
 - *Major division spacing:* The *Major division spacing* parameter adjusts the spacing between the markers on the ruler. These divisions are the only divisions that receive text labels, with every major division receiving exactly one text label. The measurement is interpreted in database units. The default value is the value that was last used in the previous command session. If the command has not been run yet, then a default value is chosen based on the grid settings. If you change the major division spacing, the minor division spacing is also changed so that there will always be ten minor divisions in each major division. Changing the major division spacing will also change the currently selected text block so that the largest possible text size is selected by default.
3. Click on an object in the design to begin the ruler.
A rubberband attaches to the cursor as you drag the ruler across the design window to its ending point.
4. Click a second time on the object you want to measure to.
The ruler appears with incremental measurements displayed in the default units.
5. Click again to continue adding ruler segments to this ruler instance.
Additional ruler segments appear between each endpoint, creating a multi-segment ruler.
6. To start a new ruler instance, right-click and choose *Next* to continue adding more rulers between objects.

✓ At any point during the command, you can right-click and choose *Oops* to undo the last pick or subcommand. Right-click and choose *Cancel* to delete the rulers that you added during the command session. Right-click and choose *Toggle* to switch the ordering of the line segments when using the `45 °` or `90 °` line lock options. (Toggle does not work if line lock is turned off.)
7. Right-click and choose *Done* to finish and save the parameters you set.
This ends the Rulers session. Any rulers you created are added to the design.

Controlling Display of Static Rulers

To disable the display of static rulers, perform the following steps:

1. From the *Display* menu, choose *Color/Visibility*.
The *Color* dialog box appears.
2. Select *Geometry*.
3. Locate the subclass *Ruler* under the *Substrate Geo* class and disable the check box.
4. Click *Apply* and *OK* to exit the dialog box.

add spacer

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

The `add spacer` command lets you create spacer symbols in real time and add them to a die stack. You can create new spacer symbols or seed the Add Spacer dialog box with an existing symbol and change the name and modify parameters to create a new one.

Spacers represent blocks of insulating material or adhesives used between die-stack objects to provide necessary clearance or adhesion to each other. Using spacers, you can accurately model the true manufactured height of a die stack. Place spacers on named non-substrate DIELECTRIC layers and at orthogonal rotations of 0, 90, 180, or 270 degrees.

The layout editor automatically attaches the LOCKED property to a spacer so that you cannot accidentally edit (move, delete, rotate) the symbol children, for example, place-bounds, assembly-rectangles, and so on. Although you can edit this property, it is recommended that you do not as corruption can occur if symbol children are edited. Whenever you update the spacer, the layout editor automatically adds the property to the spacer if you have removed it.

Building the Symbol

Before adding a spacer to the die stack when you build the symbol in the Symbol Editor, add the following to the mechanical symbol (.bsm):

- A filled rectangle on PART_GEOMETRY/PLACE_BOUND_TOP
- A filled rectangle on CONDUCTOR/TOP class and subclass
- Ref ID text on REF_DES/ASSEMBLY_TOP
- A rectangle on the PART_GEOMETRY/ASSEMBLY_TOP class and subclass (optional)

Specifying Properties

When you create a spacer symbol in the symbol editor, you can specify properties for a spacer's thickness, material, and part number using the drawing properties (`property edit` command). Each spacer symbol instance in a package design inherits these properties.

You need to enter valid values for *Material Name* and *Thickness* before APD can place the symbol. The property names are:

- DIELECTRIC_THICKNESS, a number, for example, 100.00DIELECTRIC_MATERIAL, a material existing in the material file (`mcmmat.dat`), for example, PHENOLIC
You can invoke the Material Browser by clicking the ... button that follows the *Material* text box in the Add Spacer dialog box.
- PART_NUMBER, an alphanumeric string, for example, 1ZX-256X-CL4

The `add spacer` command does not generate a log file.

Related Topics

- [Creating Spacer Symbols](#)

Add Spacer Dialog Box

Access Using

- Menu path: *Add – Spacer*

The Add Spacer dialog box appears when you run the `add spacer` command.

<i>Ref ID</i>	Specifies the reference designator of the spacer. If you place multiple instances of the spacer, this value increments to a unique value after you place each spacer symbol. You can edit this value before placing the next symbol instance.
<i>Symbol Name</i>	Specifies the symbol name of the spacer. You can: <ul style="list-style-type: none"> • Use an existing symbol. • Create a new symbol. • Use an existing symbol to seed the dialog box with its values. <p>If you are using an existing symbol to seed the dialog box with its values, click ... to display the Select Symbol dialog box and find the directory where the library of symbol names is located. After you select the symbol and the data appears in the dialog box, change the symbol name and then modify the values.</p> <p>APD creates a new symbol definition in the current design only. It does not create a <code>.dra</code> or <code>.bsm</code> file. To create these files, use the dump libraries command.</p>
<i>Part Number</i>	Specifies the alphanumeric part number of the spacer used in the Bill of Materials (optional).
<i>Material</i>	
<i>Name</i>	Specifies the name of the spacer material. Click ...to display the Select Material dialog box and browse the Materials Editor for the selection of a spacer material. For a description of this dialog box, see the <code>define materials</code> command in the <i>Allegro PCB and Package Physical Layout Command Reference</i> . To edit a material, use the define materials command.
<i>Thickness</i>	Specifies the thickness of the spacer.
<i>Dimensions</i>	
<i>Length</i>	Specifies the length of the new symbol definition.
<i>Width</i>	Specifies the width of the new symbol definition.
<i>Derive from</i>	Specifies dimensions in terms of the die. You can select a die from the list. The <i>Length</i> and <i>Width</i> field are not available if this option is selected.
<i>Shrink factor</i>	Specifies the factor that is used to derive the dimension of the spacer from the size of the die specified in <i>Derive from</i> .
<i>Placement</i>	
<i>Layer</i>	Use the drop-down list to specify the non-substrate DIELECTRIC layer on which you are placing the spacer.
<i>Rotation</i>	Use the drop-down list to specify the angular rotation of the spacer.
<i>OK</i>	Saves the placements and dismisses the dialog box.
<i>Place</i>	Enables placement of instances of the spacer in the design if you completed all the fields in the dialog box (<i>Part Number</i> is optional). APD places the symbol on the cursor and places multiple instances of the spacer when you pick an X, Y location in the Design Window or type X,Y coordinates at the console window prompt.
<i>Cancel</i>	Removes the placements and dismisses the dialog box.
<i>Help</i>	Displays the Help Window.

Creating Spacer Symbols

You can either create a mechanical symbol for the spacer in the symbol editor as part of a spacer library and add it to the die-stack editor using this command, or create a spacer in real time.

To add a spacer:

1. Run the `add spacer` command.
2. Complete the fields in the Add Spacer dialog box.
3. Click *Place* to place an instance of the spacer in the design.
The symbol appears on the cursor.
4. To place one instance of the spacer, either pick an X, Y location in the plan view or type an X, Y coordinate at the console window prompt, for example, x 2500 3000.
The dialog box remains open.
5. To add another instance of the same spacer, edit the value of *Ref ID* (or use the default value provided) and change the layer as required, then click *Place* again.
6. Click *OK* to save the placements and close the dialog box.
or
Click *Cancel* to remove the placements and close the dialog box.

Related Topics

- [add spacer](#)

add tangent line

The `add tangent line` command creates lines tangent to an existing circle or arc segments.

Access Using

- Menu path: *Manufacture – Drafting – Add Tangent Line*

Adding Tangent Lines to an Arc or a Circle

Perform the following steps to add a line which is tangent to an existing arc or circle in the design:

1. Choose *Manufacture – Drafting – Add Tangent Line* or run the `add tangent line` command.

OR

1. Set *General Edit* application mode and select a circle or an arc segment. Right-click and choose *Drafting – Add Tangent Line*.
2. Select a circle or an arc segment as a first element.
3. Select another circle or an arc segment as a second element.
All the possible tangents between the two elements are displayed.
4. Click to add the tangents.
5. Right-click and choose *Next* to continue or *Done* to complete the operation.

add taper

The `add taper` command generates fillets at the junction of two clines of different width. Before executing the command, set the parameters that govern tapering in the *Fillet and Tapered Trace* dialog box. To open the dialog box, choose *Route – Teardrop/Tapered Trace – Parameters* ([gloss param fillet](#) command).

You cannot run this command if the *Dynamic* option is enabled in the *Fillet and Tapered Trace* dialog box. No fillets are generated in areas specified as NO_GLOSS.

Access Using

- Menu path: *Route – Teardrop/Tapered Trace – Add Tapered Trace*

The only configurable options for this command are the active class and subclass.

Adding Tapers

Perform the following steps to add a taper at the junction of two clines with different widths:

1. Choose *Route – Teardrop/Tapered Trace – Add Tapered Trace* (`add taper` command).
The *Options* panel displays the active class and subclass. The *Find* filter defaults to *Nets* as the active design object.
2. If you are performing the operation on multiple traces, choose the *Temp Group* or *Window Select* from RMB menu.
3. Right-click and choose *Done* or *Complete* from the pop-up menu.

add testpoint

The `add testpoint` command lets you create a testpoint on a pin or via, or assign a testpad to cline segments.

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

Prior to using the command, set relevant parameters using the *Edit testprep parameters* button on the *Mfg Applications* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* ([prmed](#) command)

Elements ineligible for use with the command generate a warning and are ignored. Valid elements are:

- Cline segments
- Vias
- Pins

Adding Testpoints

Perform the following steps to add testpoints to your design:

1. Hover your cursor over the element to which to add a testpoint or testpad. The layout editor highlights the element and a data tip identifies its name.
2. Right-click and choose *Add Testpoint* from the pop-up menu.
The layout editor adds the testpoint or testpad as appropriate.

add text

The `add text` command creates free-form text on the design. Use this command to write simple notes and otherwise annotate the design.


The `add text` command does not let you enter an exclamation point (!) in your database, since `extracta` uses that character as a field delimiter. Be aware of the possible consequences of this condition if you read into your database a file that contains an exclamation point.

Related Topics

- [label device](#)
- [label refdes](#)
- [Adding Text to a Design](#)
- [Assigning a Room Name](#)

Add Text Command: Options Panel

Access Using

- Menu path: *Add – Text*
- Toolbar icon: 

<i>Mirror</i>	Specifies whether the text should be added in mirrored mode. If unchecked, the text enters from left to right. If checked, the text enters right to left and mirrored.
<i>Marker size</i>	Specifies the size, in user units, of the marker that identifies the location of the text.
<i>Rotate</i>	Determines the angle of rotation. Type in a number between 0.000 and 360.000 or choose from the pop-up menu (options are: 0, 45, 90, 135, 180, 225, 270, and 315).
<i>Text block</i>	Indicates the text block you want to use for the text you are entering.
	A text block defines the size and spacing of the text you add to the design. Using the <code>define text</code> command, you can define up to 16 text blocks. The default is <i>1</i> .
<i>Text name</i>	Specifies the name of the text block.
<i>Text just</i>	Specifies where the selected text should align, relative to the text marker. The choices are: Left (default), Right , and Center .


Related Topics

- [define text](#)
- [Assigning a Room Name](#)

Adding Text to a Design

To add a text to a design, follow these steps:

1. Run the `add text` command.
2. Complete the *Options* panel.
3. Position the cursor and click at the location for the text.
4. Enter the text in the design window.
 - Limit text lines to 200 characters, including spaces.
To correct errors, press *Delete* or *Backspace*.
 - Press *Enter* to start a new line of text with line spacing set by the parameter block.
5. When you have entered all text required for the current point, click right to display the pop-up menu, and choose *Done*.

 To import a text file into the design, run `add text`, click right, and choose *Read from file*.

Changing Class and Subclass of Text

You can however, change the class and subclass of the text after addition. To do this, follow these steps:

1. Select the text and right-click to choose *Change class/subclass* command.
2. Choose a new class and subclass from the list appears.

Related Topics

- [Moving Elements to Other Classes](#)
- [text edit](#)
- [define text](#)
- [add text](#)

Assigning a Room Name

After you create a room, you give a unique name to each room by adding text to it, and then assign that room name to the appropriate components with the ROOM property.

1. Run the `add text` command.
The message area prompts:
`Pick an element to attach text to`
2. Click on the rectangle you created.
The rectangle appears highlighted. The message area prompts:
`Pick text location`
3. Set the *Options* panel as follows:
 - Class = BOARD/SUBSTRATE GEOMETRY
 - Subclass = TOP/SURFACE_ROOM, BOTTOM/BASE_ROOM, or BOTH_ROOMS
4. Click left to select the rectangle to name.
5. Click left inside the highlighted rectangle to indicate where to place the room name.

If required, use the *Options* panel to specify how the text appears in a design. For example, to rotate text, enter an angle in the Rotation field.

6. Type the room name.

The name that you assign to a room lets you identify it as the area for placement, ping, routing, or placement evaluation.

7. Choose *Done* from the pop-up menu.

Related Topics

- [add text](#)
- [Add Text Command: Options Panel](#)

add vertex

The `add vertex` command Inserts a new vertex within a segment.

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

Prior to using the command, set relevant parameters in the *Edit Vertex* section of the *Route* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* ([prmed](#) command). You may also set them by right-clicking to display the pop-up menu from which you may choose:

- Design Parameters to access the Design Parameter Editor
- Options

Changing a parameter using either of these pop-up menu choices automatically updates the *Options* panel as well.

Valid elements are:


- Cline Segs
- Other Segs

Related Topics

- [Adding a Vertex](#)

Add Vertex Command: Options Panel

Access Using

- Toolbar icon: 

When you access the command in the pre-selection use model from the right mouse button pop-up menu, the *Options* panel is not available for you to change settings.

<i>Active Class and Subclass</i>	The upper drop-down list box displays the current class; the lower drop-down list box, the current subclass with choices for modifying the value.
<i>Net</i>	<p>Identifies the net assigned to the element you select. If no net is assigned, the value is NULL NET.</p> <p>To the left of the field name is an indicator for the nets. When you are routing a single net, the indicator shows one net. When you are performing differential pair routing, the indicator shows two nets. If you are performing group routing, the indicator shows multiple nets. If you are in single trace mode in differential pair routing or group routing, the indicator shows only the control trace highlighted. The field always shows the net name of the control trace (for both differential pairs or single traces) and never the differential pair name.</p>
<i>Bubble</i>	<p>Controls any automatic bubbling (moving of existing connections) to resolve DRC errors with the following options:</p> <p><i>Off</i>: The clines you route start at the location you indicate, and no bubbling occurs. DRC flags all clearance violations with error markers.</p> <p><i>Hug Only</i>: Where possible, the routed cline contours around other etch/conductor objects to avoid spacing DRCs. Other etch/conductor remains unchanged.</p> <p><i>Hug Preferred</i>: Where possible, the routed cline contours around other etch/conductor objects to avoid spacing DRCs. If not possible, the layout editor tries shoving other etch/conductor objects to open routing paths.</p> <p>Note: This method is more aggressive than <i>Hug Only</i>.</p> <p><i>Shove Preferred</i>: Where possible, the routed cline pushes and shoves other etch/conductor objects to avoid spacing DRCs. If not possible, the layout editor attempts to hug other etch/conductor objects.</p>

A Commands

A Commands--add vertex


<i>Shove vias</i>	<p>Allows the bubble functionality in shove mode to move vias when you are editing etch/conductor. It is only active when <i>Bubble</i> is enabled. The following are the options</p> <p>Full: Vias are shoved in a shove-preferred manner. Any new or edited etch/conductor always shoves vias out of the way.</p> <p>Minimal: Vias are shoved in a hug-preferred manner. Vias are not moved unless there is no way to draw a connect line around them.</p> <p>Off: Vias are not shoved</p>
<i>Clip dangling clines</i>	<p>Active for shove-preferred mode and controls whether the layout editor clips back dangling clines to fix DRC errors. When disabled, the dangling cline endpoints remain unchanged, and the layout editor corrects the DRC errors, if possible, by bubbling the new cline around the dangling endpoints (similar to hug-preferred mode).</p>
<i>Smooth</i>	<p>Active when you set the <i>Bubble</i> field to hug- or shove-preferred mode and controls whether smoothing occurs on the cline to minimize segments between the start and finish points. Smoothing occurs dynamically as you move the mouse on cline segments close to the segment you selected.</p> <p>Performance with the <i>Smooth</i> option active may be somewhat slower than when it is inactive.</p> <p>These are the choices:</p> <p><i>Minimal</i>: Executes dynamic smoothing to minimize unnecessary segments.</p> <p><i>Full</i>: Executes more extensive smoothing to remove any unnecessary jogs.</p> <p><i>Off</i>: Disables smoothing.</p> <p>Note: Full smoothing does not smooth the cline you are adding back to its source. Rather, it smooths the newly created etch/conductor back to your last pick. Additionally, parts of other clines that are shoved during this procedure may also be smoothed.</p>
<i>Allow DRCs</i>	<p>Specifies that design rules can be violated to make a connection. If <i>Bubble</i> is disabled, the vertex is set at a point between the last good point and the current point that does not cause a DRC error.</p>
<i>Allow Gridless</i>	<p>Specifies that the etch (or conductor) can go off the routing grid. Gridless routing lets the layout editor add connections at maximum density while accommodating varying design rules and line widths. The DRC minimum space separates objects.</p> <p>When <i>Bubble</i> is disabled, the <i>Allow Gridless</i> field controls the removal of a small segment at the end of the new route when in <code>add connect</code> mode. Normally, if the last segment is small, the layout editor does not add it (to avoid adding a little jog). If <i>Allow Gridless</i> is off, the layout editor adds the segment.</p>

Adding a Vertex

Perform the following steps to add a vertex within a segment:

1. Hover your cursor over the segment to which to add a vertex. The layout editor highlights the element and a data tip identifies its name.
2. Right-click and choose *Add Vertex* from the pop-up menu.

✔ Consider using the right mouse button pop-up menu option *Snap pick to*, which snaps the connect line to database elements such as segment vertex or grid point or intersection and so on.

⚠ The vertex cursor appears when you hover over a vertex. 

The layout editor adds the vertex to the segment.

Related Topics

- [add vertex](#)

add xshape

The `add xshape` command adds closed, cross-hatched filled shapes or elements only on ETCH/CONDUCTOR subclasses of your design. The shapes are made from a continuous series of line/arc segments and filled with a solid field of copper. You create line segments by continuous mouse clicks or by entering coordinates at the command line. The shape between the first and last points is completed when you choose *Done* from the pop-up menu. At that point, the UI changes to the Shape editor.

Related Topics

- [Adding Closed, Cross-hatched Filled Shapes, or Elements To a Design](#)

Add Xshape Command: Options Panel

<i>Line lock</i>	Specifies whether the shape is drawn with straight lines or with arcs and defines the angle of the corner when a line segment changes direction. The choices are <i>Line</i> , <i>Arc</i> and Off, 45, and 90.
<i>Line width</i>	Specifies the width of the line in user units.
<i>Line font</i>	Is not active in <code>add xshape mode</code> .

Adding Closed, Cross-hatched Filled Shapes, or Elements To a Design

Follow these steps to add closed, cross-hatched filled shapes to your design:

1. Run the `add xshape` command.
2. Configure the *Options* panel in the Control Panel.

⚠ You can create filled shapes only on ETCH/CONDUCTOR subclasses.

3. Ensure that the subclass you are drawing the shape on is visible.
4. Left click at the vertices of the shape outline that you want to create.
5. When you are ready to complete the shape, do one of the following:
 - Close the shape by picking the starting point again (closing the shape outline), and then click right and choose *Done* from the pop-up menu.
 - Click right and choose *Done* from the pop-up menu.

⚠ Note: When the shape outline is complete, the design area changes from layout editor to the shape editor. You can only edit one shape at a time while in the shape editor. The active shape is the last shape selected in your layout before you entered the shape editor.

6. Attach the shape to a net using one of the following techniques
 - Choose *Edit – Change Net (Pick)* and pick any object already associated with the net you require, such as a pin, connect line, via, or another shape.

-or-

- Choose *Edit – Change Net (Pick)* and enter the net name, at the fill-in, with which to associate the shape. Then click *Close*.

This makes the shape part of the net you select. Until you do this step, an etch/conductor shape is on a dummy net (which means no net). Non-etch/conductor shapes are never on a net.

7. Continue to define the shape, if need be.
8. When the shape meets your requirements, run `shape fill`.
The shape fills and you return to the layout editor. DRC is performed on the shape during the shape fill process.

Note: Choosing `shape fill` is the only method to exit the Shape editor.

Setting the Shape Parameters

After you create a shape outline, you must specify the shape parameters. The parameters determine the following:

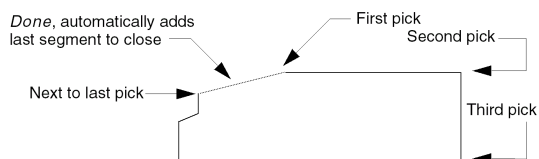
- The type of shape fill
- How voids are generated
- Void clearances
- How thermal-relief connect lines are generated

For details on how to specify these parameters, see [shape param](#).

Example

Figure 1-2 shows how to create a shape using mouse picks.

Figure 1.2: Creating a Shape Using Mouse Picks



Related Topics

- [add xshape](#)

advanced highlight

The `advanced highlight` command extends the capabilities of standard `assign` and `color` commands by allowing you to set an object's color based on characteristics of the object.

Related Topics

- [Setting Object Color based on Characteristics:](#)

Advanced Highlight Dialog Box

Access Using

- Menu path: *Display – Advanced Highlight*

<i>Object Type</i>	Select the type of object you want to highlight: PINS, FINGERS, or CLINES.
<i>Attribute</i>	Select the attribute on which you base your coloring.
<i>Value</i>	Select the value for the specified attribute. Based on the attribute selected, this list shows the various selections available.
<i>Color</i>	Displays the highlight color. Click a color in the Available Colors box to change this color.
<i>Dehighlight matching items</i>	Check this box to dehighlight objects matching the criteria in Attribute and Value. The default setting is off.
<i>Temporary highlight</i>	Check this box to use temporary highlighting instead of permanent highlighting. This field (off by default) maintains the existing capability of the obsolete <code>bond finger hilite</code> command.
<i>Available Colors</i>	This grid shows all the color choices currently defined in the design. Clicking a cell in the grid changes the highlight color, and is reflected in the Color field.
<i>Update</i>	Click to update the highlighting in the design based on the current dialog box settings.
<i>Ok</i>	Click to exit the command and commit any changes made to the design.
<i>Cancel</i>	Click to exit the command without making changes.
<i>Help</i>	Click to invoke context-sensitive Help for this command

Setting Object Color based on Characteristics:

Perform the following steps to set object color based on object attributes:


1. Run the `advanced highlight` command.
2. From the *Object Type* list, select the object.
3. From the *Attribute* list, select the attribute that you want to filter.
This updates the options in the Value field, based on the current design's contents.
4. Select an option in the Value field.
5. Either choose a color from the grid of available colors or check the Dehighlight matching items box to dehighlight the selected items in the Design Window.
6. If appropriate, either window-select or use the right-click Temp Group command in the Design window to select a specified area.
You can select specific components to narrow the selection for highlighting. If you do not select any components, the highlight is applied across the entire design based on the settings in the dialog box.
7. Click *Update* to apply the highlighting to these items or to the entire design if you did not make a window selection.
To change the results, right-click and choose Oops. Then adjust the Find Filter and perform the task again.
8. Follow these steps until you color the entire design appropriately.
9. Click *OK* to commit the color changes to the database.

Related Topics

- [advanced highlight](#)

Advanced Package Router

Advanced Package Router (APR) is an any-angle topological auto-router that routes constraint driven flip-chip designs with high completion. It is initially targeted for single-die, flip-chip style designs. It supports differential pair and bundle based routes. APR employs unique algorithms and techniques with the objective of delivering high completion rates.

 Only available for Allegro X Advanced Package Designer (APD) with the *Advanced Package Router* license option, on the Windows platform.

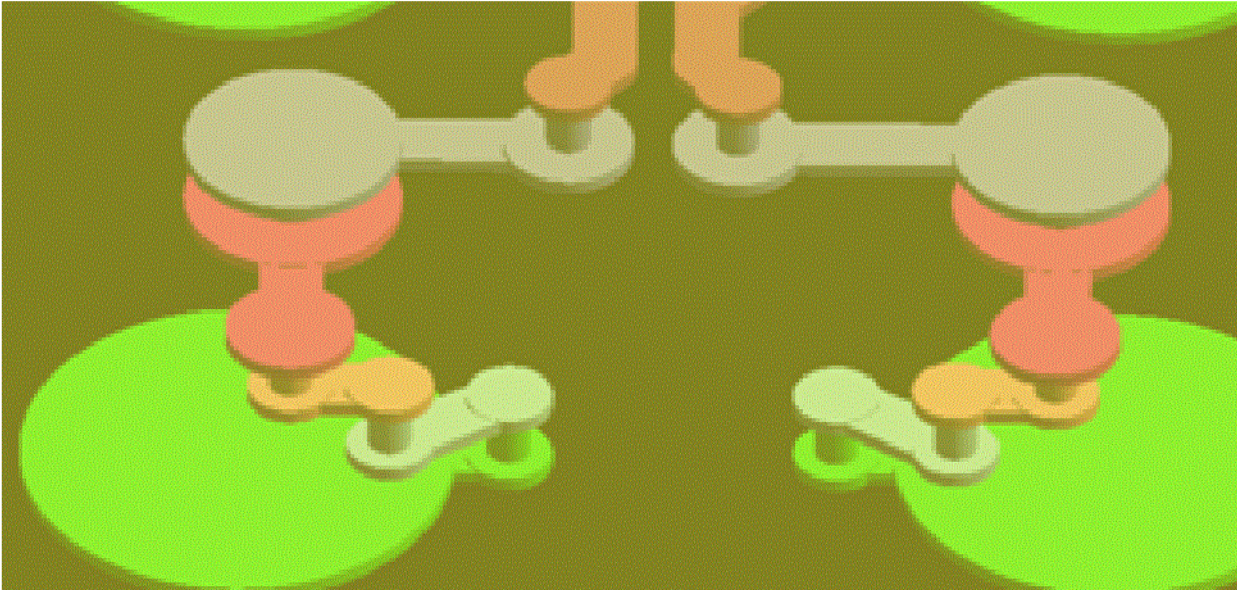
Related Topics

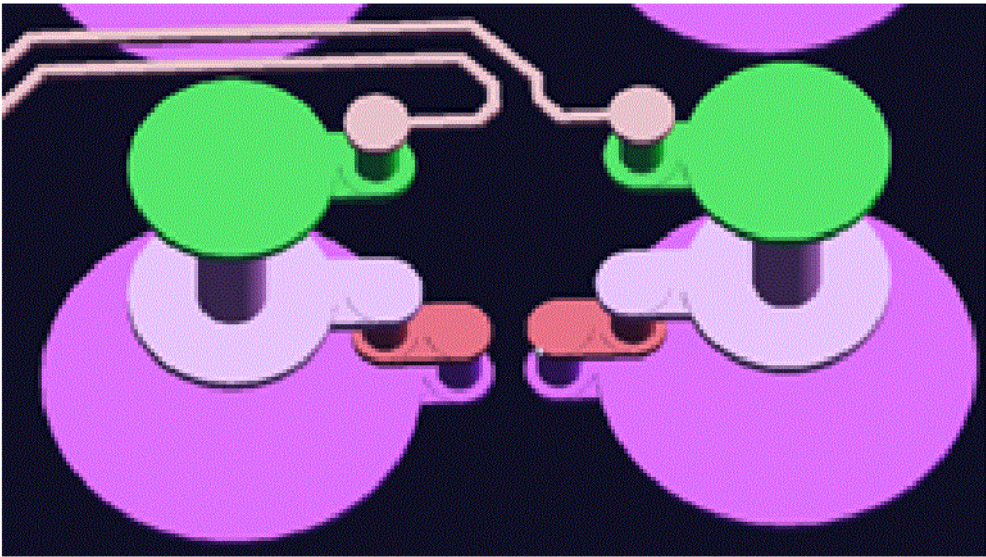
- [Advanced Selection Filtering Dialog Box](#)
- [Routing Constraint Driven Flip-Chip Designs](#)
- [Filtering Nets and Pins](#)

Advanced Package Router Dialog Box

Access Using

- Menu path: *Route – Advanced Package Router*
- Command line: `pkg_router`

Routing Layers	
	Select the routing layers from the table. Edit the Ratio and Detour Tolerance values, if needed.
Detour type	<div>Select the detour type:<ul style="list-style-type: none">◦ <i>Ratio</i>: To set Manhattan length ratio. Wires are allowed to stay on the layer if their lengths are within the set ratio on their Manhattan ler◦ <i>Length</i>: To set maximum Manhattan length. Wires are allowed to stay on the layer if their lengths are less than or equal to the set lengt Manhattan length.</div> <div>Default is <i>Ratio</i>.</div>
Options	
Via method:	<div>Define the via pattern from the list of fixed patterns: <i>Spiral</i>, <i>Stagger</i>, and <i>Staircase</i>. Default is <i>Spiral</i>.</div> <div>The different via patterns are illustrated by the following figures:</div> <div><ul style="list-style-type: none">• Spiral</div> <div><ul style="list-style-type: none">• Stagger</div>



- staircase



A Commands

A Commands--Advanced Package Router

Maximize ball vias	Select to add up to 4 vias to the BGA pins, if possible. Not selected by default.
Power/Ground core via count	Specify the number of vias to be dropped for a net that is selected for routing and that has Power/Ground properties set. Default is 1.
Allow routing under discretes	Select to place a temporary void under discrete to prevent wires and vias. Not selected by default.
Length tuning	Select to match the length of differential Pairs and bundles in groups that have Signal Integrity rules set for Max skew or CLK Nets to follow. Once this field is enabled, you can tune differential routing further by specifying values for <i>Gap</i> , <i>Max amplitude</i> , <i>Min amplitude</i> , and <i>Miter</i> . If you specify any value, the default values will be used. You can select <i>Only tandem tuning of pairs allowed</i> to ensure differential pairs are routed together or side-by-side. Not selected by default.
Routing Options	
Routing goal	Select a routing goal: <ul style="list-style-type: none"> • <i>100% Completion</i>: To try to complete the nets. This is selected by default. • <i>Estimate Only</i>: For topology route results with no post processing. • <i>Fan-out Only</i>: To create fan-out vias on bump and BGA pins. • <i>Maximum Passes</i>: To specify number of passes the router should stop at for each layer.
Strategy	Select a routing strategy: <ul style="list-style-type: none"> • <i>Maximize for completion</i>: To maximize the completion rate without conflicts or overlapping traces in final results. This is selected by default. • <i>Maximize for flow</i>: To include all nets in final results and with minimal conflicts. • <i>Allow routing vias other than escapes</i>: To route vias other than escapes
Net Options	
Nets to route	Specify the nets to be routed: <ul style="list-style-type: none"> • <i>All Nets</i>: To route all nets listed in the Available nets column of the table. This is the default. • <i>Signal Nets</i>: To route only signal nets. • <i>Selected Nets</i>: To route all nets in the Selected nets column. Click a net in the Available nets column to add it to the Selected nets column. • <i>Non-selected</i>: To route nets listed in the available nets column but not listed in the Selected nets column.
Route	Click to start routing with the specified settings.
Cancel	Exits Advanced Package Router without making any changes.
Help	Displays help for Advanced Package Router.

Related Topics

- [Routing Constraint Driven Flip-Chip Designs](#)
- [Filtering Nets and Pins](#)

Advanced Selection Filtering Dialog Box

The *Advanced Selection Filtering* option lets you filter nets or pins, or both, when you run the following commands:

- [auto assign pinuse](#)
- [auto assign net](#)
- [assign route layer](#)
- [assign plating layer](#)
- [wirebond select](#)
- [wirebond add](#)


Access Using

Select the *Use advanced selection filtering* option and then select the nets or pins in the design, the Advanced Selection Filtering dialog box appears. The tree view displays top-level items (the nets) that you can click on to see the pins associated with them.

By default, the *Filter* field displays an asterisk (*) which means that the list displays all the selected nets. You can modify this field to display a list that is easier for you to manage.

Clicking the net name automatically selects or deselects all the associated pins.

The following table describes the dialog box:

<i>Filter String</i>	<p>Lets you manage the display of the tree. When you modify this field, it does not change the selected state for any items in the design; it only changes those that are displayed in the tree view.</p> <p>The default for this field displays an asterisk (*) to show all selected nets.</p> <div> This field is case-sensitive.</div>
<i>Filter Tree</i>	<p>Displays all the items that you selected, organized by net, and filtered, based on the value of the <i>Filter String</i>. Checking the box next to an item includes it in the filtered set. Unchecking the box removes it from the set. You can use All to toggle the visibility for all items to the same state in one click. By default, all items originally selected in the design are checked.</p>
<i>OK</i>	<p>Closes the Advanced Selection Filtering dialog and returns control to the calling command. All those items that are currently checked in the tree are passed back as the new selection set.</p>
<i>Cancel</i>	<p>Exits from the Advanced Selection Filtering dialog box and indicates to the calling command that you did not select anything (canceled).</p>
<i>Help</i>	<p>Invokes context-sensitive entry for this command.</p>

Related Topics

- [auto assign pinuse](#)
- [auto assign net](#)
- [assign route layer](#)
- [assign plating layer](#)
- [wirebond select](#)
- [wirebond add](#)
- [Advanced Package Router](#)
- [Filtering Nets and Pins](#)

Routing Constraint Driven Flip-Chip Designs

Prerequisites

The following must be noted while using APR:

- Power Nets: Voltage properties should be applied to power nets. APR requires the voltage property to have a value to be considered a power net, which has significant impact on routing performance of both signal and power nets.
- Constraints: The APR router will look at and try to comply with physical and electrical constraints as entered in CM. The router performance might be impacted by overly restricted or missing constraints. Underlying constraint modes must be enabled for some APR options to work.
 - SMD Pin Modes-Via at SMD and Physical Mode-Pad 2 Pad Direct Connect mode settings must be turned on in Constraint Modes; if not, users may see little effect from the Snap to Bump Center/Snap to BGA Ball Center options.
 - Physical Mode-Pad 2 Pad Direct Connect mode setting must be turned on in Constraint Modes, or users may not see desired results from the "Snap to Core Via" option.
- Shapes: APR routing will be poor if static etch shapes exist on any layers where routing is required; this includes layers used only for stepping/staggering through to lower layers. Unless the desire is to prevent the router from interacting with a particular shape, dynamic shapes should be used in conjunction with APR.

To route a design using APR, do the following steps:

1. Choose *Route – Advanced Package Router*
The Advanced Package Router window appears.
2. Select the layers to route on and specify the detour values.
3. Specify the via options.
4. Set the routing options.
5. Select the nets to be routed.
6. Click *Route*.
A dialog box appears displaying the status of the routing attempts and completion information.
You can review the routing process by clicking *Pause*. You can then either click *Continue* to resume the routing process or click *Stop* to stop the process.
If you stop the process, you can run post-processing for the results completed till the time you stopped the process.
When routing completes the details of the route pass information is displayed and the routes are sent back to the design.

Related Topics


- [Advanced Package Router](#)
- [Advanced Package Router Dialog Box](#)

Filtering Nets and Pins

The procedure for filtering using the Advanced Selection Filtering dialog box is the same for whichever command you are running.

To filter nets and pins to the following steps:

1. Check the *Use Advanced selection filtering* box.
2. Choose the pins or nets in the design.
3. In the Advanced Selection Filtering dialog box, uncheck any nets or pins that you would like to remove from your initial selection.
4. To manage your list, modify the *Filter* field and then press the `Tab` key.

 This field is case-sensitive.


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

Related Topics

- [auto assign pinuse](#)
- [auto assign net](#)
- [assign route layer](#)
- [assign plating layer](#)
- [wirebond select](#)
- [wirebond add](#)
- [Advanced Package Router](#)
- [Advanced Package Router Dialog Box](#)
- [Advanced Selection Filtering Dialog Box](#)

adv nonstandard fillets

The `adv nonstandard fillets` command updates the fillets to either the tip style or the cline edge style in the design.

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

Advanced fillets can be generated at any time, that is, during or after routing. If a route is modified after an advanced fillet is generated, it will be removed so that orphaned shapes are not created in the design.

Related Topics

- [add fillet](#)
- [gloss param fillet](#)
- [Updating Fillets in the Design](#)

Adv Nonstandard Fillets Command: Options Panel

Access Using

- Menu path: *Si Layout – Advanced Fillets*

The following table describes the fields available in the *Options* panel when you run the `adv nonstandard fillets` command:

<i>Update Selected</i>	Updates the fillets on the selected layers in the design.
<i>Update All</i>	Updates all fillets in the design.
<i>Dynamic</i>	Select this option to dynamically update fillets as the traces they are connected to are moved in the layout.
<i>Allow DRCs</i>	Select this check box to place the fillets even if they result in DRCs to the surrounding items. This check box is selected by default.
<i>Add DRC markers for missing fillets</i>	Select this check box to add a DRC marker if a fillet is not generated for some reason but was required. This check box is selected by default.
<i>Widen short connections</i>	Select this option to widen short connections.
<i>Widen in constraint regions</i>	Select this option to widen short connections in constraint regions.
<i>Display advanced parameters</i>	Select this check box to display the <i>Cline Max Width</i> , <i>Min Angle</i> , and <i>Max Angle</i> columns in the <i>Configuration</i> table.
Configuration	
<i>Layer</i>	Displays the conductor layer on which the fillets are to be created. If pad style is <i>Adjacent</i> , two entries are displayed for each conductor layer, one for the connections in each direction and the other for the connections from top layer to the bottom layer.
<i>Pad</i>	Specifies how fillets are to be generated: Available options: <ul style="list-style-type: none"> • <i>Adjacent</i>: Separate parameter measures to the dielectric opening/hole. • <i>Drill</i>: Single parameter based on the drill hole size for the selected object. • <i>Metal</i>: Single parameter based on the metal pad size on the layer.
<i>Min Length</i>	Specify the minimum length required for the fillet. The default value is 50um.
<i>Cline Max Width</i>	Specify the maximum width beyond which fillets are not required to be generated. This field is blank by default. This field is displayed only if the Display advanced parameters check box is selected.
<i>Min Angle</i>	The minimum angle between the two sides of the pie-shaped fillet. This field is blank by default. This field is displayed only if the Display advanced parameters check box is selected.
<i>Max Angle</i>	The maximum angle between the two sides of the fillet. This field is blank by default. This field is displayed only if the Display advanced parameters check box is selected.
<i>End</i>	Specify the shape of the end of the fillet. Available options: <ul style="list-style-type: none"> • Fillet Tip: Fillet is created with a pointy end. • Cline Edge: Fillets are created with a straight end.
Advanced Fillet Tapered Traces Options	
<i>Tapered Traces</i>	Select this option to enable tapered traces.
<i>Min. Segment Angle</i>	Specify the minimum acceptable segment angle.
<i>Desired Angle</i>	Specify the desired angle for the tapered traces.
<i>Max. Length</i>	Specify the maximum acceptable fillet trace length.

Related Topics

- [add fillet](#)
- [gloss param fillet](#)

Updating Fillets in the Design

To update fillets to either the tip style or the cline edge style in the design, follow these steps:

1. Choose *Si Layout – Advanced Fillets*.
Alternatively, you can also type `adv nonstandard fillets` in the Command window.
2. Set the fillet parameters in the *Configuration* table of the *Options* panel.
3. Do one of the following to update fillets:
 - Click the *Update Selected* button to update the fillets on the layers selected in the *Layers* column.
 - Click the *Update All* button to update all the fillets on the all the layers.


Related Topics

- [add fillet](#)
- [gloss param fillet](#)
- [adv nonstandard fillets](#)

adv thieving

The `adv thieving` command, available in Allegro X Advanced Package Designer (APD), is similar to the `thieving` command available in all Cadence layout editors but has some extra options, such as support for array angles and offset. The two commands let you add a pattern of non-conductive, single-layer figures to areas on the outer layers of a printed circuit board that do not contain copper. You generate the thieving pattern to balance the plating distribution, placing it to avoid interference with the signal quality of adjacent circuits. Use thieving near the end of the design process, prior to artwork generation.

Once you generate a thieving pattern, the results appear in the Padstack Usage Report, available by choosing *Reports – Reports* (`reports` command).

 The `adv thieving` command is available in Allegro X Advanced Package Designer when the *Silicon Layout* option is chosen.

Related Topics

- [Creating a Thieving Pattern](#)

Adv Thieving Command: Options Panel

Access Using


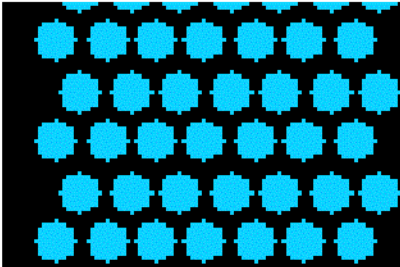


- Menu path: *Si Layout – Advanced Metal Fill*

Thieving patterns adhere to the parameters you specify in the *Options* panel, regardless of DRC rules. The parameters remain in effect until you change them.

<i>Active Class and Subclass</i>	Displays the current parameters and provides a drop-down menu for modifying them. The choices are any subclass of the <i>ETCH/CONDUCTOR</i> class and <i>SOLDERMASK_TOP</i> and <i>SOLDERMASK_BOTTOM</i> subclasses of the <i>Board Geometry</i> class.
<i>Line Lock</i>	Controls whether the segment type is a straight line or an arc. You can also define the corner angle when a segment changes direction. The choices are <i>Line or Arc</i> and <i>Off, 45, or 90</i> .
Thieving Array Parameters	
<i>Thieving Style</i>	Displays the figure style <i>within the thieving pattern</i> . You can choose any one of <i>Circle</i> , <i>Rectangle</i> , <i>Line</i> , or <i>Hexagon</i> . Default is <i>Circle</i> . If you choose <i>Hexagon</i> , the <i>Packed spacing</i> field is available to specify a consistent spacing around a staggered hexagon pattern.
Thieving outline	Displays the type of shape for creating thieving outline. You can choose <i>Shape</i> or <i>Rectangle</i> . The default is <i>Shape</i> .
<i>Size X</i>	Specifies the X dimension of the figures. The value must be a positive integer. If you choose <i>Circle</i> as the thieving style, <i>Size X</i> specifies the diameter. If you choose <i>Line</i> and if X is larger in value than Y, the line will be horizontal; X specifying the length and Y specifying the width.
<i>Size Y</i>	Specifies the Y dimension of the figures. The value must be a positive integer. If you choose <i>Rectangle</i> , equal values create a square figure. If you choose <i>Circle</i> as the thieving style, you cannot edit this field. If you choose <i>Line</i> and if Y is larger in value than X, the line will be vertical; Y specifying the length and X specifying the width.
Seperation X	Specifies the space in X-axis or horizontal direction between the figures in the pattern.
Seperation Y	Specifies the space in Y-axis or vertical direction between the figures in the pattern.
Pitch X	Specifies the center-to-center distance in the X-axis or horizontal direction between the figures in the pattern. If you rotate the array, then the pitch applies to the array before you rotate it.
Pitch Y	Specifies the center-to-center distance in the Y-axis or vertical direction between the figures in the pattern. If you rotate the array, then the pitch applies to the array before you rotate it.
Array angle	Specifies the angle to rotate the thieving array around the first location, either as an angle (<i>Angle</i>) or offset (<i>Offset Nx/Ny</i>) values. <i>Angle</i> is selected by default.
Angle	Specifies the angle by which the array is rotated around the first location. Available if you choose <i>Angle</i> in <i>Array angle</i> . The angle can be from -90 to $+90$ degrees.
Offset	
Nx	Specifies the offset in the horizontal direction or X-axis for rotation. Default value is 0.
Ny	Specifies the offset in the vertical direction or Y-axis for rotation. Default value is 0.
Starting position	Specifies the position from which the array begins. If you specify <i>Upper Left</i> , <i>Upper Right</i> , <i>Lower Left</i> , or <i>Lower Right</i> , it is the corner from which the layout editor applies the X and Y offsets. The default setting is <i>Upper Left</i> .
Offset (X and Y)	Lets you apply a consistent offset across multiple figures in pattern without relying on each specific extents each figure and makes it easier to ensure that figures on adjacent layers do not overlap.
Custom point (X and Y)	Specifies a coordinate (x,y) as the starting point. All figures are offset from this coordinate in the design.
Packed spacing	Specifies the X and Y spacing for a consistent spacing around a staggered hexagon pattern. Available only if <i>Hexagon</i> is chosen for <i>Thieving Style</i> .

A Commands

A Commands--adv thieving

<i>Clearance</i>	<p>Specifies the distance between the thieving pattern and all other objects on the active subclass. The value must be a positive integer. The layout editor uses this value when autovoiding.</p> <div>  When you generate a thieving pattern, it adheres to route or via keepout boundaries within the thieving outline. If a dynamic shape exists within the outline, the thieving pattern clears around it. </div>
Border	
Width	Specifies the width of the border surrounding the thieving area. The border is optional. The value you enter must be zero or greater.
Clearance	Specifies the clearance between the border and the pattern.
Mask layer	Specifies the mask layer.
Rotate pads at array angle	If you check this box, the layout editor rotates the pads it creates at the same angle as specified in the Array Angle field. For example, a square on a 45-degree angle appears as a diamond.
<i>Border Width</i>	.
<i>Staggered Pattern</i>	<p><i>Specifies that every other row appears in an offset pattern, as shown below:</i></p>  <p>Checked is the default. Deselect it to align the pattern in straight rows and columns.</p>
Clip to Route Keepin	<p>Keeps thieving vias within the route keepin area.</p> <p>Specifies the area for thieving. This area is defined as the intersection of the route keepin area and the thieving boundary area.</p>
All etch layers	<p>Thieving is generated for each positive etch layer of the design.</p> <div>  If enabled ignores the current settings for <i>Active Class and Subclass</i>. </div>
All soldermask layers	<p>Thieving is generated for each soldermask layer of the design.</p> <div>  If enabled ignores the current settings for <i>Active Class and Subclass</i>. </div>
Offset layers	Allows thieving to be generated on all etch/soldermask layers at the same time if these options are selected. The pattern of thieving is offset on adjacent layers.

Creating a Thieving Pattern

Perform the following steps to create a thieving pattern:

1. Choose *Advanced WLP – Metal Fill*
Alternately, type `adv thieving` in the Command window.
The *Options* panel changes to display the advanced thieving options. The console window prompt instructs you to enter a thieving outline.
2. Change the parameters in the *Options* panel.
This step is optional, because you can accept the current settings.
3. Outline the area to fill.
4. Right-click to display the pop-up menu and choose *Done*.

 Right-click and choose *Oops* to cancel the last operation or *Cancel* to cancel the last command.

The layout editor automatically completes the thieving process.

Related Topics


- [adv thieving](#)

aibt deletebreakout

The aibt deletebreakout command deletes the interconnect between existing breakouts.

You can access this command in PCB Editor if:

- Design Planning option is enabled
- Flow Planning or Etch Editing application modes is enabled
- Pop-up menu in pre-selection mode

 Selecting an object and entering the command in the Command Window will not produce the desired result.

Valid objects are:

- Bundles
- Bundle Ratsnests

Deleting Interconnect between Breakouts

Follow these steps to delete an interconnect between existing breakouts:

1. Hover your cursor over a flow segment.
The layout editor highlights the segment and a datatip identifies its name.
2. Right-click and choose *Auto-I.Delete Breakout*.
The command starts and an execution dialog box appears showing the status of the command.
3. Click *Cancel* to stop the command.
4. Right-click and choose Done from the pop-up menu to complete the command.

aibt routetrunk

The aibt routetrunk command generates the interconnect between existing breakouts. The trunk router generates interconnects only when the breakout routing exist at both ends of the connection.

You can use this command in the PCB Editor when:

- Design Planning option is enabled
- Flow Planning Application Mode is enabled
- Pre-selection mode is enabled and specific right-click menu option is selected

❗ Selecting an object and entering the command in the Command Window will not produce the desired result.

You can set the command parameters by right-clicking and choosing *Quick Utilities – Design Parameters – Route – Auto-I. Trunk Route* or from the *Setup – Design Parameters – Route – Auto-I. Trunk Route*.

Valid objects are:

- Bundles
- Bundle Ratsnests

The Auto-I. Trunk Route Parameters

Ripup existing	Rips up the existing routing between breakouts. Connections are trimmed back to breakout ends and creates new route patterns depending on the current selection. If a fully routed Bundle is selected, the command only re-routes connections between the breakouts. By default, this option is set to Yes.
Compress	Gathers routing connections, as a group, so that they can be routed together between the breakouts. By default, this option is set to No.

Connecting the Breakout on Both Ends of a Bundle

To connect the breakout on both ends of a bundle, follow these steps:

1. Hover your cursor over a flow segment. The layout editor highlights the segment and a datatip identifies its name.
2. Right-click and choose *Auto-I.Trunk Route*.
The command starts and an execution dialog box appears showing the status of the command.
3. Click *Cancel* to stop the command.

aibt single

The aibt single command operates on a user-defined set of bundles to create a pattern that utilizes optimum channel usage and layer distribution. These pattern are then used by a auto-router.

You can use this command in the PCB Editor when:

- Design Planning option is enabled
- Flow Planning Application Mode is enabled
- Pre-selection mode is enabled and specific right-click menu option is selected

❗ Selecting an object and entering the command in the Command Window will not produce the desired result.

Breakout is the process of creating short routes that are exiting out from under a component (usually BGA) to some pre-set distance, mostly outside the component boundary.

The aibt command generates breakouts on the selected bundle(s) or bundle ratsnest(s). Using this command you can either generate:

- the ideal breakout pattern for each selected route, if no sequence exists.
or
- breakout using layers and bit pattern defined by the selected rat sequence.

Creation of breakout depends on the following two parameters:

- location of gather point. The location of gather point determines how far the breakout will go from the component. Currently, the length of the breakouts is equal to the half of the bundle width and is defined by the breakout bar.
- direction. Currently, the command supports 8 directions based on the 45 degree rotations.

The command creates breakout using 45 degree routing, leaves DRC errors, and displays any angle routes. The any angle routes are the potential routes considered by breakout router. You can manually edit these any angle routes by changing the rat sequence, or layer distribution or exit angle.

You can set the command parameters by right-clicking and choosing *Quick Utilities – Design Parameters – Route – Auto-I. Breakout* or from the *Setup – Design Parameters – Route – Auto-I. Breakout*.

Valid objects are:

- Bundles
- Bundle Ratsnests

The Auto-I. Breakout Parameters

Ripup existing	Rips up the existing breakout patterns that are partially routed and create new route patterns depending on the current selection. If a fully routed Bundle is selected, the command only re-routes it. By default, this option is set to Yes.
Allow bundle routes to intermix	Allows bundles that share common route channels to breakout together. By default, this option is set to No.

Related Topics

- [Breaking Out both sides \(ends\) of a Bundle](#)

Breaking Out one side (end) of a Bundle

To breakout one side (or end) of a bundle, follow these steps:

1. Hover your cursor over a flow segment, at the end of the bundle that you want to breakout. The layout editor highlights the segment and a datatip identifies its name.
2. Right-click and choose *Auto-I. BreakOut Closest End*.
The command starts and an execution dialog box appears showing the status of the command.
3. Click *Cancel* to stop the command.

Breaking Out both sides (ends) of a Bundle

To breakout both sides (or ends) of a bundle, follow these steps:

1. Hover your cursor over a flow segment. The layout editor highlights the segment and a datatip identifies its name.
2. Right-click and choose *Auto-I. BreakOut Both Ends*.
The command starts and an execution dialog box appears showing the status of the command.
3. Click *Cancel* to stop the command.

Related Topics


- [aibt single](#)

aibt trimtobreakout

The aibt trimtobreakout command adjusts the existing breakouts on a bundle. On a fully routed bundle, the command trims the breakouts at the end of the bundle by either extending or deleting them.

The command aibt trimtobreakout is available in the PCB Editor with:

- Design Planning option,
- In Flow Planning or in Etch Edit application modes when Groups are selected in Find Filter, and
- In pre-selection mode with specific right-click menu option.

 Selecting an object and entering the command in the Command Window will not produce the desired result.


Valid objects are:

- Bundles
- Bundle Ratsnests

Adjusting Breakouts on a Bundle

To adjust breakouts on a bundle, follow these steps:

1. Hover your cursor over a flow segment that is routed at both the ends. The layout editor highlights the segment and a datatip identifies its name.
2. Right-click and choose *Auto-I.Trim To Breakout*.
The command starts and an execution dialog box appears showing the status of the command.
When finished, the breakout sections are created on both sides at the end of the bundles.
3. Select the breakouts at the end of the bundle and move either forward or backward or at any angle.
A breakout bar is displayed and you can move the breakouts along the breakout bar in a chosen direction.
4. Right-click and choose *Auto-I.Trim To Breakout*.
The trunk has been extended or deleted along the breakout bar.
5. Click *Cancel* to stop the command.
6. When finished, right-click and choose Done from the pop-up menu.

 The off-angle routing is not supported with this command.

aidt

The `aidt` command (Auto Interactive Delay Tune) computes the required length for the clines to meet timing constraints and utilizes controlled shove/push techniques to generate tuning patterns on existing clines.

⚠ AiDT elongation is limited to segments at 45 and 90 degrees only. AiDT does not elongate to odd-angle segments.

In addition to setting parameters relevant for this command on the *Options* panel, you may also set them by right-clicking to display the pop-up menu from which you may choose:

- Design Parameters to access the Design Parameter Editor
- Options

Changing a parameter using either of these pop-up menu choices automatically updates the *Options* panel.

⚠ When you use this command in a pre-selection use model, you cannot access the *Options* panel to change the settings.

Valid objects are:

- Clines
- Cline Segments
- Bundles

⚠ In PCB Editor, this command is available with the High-Speed option only.

Related Topics

- [Using the Auto-interactive Delay Tune \(AiDT\) Command](#)
- [Auto-Interactive Delay Tuning](#)

AiDT Command: Options Panel

Access Using

- Menu path: *Route – Auto-interactive Delay Tune*

✔ The Design Parameter Editor is also available for editing the parameters listed on the *Options* panel. Choose *Setup – Design Parameters* (`prmed` command), click the *Route* tab, and choose the *Auto-I. Delay Tune Parameters* folder.

<i>Active etch subclass</i>	Indicates the etch subclass currently showing in the design.
<i>Override bundle params</i>	Determines if existing <i>Options</i> panel settings should override and bundle properties control <code>aidt</code> . If this option is selected routing is part of a bundling.

A Commands

A Commands--aidt

<i>Allow in cns areas</i>	<i>Allows use of tuning pattern inside constraint region. The default value is Yes.</i>
<i>Exclude smart criticals</i>	Is valid only if the <i>Timing Mode</i> is set to <i>Smart Timing</i> in the tvision command. Excludes the longest net in the Timing Group. With this option, you can choose the complete group of nets for automatic tuning pattern creation, and the critical net is ignored by the <code>aidt</code> command. The default value for <i>Exclude smart criticals</i> is <i>Yes</i> .
<i>Tuning Pattern</i>	Specifies style of tuning pattern. The default is <i>Accordion</i> .
<i>Accordion</i>	
<i>Gap</i>	Specifies the desired gap between the sides of the <i>Accordion</i> pattern. You can enter: Special values, such as $[N] \times \text{width}$, where $[N]$ is an integer that means N times the line width (default). Special values such as $[N] \times \text{space}$, where $[N]$ is an integer that means N times the line-to-line spacing. The default value is $3 \times \text{width}$.
<i>Min Amplitude</i>	Specifies the minimum desired height of the <i>Accordion</i> pattern. You can enter: Special values, such as $[N] \times \text{width}$, where $[N]$ is an integer that means N times the line width (default). Special values such as $[N] \times \text{space}$, where $[N]$ is an integer that means N times the line-to-line spacing. The default value is $3 \times \text{width}$.
<i>Max Amplitude</i>	Specifies the maximum desired height of the <i>Accordion</i> pattern. You can enter: Special values, such as $[N] \times \text{width}$, where $[N]$ is an integer that means N times the line width (default). Special values such as $[N] \times \text{space}$, where $[N]$ is an integer that means N times the line-to-line spacing. The default value is $40 \times \text{width}$.
<i>Corner Type</i>	Specifies the corners for <i>Accordion</i> pattern. This option is disabled. By default <code>aidt</code> generates 45 degrees corners on <i>Accordion</i> pattern.
<i>Miter Size</i>	Specifies the corner size for the <i>Accordion</i> pattern. You can enter: Special values, such as $[N] \times \text{width}$, where $[N]$ is an integer that means N times the line width (default). Special values such as $[N] \times \text{space}$, where $[N]$ is an integer that means N times the line-to-line spacing. The default value is $1 \times \text{width}$.
<i>Trombone</i>	
<i>Max Levels</i>	Specify an integer value that represents the maximum number of loops to create for each Trombone pattern. Default value is 1.
<i>Gap</i>	Specifies the desired gap between the sides of the Trombone pattern. You can enter: Special values, such as $[N] \times \text{width}$, where $[N]$ is an integer that means N times the line width (default). Special values such as $[N] \times \text{space}$, where $[N]$ is an integer that means N times the line-to-line spacing. The default value is $3 \times \text{width}$.

<i>Min Amplitude</i>	<p>Specifies the minimum height of the Trombone pattern.</p> <p>You can enter:</p> <p>Special values, such as $[N] \times \text{width}$, where $[N]$ is an integer that means N times the line width (default).</p> <p>Special values such as $[N] \times \text{space}$, where $[N]$ is an integer that means N times the line-to-line spacing.</p> <p>The default value is $3 \times \text{width}$.</p>
<i>Corner Type</i>	<p>Specifies the corners for Trombone pattern. This option is disabled. By default <code>aidt</code> generates 45 degrees corners on <i>Trombone</i> pattern.</p>
<i>Miter Size</i>	<p>Specifies the desired corner size for the Trombone pattern.</p> <p>You can enter:</p> <p>Special values, such as $[N] \times \text{width}$, where $[N]$ is an integer that means N times the line width (default).</p> <p>Special values such as $[N] \times \text{space}$, where $[N]$ is an integer that means N times the line-to-line spacing.</p> <p>The default value is $1 \times \text{width}$.</p>

Using the Auto-interactive Delay Tune (AiDT) Command

Follow these steps:

1. Choose *Route – Auto-interactive Delay Tune*.
2. Adjust the auto interactive delay tune parameters in the *Options* panel.
3. Hover your cursor over a cline or cline segment for tuning. You can select cline with a single pick, window drag, *Select by Polygon*, and *Temp Group* selection modes. The layout editor highlights the segment on which you are routing, and a datatip identifies its name. The command starts as soon as selection is completed. An execution dialog box appears showing the status of the command. When the `aidt` run is complete, the editor displays result summary in the command window.
4. Click the right mouse button and choose *Done* from the pop-up menu or *Oops* to rollback the last run.

Related Topics

- [aidt](#)

aif in

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

The `aif in` command imports design data from an AIF file. The AIF format is a simple ASCII file that describes the die and package shell. It includes die pin coordinates, die size, fingers, rings, wires, balls and netlist. It is also useful for exchanging information with package vendors and designers.

The AIF2APD module reads AIF and creates package and die symbols, builds the required padstacks, imports the netlist, places the symbols, places and labels the bond fingers and draws the rings. The APD2AIF navigates the APD database to extract the same items.

For questions and problems, contact Artwork Conversion Software at:

- support@artwork.com
- <http://www.artwork.com/package/aif/index.htm>
- Tel: 831 426-6163.

aif out

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

The `aif out` command exports design data to an AIF file. The AIF format is a simple ASCII file that describes the die and package shell. It includes die pin coordinates, die size, fingers, rings, wires, balls and netlist. It is also useful for exchanging information with package vendors and designers.

The AIF2APD module reads AIF and creates package and die symbols, optionally subtracts scribe dimensions from the die extents, builds the required padstacks, imports the netlist, places the symbols, places and labels the bond fingers and draws the rings. The APD2AIF navigates the APD database to extract the same items.

For questions and problems, contact Artwork Conversion Software at:

- support@artwork.com
- <http://www.artwork.com/package/aif/index.htm>
- Tel: 831 426-6163.

aipt

The `aipt` command (Auto Interactive Phase Tune) is used to meet the specific static and dynamic phase requirements of the differential pairs. This command operates on a user-defined selection set of clines and/or cline segments to modify the positive/negative halves of a differential pair.

The `aipt` command:

- uses information calculated by Timing Vision to determine the length of the phase imbalance within a differential pair. These phase mismatches are the amount of length that `aipt` will try to add/remove from the differential pair.
- uses a prioritized list of user-defined phase compensation techniques.
- modifies the selection set using specialized algorithms focused on the phase problem only.

 In PCB Editor, this command is available with the High-Speed option only.

Related Topics

- [Using the Auto-interactive Phase Tune \(AiPT\) Command](#)
- [Auto-Interactive Phase Tuning](#)

AIPT Command: Options Panel

Access Using

- Menu path: *Route – Auto-interactive Phase Tune*

✔ Design Parameter Editor is also available for editing the parameters listed on the *Options* panel. Choose *Setup – Design Parameters* (prmed command), click the *Route* tab, and choose the *Auto-I. Phase Tune Parameters* folder.

<i>Compensation Loc.</i>	
<i>Any</i>	<p>Lets the layout editor place the allowed compensation technique preferably at either end of the differential pair to satisfy static phase constraints. This option does not restrict pin/via pad modifications and bump location depending on the compensation techniques selected.</p> <p>When the <i>Allow Uncoupled Bumps</i> technique is enabled with Dynamic Phase constraints, this option puts phase compensation bumps anywhere along the cline paths.</p>
<i>High Pin Comp</i>	<p>Specifies that only the end of the differential pair that connects to the highest pin count component can be modified in the pin/via pad entry area. For example, the layout editor can modify the BGA end of the memory system.</p> <p>⚠ Use of <i>High Pin Comp</i> option does not affect <i>Allowed Uncoupled Bumps</i> locations for solving dynamic phase.</p>
<i>Low Pin Comp</i>	<p>Specifies that only the end of the differential pair that connects to the lowest pin count component can be modified in the pin/via pad entry area. For example, the layout editor can modify the DIMM end of the memory system.</p> <p>⚠ Use of this option does not affect <i>Allowed Uncoupled Bumps</i> locations for solving dynamic phase.</p>
<i>Compensation Techniques</i>	<p>This section controls the techniques that allow the layout editor to use at the appropriate locations set by <i>Compensation Loc.</i></p>
<i>Pad Entry Shortening</i>	<p>Enables or disables the layout editor to shorten the longer half of the pair. It focuses on the region from the gather point to the pin or via. The <i>Pad Entry Shortening</i> technique uses the <i>Allow off-angle segs</i> technique, if enabled.</p>
<i>Pad Entry Lengthening</i>	<p>Enables or disables the layout editor to lengthen the shorter half of the pair. It focuses on the region from the gather point to the pin or via as it wraps around the pad. The <i>Pad Entry Lengthening</i> technique uses only 45 degree segments in the wrap and uses the <i>Allow off-angle segs</i> technique, if enabled.</p> <p>⚠ <i>Pad Entry Lengthening</i> does not wrap more than 180 degrees around the pad.</p>
<i>Allow off-angle segs</i>	<p>Allows the layout editor to create off-angle pad entry segments. This option is used in tight pin fields, or when just slight shortening of one half of the pair is required.</p>
<i>Allow gather move</i>	<p>Allows the layout editor to modify the actual differential pair gather point.</p>
<i>Uncoupled Bumps</i>	<p>Specifies the settings to put phase compensation delay bumps into the clines to bring the pair within tolerance. You can define the value to create the bump.</p>

<i>More Options</i>	Displays the Design Parameter Editor dialog box to let you fine tune the Auto-interactive Phase Tune parameters for creating <i>Uncoupled Bumps</i> . In order to define the bumps, you have two options: Accordion and Sawtooth.
<i>Sawtooth bump parameters</i>	
<i>Min Height(H)</i>	Specifies the length of each bump.This length determines the minimum segment length between the new bump and the vertices of the originally selected segment. The default value is two-times the minimum-line-width from the design cset ($2 \times \text{width}$) .
<i>Max Height(H)</i>	Specifies the width of each bump. The default value is two-times the minimum-line-width from the design cset ($2 \times \text{width}$) .
<i>Length(L)</i>	Specifies the length each bump will added to cline. The default value is two-times the minimum-line-width from the design cset ($2 \times \text{width}$) .
<i>Accordion bump parameters</i>	
<i>Gap</i>	Specifies the desired gap between the sides of the <i>Accordion</i> pattern. The default value is $3 \times \text{width}$.
<i>Min Amplitude</i>	Specifies the minimum desired height of the <i>Accordion</i> pattern. The default value is $3 \times \text{width}$.
<i>Max Amplitude</i>	Specifies the maximum desired height of the <i>Accordion</i> pattern. The default value is $40 \times \text{width}$.
<i>Corner Type</i>	Specifies the corners for the <i>Accordion</i> pattern. By default, <code>aidt</code> generates 45 degrees corners on <i>Accordion</i> pattern.
<i>Miter Size</i>	Specifies the corner size for the <i>Accordion</i> pattern. The default value is $1 \times \text{width}$

Using the Auto-interactive Phase Tune (AiPT) Command

Perform the following steps:

1. Choose *Route – Auto-interactive Phase Tune*.
2. Adjust the parameters in the *Options* panel.
3. Hover your cursor over a cline or cline segment for tuning.
You can select cline with a single pick, window drag, *Select by Polygon*, and *Temp Group* selection modes. The layout editor highlights the segment and data tip identifies its name.
The command starts as soon as the selection is completed. A progress dialog box appears showing the status of the command.
When the `aipt` run is complete, the editor displays result summary in the command window.
4. Right-click and choose *Done* from the pop-up menu or *Ops* to rollback the last run.

Related Topics

- [aipt](#)

alias

The `alias` command lets you create shortcuts for commands you use most often. In addition to using alphanumeric characters as an alias, you can also use function keys with or without Shift and Control keys, to create a function alias for executing commands. The alias and function alias are alternative ways of entering a command, but they do not disable the full commands. You can still use the standard form of the command.

You can also enter chained commands, representing more than one consecutive action or macro command file, at the console window prompt, or define them as an alias. Use a semicolon (;) to separate the commands and enclose the commands in quotes.


Aliases and function aliases work only in the Cadence layout editor, not at the operating system level. When you create an alias or a function alias, it is active only for the current work session. When you exit the layout editor and return to the operating system, aliases and function aliases are lost. To use aliases and function aliases repeatedly, define and save them in a local environment file.

Some default aliases and function aliases are provided with the product. The Default Aliases/FuncKeys list (global environment file) includes the default aliases for the typed commands and the function keys. It also lists any aliases that you entered in the local environment file. You can access the Default Aliases/FuncKeys list by typing `alias` or `funckey` at the console window prompt. You can also choose *Tools – Utilities – Aliases/Function Keys*.

In addition to using standard commands, you have several options at the keyboard when using the `alias` command. You can:

- Use the default aliases.
- Define temporary aliases for an individual work session by typing `alias` and the arguments at the console window prompt.
- Establish aliases in a local environment file that remain in effect at every login until you change the environment file.

The `unalias` command deletes aliases and function aliases.

 The function alias feature has been expanded to include alphanumeric keys. For additional information, see [funckey](#).

Syntax

```
alias
```

```
alias <user-defined name> <command to execute>
```

```
alias <Fkey> <command to execute>
```

user-defined name	A group of characters or an abbreviation that you assign as a shortcut to execute a specified command. This name cannot contain any blank spaces. When you type the <i>user-defined name</i> at the console window prompt to execute the specified command, you need to press <code>Enter</code> .
command to execute	Specifies the command(s) to be executed when you type the <i>user-defined name</i> at the console window prompt or when you press the function key. When entering multiple commands, enclose them with quotation marks (" ") and separate them with semicolons (;).
Fkey	Specifies the function key you assign as a shortcut to execute a specified command. When you press the function key, you do not need to press <code>Enter</code> .

Creating a Command Alias

To create a command alias for the current work session:

1. At the console window prompt, type `alias`, the *user-defined name*, and the command string to which you are applying the `alias` command.

```
alias <user-defined name> <command(s)>
```

1. Type the *user-defined name* and press `Enter` to execute the specified command.

Examples

The following examples use the `alias` command:

- `alias`
When you type `alias` at the console window prompt, the layout editor displays the Defined Aliases/Funckeys list of aliases and function keys that have been defined in your environment file.
- `alias al add line`
After you define this alias, type `al` at the console window prompt and press `Enter` to run the `add line` command.
- `alias ll5 "add line; setwindow form.mini; FORM mini line_width 15.0"`
After you define this alias, type `ll5` at the console window prompt and press `Enter` to run these commands.
- `alias glp gloss param`
After you define this alias, type `glp` and press `Enter` to run the `gloss param` command.
- `alias F2 shell`
After you define this function alias, press `F2` to invoke the `shell` command. You do not have to press `Enter`.
- `alias F1 add connect`
After you define this function alias, press `F1` to invoke the `add connect` command. You do not have to press `Enter`.

alias_protect

The `alias_protect` command lets you assign an alias "read-only" status, effectively disabling the ability to unalias a command.

Syntax

```
alias_protect [-n] [-y] <alias>
```

Assigning an Alias Read-Only Status

To apply alias protection:

1. Type `alias_protect` at the layout editor's user interface console command.
2. To apply protection to an aliased command, type `-y` and the alias name.
The defined alias is now marked read-only and cannot be unaliased.

To remove alias protection:

1. Type `alias_protect` at the layout editor's user interface console command.
2. To apply protection to an aliased command, type `-n` and the alias name.
The defined alias is can now be unaliased.

Example

In this example, alias protection is applied to the alias `gp` (for `gloss parameters`). An attempt is then made to unalias the command, resulting in an error message.

Command > `alias_protect`

W- Usage: `alias_protect [-n] [-y] <alias>`

Command > `alias_protect -y gp`

Command > `unalias gp`

W- alias `gp` is marked read-only, not changed

In this example, alias protection is removed from the alias `gp`, followed by applying `unalias` to it. An attempt is then made to run the command using its previously defined alias, resulting in an error message.

Command `alias_protect -n gp`

Command > `unalias gp`

Command > `gp`

E- Command not found: `gp`

align components

The `align components` command fine-tunes the alignment of already placed components to maximize routing channels and printed-circuit board real estate, using the following criteria:

- Components must exist on the same subclass.
- More than one component must be chosen.
- Components must not have the FIXED property assigned to them.
- Components are aligned according to their place bound extents.

Available only in the Placement application mode, this command functions in a pre-selection use model, in which you choose at least two components first, then right click on the component you want to serve as the reference and execute the command.

Valid objects are Components (refers to the database element SYMBOL_INSTANCE).

Related Topics

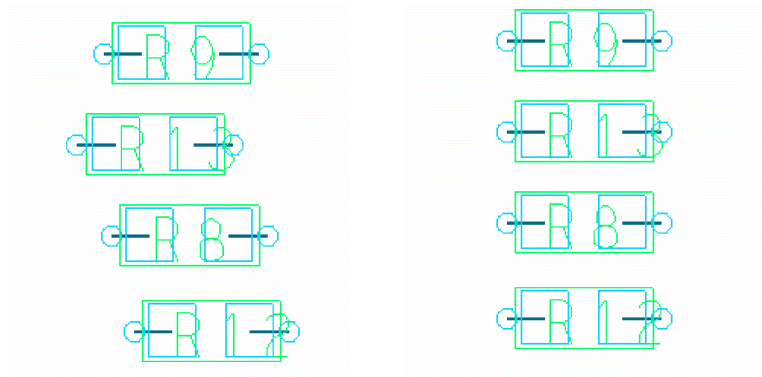
- [Aligning Components to Optimize Routing Channels](#)

Align Components Command: Options Panel

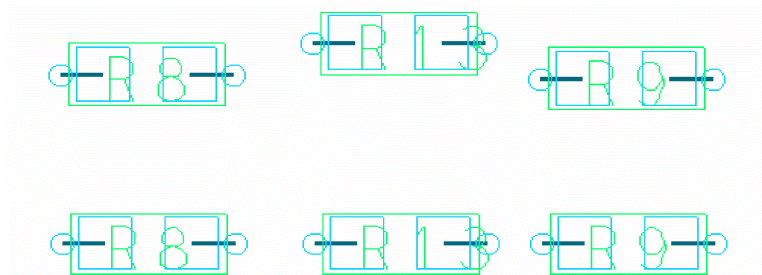
<i>Alignment Direction</i>	Specifies alignment orientation. The choices are <i>Horizontal</i> and <i>Vertical</i> .
<i>Alignment Edge</i>	Specifies the alignment edge. <ul style="list-style-type: none"> Horizontal: Choose top or bottom edge or the center of components. Vertical: Choose right or left edge or centre of components.
<i>Spacing</i>	Specifies the spacing between the components, defined in the user units. <ul style="list-style-type: none"> Off: Choose to align components without spacing between them. <i>Use DFA constraints</i>: Choose to align components with minimal spacing as defined in the <i>DFA Constraints Dialog spreadsheet</i>. <i>Equal spacing</i>: Choose to align components with initial spacing between them. To set the new spacing value, use the field adjacent to equal spacing checkbox. You can increase and decrease the spacing value using the + and - buttons. You can also set t in the field adjacent to the +/- buttons.

Chosen components are align to the reference component by body center in a row or column as shown. You specify the reference component by hovering your cursor over it, ensuring that it has been included in the selection set of chosen components. Any fanout etch associated with a component moves along with it.

Column Alignment

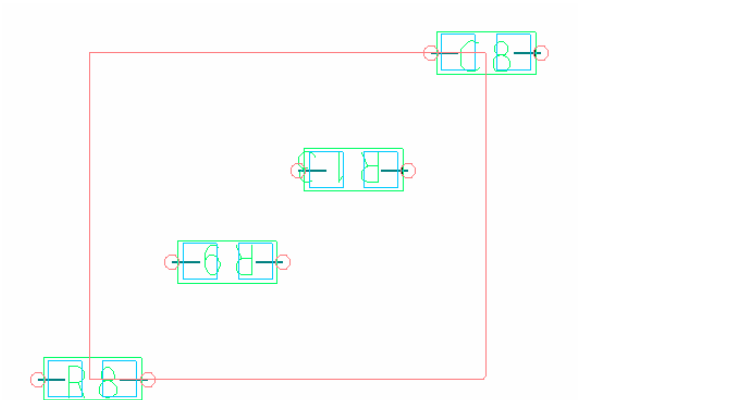


Row Alignment



When alignment of components may occur in either a row or column without violations, the align components function tries to determine the intended alignment direction.

Possible Horizontal or Vertical Alignment



When the reference component's rotation is a multiple of 90° , components align in a row or a column based on whether the component rectangles overlap. If an overlap in one direction occurs, then components align in the opposite direction.

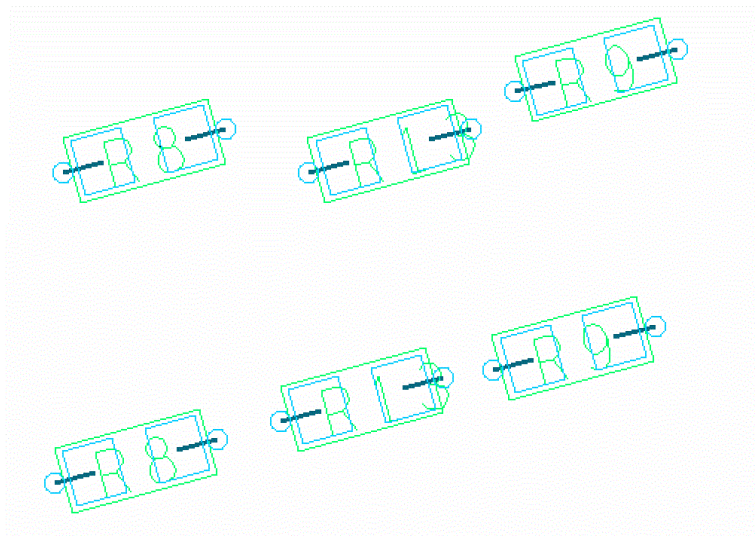
The component place boundary is used unless the *DFA package-to-package constraints* option is enabled on the *DFA Constraints Dialog spreadsheet*, available by choosing *Setup - Constraints - DFA Constraint Spreadsheet* ([dfa_spreadsheet](#) command), in which case, the component DFA place boundary is used. However, the DFA constraints will not be enforced.

If no violation occurs in either direction, then the components align as follows:

- in a row if the rectangle is longer horizontally;
- in a column, if longer vertically;
- if more approximate to a square, a prompt appears to specify row or column alignment

When the reference component's rotation is not evenly divisible by 90° , as in the figure below, and the rotation of the chosen components matches that of the reference component, (or the rotation of the reference component plus 90°), then the components align at the same angle as the reference component, passing through its origin. Otherwise, the components align in a row or a column.

Components with a 15° angle align at 15°



Aligning Components to Optimize Routing Channels

Follow these steps to align components to better optimize routing channels and utilize board space more efficiently:

1. Choose *Setup – Application Mode – Placement Edit* to access the placement application mode, or right click and choose *Application Mode – Placement Edit*.
2. Choose at least two components to align, ensuring that they are on the same subclass.
3. Hover your cursor over the component you want to serve as the reference component, ensuring that it has been included the selection set.
4. Right click and choose *Align components* from the popup menu.
The components are aligned using settings in the *Options* panel.

Related Topics

- [align components](#)

align groups

The `align group` command fine-tunes the alignment of already placed groups to maximize routing channels and printed-circuit board real estate.

Available only in the Placement application mode, this command functions in a pre-selection use model, in which you choose at least two groups first, then right click on the group you want to serve as the reference and execute the command.

Valid object is:

- Groups

Aligning Groups to Optimize Routing Channels

Follow these steps to align groups to better optimize routing channels and utilize board space more efficiently:

1. Choose *Setup – Application Mode – Placement Edit* to access the placement application mode, or right click and choose *Application Mode – Placement Edit*.
2. Choose at least two groups to align.
3. Hover your cursor over the group you want to serve as the reference group, ensuring that it has been included the selection set.
4. Right click and choose *Align groups* from the popup menu.
The groups are aligned using settings in the *Options* panel.

align modules

The `align modules` command fine-tunes the alignment of module instances.

For example, module instances created with the [place replicate](#) suite of commands. This functionality is similar to that of the [align components](#) command.

Available only in the placement edit application mode, this command functions in a pre-selection use model, in which you choose at least two module instances first, then right click on the module you want to serve as the reference and execute the command.

Valid object is:

- Module instance

Aligning Modules to Optimize Routing Channels

Follow these steps to align modules to better optimize routing channels and utilize board space more efficiently:

1. Choose *Setup – Application Mode – Placement Edit* to access the placement application mode, or right click and choose *Application Mode – Placement Edit*.
2. Right-click and set the *Super Filter* to *Module*.
3. Choose at least two module instances to align.
4. Hover your cursor over the place replicate module you want to serve as the reference module, ensuring that it has been included the selection set.
5. Right click and choose *Align modules* from the popup menu.
The module instances are aligned using settings in the *Options* panel.

allegro

The `allegro` batch command starts Allegro X PCB Editor or Allegro SI.

If you do not include a design name and you have previously run this version of the layout editor, the last saved design in the previous session opens, based on information written to the `master.tag` file. If you do not want the layout editor to open the last design, move or delete `master.tag`. A new, unnamed board appears.

To find `master.tag`, open the `.ini` file, located in your `pcbenv` directory, and search for `directory`.


Syntax

```
allegro <args> [<allegro database>][-s <script>][-S <script>][-p <startdir>] [-j|-o<journal>][proj<cpm file>][-product<product name>][-option <option name>][-expert|-designer|-pcb] [-sq] [-expert|legacy] [-orcad] [-nographic|-nograph] [-readonly] [-safe] [-noopengl] [-mps<XXX>] [<design name>] [-help] [-version] [-versionLong]
```

-s<script>	Executes the specified script. The default extension is <code>.scr</code> . If you do not run the command in the directory where the script is located, you must include the path in the script's file name. Multiple <code>-s</code> options may be specified, and they replay sequentially on the command line. Example: <code>-s script1 -s script2 ... -s scriptN</code> Up to 63 scripts are supported.
-S<script>	Executes the specified script file and sets the startup directory to the last directory stored in the <code>allegro.ini</code> file. Multiple <code>-S</code> options are permitted (identical to those for <code>-s</code>).
-p<start directory>	Specifies the startup directory. If you run this command with a design file name that includes a path—for example, <code>/home/dkm/pcb/boards/layout123</code> —other files created during processing, such as log files, are created in the directory you specified and not in the directory where the design is located.
-j -o<journal file>	Opens a journal file that records your work session. The <code>.jrl</code> extension is appended to the specified file name. Default is <code><prog>.jrl</code> .
-proj<cpm file>	Reads the HDL-indicated <code>.cpm</code> file on startup. The initial starting directory and design name should be specified in the <code>.cpm</code> file.
-product<product name>	Starts the specified product tier of the layout editor for which you are licensed. If you do not specify a tier, the Cadence Product Choices dialog box appears, from which you choose one. Use <code>-product help</code> for a list of available products.
-option <option name>	Specifies the options to be run. Used with the product option to specify the product and option required. The option may be specified multiple times. Use <code>-product help</code> for a list of available products and options.
-expert -designer -pcb -gxl	Specifies a legacy program tier of Allegro to be run. This overrides any default set in a <code>.ini</code> file.
-sq	Starts Allegro with filtered set of SI licenses.
-geo	Specify location and size of window.
-orcad	Specifies the program tier of OrCAD. This overrides any default set in a <code>.ini</code> file. If an Allegro OrCAD license is not found, you can still use the layout editor in Demo mode.
-readonly	Disables saving the design. The title-bar displays Read Only with the design name.
-nographic -nograph	Launches application in non-graphic mode. Usually used when running scripts without launching the application. For more information, see the description of -nographic .
-safe	Launches application without user or site configuration files and settings. For more information, see <code>\$CDSROOT/share/pcb/batchhelp/safe.txt</code> .
-noopengl	Disables OpenGL.

A Commands

A Commands--allegro

-mps<XXX>	Standard Cadence MPS argument support (This is not typically required.)
<allegro database>	Start editing with this database (ignore <code>.ini</code> file); default extensions are <code>.brd</code> , <code>.mcm</code> , <code>.dra</code> , <code>.mdd</code> .
<design name>	<p>Opens the specified design file. The default extension is <code>.brd</code>.</p> <p>If you do not run the command in the directory where the script is located, you must include the path in the script's file name.</p> <p>If you do not include a design name or if the layout editor cannot find the specified file, the layout editor opens a default file called <code>unnamed.brd</code>.</p>
-help	<p>Prints information about this command.</p> <div> To view the information Windows, redirect the standard output in text file. For example,</div> <pre>allegro -help > myfile.txt</pre>
-version	Prints the application's version and exits.
-versionLong	Prints the application's long version, if available, and exits.

Examples

This first example starts the Allegro PCB Designer. It opens the `c123board` design file and runs the `setcolors` script against it.

```
allegro -product Allegro_performance -s setcolors c123board
```

The following example starts Allegro SI and opens the `021231demo` design file.

```
allegro -sq -product SPECCTRAQuest_SI_expert 021231demo
```

The following example starts OrCAD PCB Editor


```
allegro -orcad
```

allegro_component

The `allegro_component` command lets you generate the necessary files for placing instances of an IC package into the PCB Editor. You can also generate the files to fabricate and manufacture your new package design. The layout editor includes several manufacturing outputs and interfaces to accomplish these tasks.

The layout editor initially generates a directory, `./component` in your current working directory, for the export files. If this directory already exists, you are asked if you want to overwrite it. The data translation takes place without any further input from you. The `./component` directory contains the following files:

- A `chips_prt` (for SCALD) or `chips.prt` (for HDL) file for Design Entry HDL or System Connectivity Manager that describes the pins and their functionality.

 By default, pin names are of the form `<net name>_<pin name>` such as `VDD_B1`, where `VDD` is the net name and `B1` is the pin number. You can also have pin names of the form `<net name>_<index>` by setting the `lc_Packaging--allegro_component_seq_pin_names` in User Preferences. The index is padded with 0s to have uniform name length. The order of the suffix counters is sorted by the pin numbers. For example, for a net `VDD` with 10 pins, the names will be `VDD_01`, `VDD_02`, `VDD_03`, and so on. Similarly, for a net `VSS` with 100 pins, the names will be `VSS_001`, `VSS_002`, `VSS_003`, and so on.

- A `.txt` file that contains device information, needed for third-party tools.
- A `package_pin_delay.rpt` file (in units of time—picoseconds) or a `package_pin_delay_length.rpt` (units of length—microns) containing pin delay information for each signal package pin in the selected symbol.
The report lists each BGA ball pin, its assigned net, and the signal delay from that ball to the farthest die pin on the same net.
Be sure that the package pins are completely routed to ensure that output results are accurate. Additionally, you should correctly identify the voltage properties of all power and ground nets; that is, with a 0.0v value for ground nets and a non-zero value for power nets.
- A `.pad` file that contains the padstack information. In addition to the `.pad` file, the layout editor creates a `.ssm` if the padstack contains shape symbols.
- A `.dra` file that contains the symbol design. To create a `.dra` file from the `.mcm`, use the `dump_library` utility.
- A `.psm` file that contains the locations of the pins, the geometry from PART GEOMETRY/ASSEMBLY TOP, and any reference designators.
- A `.tsg` file necessary to create a Design Entry HDL or System Connectivity Manager body file, if necessary.
- A `symbol.css` Concept symbol file
- A `dump_libraries.log` file

Related Topics

- [Exporting from a .mcm Design](#)
- [Importing Part Information into Design Entry HDI or System Connectivity Manager](#)

Component Options Dialog Box


Access Using

- Menu path: *File – Export – Board Level Component*

BGA	Select the BGA to be exported from the list.
Select output format	Highlight the format type that supports the component you are exporting. Choices are <i>HDL</i> or <i>SCALD</i> .
Library name	Specifies the directory where the output files are written.
Export flattened hierarchy	Check to export a flat hierarchy where all files are written directly in the directory specified in <i>Library name</i> . If it is not checked, which is the default, sub-directories are created for different files. For example, the <code>chips.prt</code> file is written in the <code>chips</code> sub directory and so on.
Delay Report Options	
Time delay report	If checked, specifies that the layout editor generates the time delay report and saves it to disk.
Length delay report	If checked, specifies that the layout editor generates the length delay report and saves to disk.
OK	When you click <i>OK</i> , the layout editor displays the Padstack for Component Dialog Box .
Cancel	Exits the command without generating any files or report.
Help	Displays the Help Window for this command.

Padstack for Component Dialog Box

The Padstack for Component dialog box, which is a browser that appears when you click *OK* in the *Component Options* dialog box, lets you find and choose an object easily. All listed objects—in this case, padstacks—are listed in alphabetical order.

Search field	<p>Type the name of the padstack in the search field or highlight it in the list box.</p> <p>To narrow the list, enter a search string in the search field. For example, a search string of <code>MTG*</code> returns all objects beginning with <code>MTG</code>. The asterisk (*) displays the complete list of padstacks.</p> <p>The layout editor remembers your last search.</p>
List box	Specifies the list of padstacks in the database or in the library.
Database	Specifies the default setting of padstacks in the database.
Library	<p>Check this box to display the list of padstacks in the library.</p> <div style="border: 1px solid #fde725; padding: 5px; margin: 5px 0;"> <p> The objects listed in Library mode may sometimes include items already in the design. This is because database items remain displayed in the list box when the library option is checked.</p> </div> <p>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 dialog box; that is, the layout editor selects the database object.</p>
OK	Generates the files.
Cancel	Dismisses the Padstack for Component dialog box and the <i>Component Options</i> dialog box without generating any files.
Help	Displays the Help window for the Padstack for Component dialog box.


Related Topics

- [Importing Part Information into Design Entry HDI or System Connectivity Manager](#)

Exporting from a .mcm Design

Use the following procedure to transfer a .mcm design package footprint into a format that can be used in a PCB (.brd) and schematic design:

1. Load the design (.mcm file) from which you want to export data.
2. Choose *File – Export– Board Level Component* from the menu or run the `allegro_component` command to display the *Component Options* dialog box.
3. Select a BGA from the list.
4. Choose an output format (*HDL* or *SCALD*).
5. To generate reports, check the appropriate boxes.
6. Click OK to display the Padstack for Component dialog box. If there are multiple symbols, you are prompted to pick the IO symbol to export.
7. Choose a padstack to use in generating the symbol for the PCB.

 The padstack should be the one to use in the target PCB design and not the one used by the footprint in the source database. The padstack in the target and the footprint are defined on different layers.

8. Click OK on the Padstack for Component dialog box.
The layout editor initially generates a directory, `./component`, for the export files. If this directory already exists, you are asked if you want to overwrite it.

Related Topics

- [allegro_component](#)

Importing Part Information into Design Entry HDI or System Connectivity Manager

Use the following steps:

1. Create a new project with the Project Wizard. The following are possible entries.

Project name: mcmproject

Location: \\cds_work\projdir

Project Libraries: Accept defaults or add required libraries

Library: Accept default (mcmproject_lib)

Design Name: top

The following directory structure is thereby created:

\\CDS_WORK\PROJDIR\

cds.lib

mcmproject.cpm

\\CDS_WORK\PROJDIR\temp\

cfg_package.log

cfg_pic.log

cfg_verilog.log

cfg_vhdl.log

\\CDS_WORK\PROJDIR\worklib\

\\CDS_WORK\PROJDIR\worklib\top\

\\CDS_WORK\PROJDIR\worklib\top\cfg_package\expand.cfg

\\CDS_WORK\PROJDIR\worklib\top\cfg_pic\expand.cfg

\\CDS_WORK\PROJDIR\worklib\top\cfg_verilog\expand.cfg

\\CDS_WORK\PROJDIR\worklib\top\cfg_vhdl\expand.cfg

1. Create a directory in the *worklib* directory with the same name as that of the I/O symbol you exported (for example, *worklib\newpart*).
2. Create a directory, *chips*, in the new *worklib\newpart* directory.
3. Copy the *chips.prt* file from the *mcm* directory into the new *worklib\newpart\chips* directory.
4. Create a directory, *sym_1*, in the *worklib\newpart* directory. This is the symbol view that holds the soon-to-be-created symbol file.
5. Copy the *.tsg* file from the *mcm* directory into the new *worklib\newpart\sym_1* directory (for example, *<tsgfile>.tsg*).
6. From an operating system prompt, change directories into the *worklib\newpart\sym_1* directory.

7. From an operating system prompt, convert the *<tsgfile>.tsg* file into a symbol file with the following command:

```
<path_to_tools>/tools/fet/concept/bin/bodygen -p <tsgfile>.tsg -b symbol.css
```

1. Run Design Entry HDL or System Connectivity Manager using the project file created previously. The design for this example is called `top`.
2. Place the newly created part `newpart` using the Design Entry HDL or System Connectivity Manager command Component – Add .
3. When finished, save the schematic and exit Design Entry HDL or System Connectivity Manager.

Example Length Delay Report

The following item types will have their length calculated and added into the total length associated with a pin: for the Package Length Delay Report.

- Pins
- Vias
- Clines
- Bond Wires (2D length only)

These objects are not calculated:

- Bond Fingers
- Shapes
- Fillets
- Ratsnest Lines

PIN DELAY

REFDES BGA

DEVICE UNNAMED_BGA

UNITS microns

A1	1414.213562	VDD
A10	15523.069481	NET602
A11	0.000000	VSS
A12	14830.229663	NET592
A13	15764.369319	NET590
A14	0.000000	VSS
A15	14417.091139	NET580
A16	13795.198393	NET570
A17	0.000000	VSS
A18	13466.707194	NET560
A19	14386.601592	NET558

A Commands

A Commands--allegro_component

A2	0.000000	VSS
A20	0.000000	VSS
A21	14386.601592	NET549
A22	13466.707194	NET547
A23	0.000000	VSS
A24	13795.198393	NET537
A25	14417.091139	NET527

Related Topics

- [allegro_component](#)
- [Component Options Dialog Box](#)

allegro_cshrc

The `allegro_cshrc` is a file that sets up the layout editor's environment running under UNIX. It initializes all environment files and paths and lets you run the layout editor's programs. It is designed to be included in your personal `.cshrc` file, where it then runs automatically when you log in.

Use the directory structure implemented at your site.

Syntax

```
allegro_cshrc
```

Example

Enter the following line in your `.cshrc` file:

```
source /usr/cds/tools/pcb/bin/allegro_cshrc
```

allegro_downrev_library

The `allegro_downrev_library` command is a batch program. It re-evaluates the library parts and removes or converts all new functionality added in current releases that is not supported in previous release.

This command converts library parts from 17.x to 16.6 release.

The supported database types are `.psm`, `.bsm`, `.osm`, `.fsm`, `.ssm`, and `.pad`.

 The command does not downrev board (`.brd`) and drawing (`.dra`) files.

Syntax

```
allegro_downrev_library <input design(s)> [-outfile <output design>]
```

input design(s)	Specify the input design name. Use wildcard to process multiple designs at a time.
output design	Specify the output design name. If not defined, the input design is saved with <code>.orig</code> extension. Using wildcard downrev will overwrite design file names.

Changes in Padstack Data after Downrev

Migrating padstack back to the previous release(16.6), deletes the following data from the `.pad` files:

- Adjacent keepout definitions
- Same layer keepout definitions
- Anti-route keepout (ARK) definitions
- Backdrill information
- Counter bore/sink settings
- Coverlay pad definitions
- New padstack usage types
- Any properties associated with padstack

The downrev process fails if the following data exists in the pad definition:

- New pad geometry
 - Rounded rectangle
 - Chamfered rectangle
 - Donut
 - n-Sided polygon
- Flash symbol in place of thermal pads
- More than 16 user-defined mask layers
- Square drill hole

Changes in Symbols Files after Downrev

When downrev symbol files (`.psm`, `.bsm`, `.ssm`, `.fsm`, and `.osm`) the following information converts as follows:

- Objects defined on `DESIGN_OUTLINE` or `CUTOUT` class are deleted and recreated on the `BOARD_GEOMETRY/OUTLINE` class.
- Objects on `RIGID_FLEX` or `SURFACE_FINISHES` classes are entirely deleted.

Downrev fails if the symbol file:

- Contains pins which are specified for Chip on Board connection.

allegro_plot

The `allegro_plot` command provides one of the methods you can use to create plots on UNIX workstations. You can run `allegro_plot` as a batch command or by way of graphical plotting interfaces that you run from your UNIX command prompt. `allegro_plot` uses the Cadence corporate plotting package `plotServ` and provides a common group of plotting drivers for a wide range of available output devices.

You must create or have available for use intermediate plot (IPF) files and control files from a design, Gerber file, or Excellon drill file.

When you run the `allegro_plot` command on a UNIX workstation, the `.cdsplotinit` plotter configuration file, which lists available printers/plotters, must reside in `<install_path>/tools/plot`, the current working directory, or your home directory.

For more information, see *Preparing Manufacturing Data* in the user guide. For additional details on plotting, including creating and using Intermediate Plot Files (IPF), control, and stipple files, see *Preparing Manufacturing Data* in the user guide.

Syntax

```
allegro_plot [-p] [-s] [-c] [-b] [-o] [-m] <penplot_filename>
```

- p	Specifies parameter file name to be loaded upon startup. The default parameter file name is <code>allegro_plot_param.txt</code> .
- s	Specifies the script file name to be played upon start-up. Note that the <code>-s</code> and <code>-b</code> options may not be used at the same time.
- c	Specifies number of copies to be plotted.
- b	Specifies batch mode. When this switch is present, no graphical user interface appears. The specified IPF file is immediately processed according to the parameters specified in the parameter file indicated with the <code>-p</code> option. If no parameter file is specified, the program reads the default parameter file <code>allegro_plot_param.txt</code> . If the default parameter file does not exist, then the program uses default values. Note that the <code>-s</code> and <code>-b</code> options may not be used at the same time.
- o	Specifies an output file name. If this option is specified, plot output is directed to the given file name rather than to the plotter specified in the parameter file. Using this option is equivalent to pushing the "output to file only" button on the main <code>allegro_plot</code> dialog box.
- m	Specifies that mail is to be sent to you when the plot is complete. Using this option is equivalent to pushing the Send Mail to User button on the <code>allegro_plot</code> dialog box.

Related Topics

- [Running the Allegro Plot Command in Graphical Mode](#)

Allegro Plot Dialog Boxes

Unless the `-b` option is used, `allegro_plot` defaults to a graphical user interface (GUI). The GUI consists of the following three dialog boxes:

- Allegro Plot Dialog Box
- Plot Options Dialog Box
- Queue Status Dialog Box

Controls in these dialog boxes are analogous to the arguments used when you run the command in batch mode.

Allegro Plot Dialog Box

The *Allegro Plot* dialog box is the base from which all plotting operations are conducted. The three sections comprising the dialog box are Commands, Setup, and Form Control.

Commands Section

The *Commands* section of the `allegro_plot` dialog box is used to run the three major operations from within the `allegro_plot` program:

- Setting up the plotting options
- Checking the status of the available plotting devices
- Creating an actual plot or plot output file.

The *Commands* section consists of three buttons:

Plot options displays the *Allegro Plot Options* dialog box used for setting the plotting parameters for the current plot.

Plot causes the file specified in the IPF File Name field in the Setup section of the `allegro_plot` dialog box to be sent to either the current plotter or to an output file that you specify. The IPF is sent with the parameters that are currently set in the `allegro_plot Options` dialog box.

Queue status displays the *Queue Status* dialog box which shows the current status of the available plotting devices.

Setup Section

The *Setup* section of the `allegro_plot` dialog box is used to specify how you would like the IPF files to be plotted. The Setup section consists of various fields described below.

<i>IPF file name</i>	Specifies the IPF file that is to be plotted. The file is read immediately after the name is entered in this field. If the file is changed at any time after it has been entered and before it has been plotted, the name needs to be re-entered here.
<i>Current plotter</i>	An information-only field that indicates the plotter that is currently active. If you choose the Plot button in the Commands section, this is the plotter to which the IPF is sent, unless the Send Only To File button is selected. You choose the current plotter from the Plotter Setup section of the <code>allegro_plot Options</code> dialog box.
<i>Parameter file</i>	Specifies the name of the current parameter file. The parameter file contains information for each of the user accessible fields in the <i>Options</i> dialog box. The default parameter file is <code>allegro_plot_param.txt</code> .
<i>Load</i>	Selected to read the parameter file currently specified in the Parameter File field and update the <i>Options</i> dialog box according to the information found in the parameter file. If the parameter file specified does not exist, then an error message displays and you are prompted for a valid file name.
<i>Save</i>	Selected to create the parameter file specified in the Parameter File field. If the specified parameter file already exists, then you are prompted to indicate whether or not the existing file should be overwritten. If you respond yes , then the old file is overwritten by the new information. If you respond no , then no save is performed.
<i>Number of copies</i>	Specifies the number of copies you want to plot. The range is 1 to 99. Note that this value has no effect if the Send Only To File button is checked.
<i>Send only to file</i>	When checked, sends the plotter specific output to the file name specified, rather than to the plotter specified in the Current Plotter field.
<i>Mail log to</i>	When checked, sends a message notifying you when the plot is complete.
<i>Plot header</i>	When checked, plots a header containing plot information along with the plot on a separate page. Any text entered in the three lines of up to 80 characters each, is included in the header page.

Form Control Section

The *Form Control* section consists of four buttons described below.

<i>OK</i>	Closes the allegro_plot dialog box and any other forms that were invoked from the allegro_plot dialog box. Selecting OK also stops execution of the allegro_plot program. If the parameters have been changed since the last parameter file read or save, then you are prompted whether or not to save the current parameters. If you choose Yes to save the current parameters, then the current parameters are written to the current parameter file name. If you choose No, then the program simply exits.
<i>Reset</i>	Sets all of the fields on the allegro_plot dialog box to the values that existed when the dialog box first displayed. Note that this only includes those values that can be changed on the allegro_plot dialog box. Values such as the Current Plotter field are changed from the allegro_plot <i>Options</i> dialog box and are therefore not affected by the Reset button.
<i>Help</i>	Displays information on how to use the dialog box.
<i>Script</i>	Used to record and replay scripts.

Plot Options Dialog Box

The *Options* dialog box, which consists of five basic sections, is used to specify the output parameters for plotting the current job. The dialog box itself is displayed below. The five sections comprising the dialog box are Plotter Setup, Orientation Setup, Line/Arc Options, Filled Object Options, and Form Control buttons.

Plotter Setup Section

The *Plotter Setup* section of the *Options* dialog box is used to choose and display information to be used in configuring the current plotter. The Plotter Setup section consists of the fields described below.

<i>Plotter name</i>	Specifies the plotter to which the IPF should be directed. This is a pull-down list with which you must choose one of the available plotters specified in the .cdsplotinit file.
<i>Paper size</i>	Specifies the paper size to be used for the plotter specified in the Plotter Name field. Note that this is a pull-down list of available paper sizes for the plotter as specified in the .cdsplotinit file. Note that if the current plotter name is changed and the current paper size is not available on the new plotter, then the paper size automatically defaults to the first paper size specified for the current plotter in the .cdsplotinit file.
<i>Stipples file</i>	Specifies the name of the stipples file to be used for the current plot. The stipples file describes both the colors and fill patterns to be used for the various pens needed for the plot. The format of the stipples file is described later in this chapter. Note that the stipples file is utilized for all output devices supported by allegro_plot.
<i>Sizing units</i>	Specifies the units in which the plot data is displayed in the <i>Options</i> dialog box. You can specify inches, centimeters, or millimeters. Note that this field has no effect on the actual plot data. This only affects how the information is displayed in the <i>Options</i> dialog box. Specifically, the Page Extents, IPF Extents, Offset, and Plot Size fields are displayed in the units specified by the Sizing Units field.
<i>Page extents</i>	An information-only field that displays the extents of the current paper size for the current plotter in the units specified in the Sizing Units field.
<i>IPF extents</i>	An information-only field that displays the extents of the information contained in the current file. This value is expressed in the units specified in the Sizing Units field. Note that if no IPF file has been specified, or if the specified file has an invalid format, the IPF Extents reads zero.

Orientation Setup Section

The *Orientation Setup* section is used to specify those plotting options that affect the overall page layout of the plot.

A Commands

A Commands--allegro_plot

Centering	<p>Specifies the type of centering to be used. There are three types of centering available as described below.</p> <ul style="list-style-type: none">• None specifies that no automatic centering of the plot is performed.• Auto Center specifies that the x and y offsets are automatically adjusted to center the given plot on the page or pages according to the current scale factor. Note that if the scale factor is changed or the Rotate button is checked while auto center is selected, the x and y offsets automatically update to keep the plot centered under the new conditions. Note also that if you change the x or y offset values manually, then the Centering field is automatically set to None .• Fit To Page automatically adjusts the scale to make the plot as large as possible within the extents of a single page. Also, the x and y offsets are set so that the plot is centered on the page. Note that if the Rotate button is checked while Fit To Page is selected, then the scale and the x and y offsets are automatically updated to give the plot a best fit under the new conditions. If the x or y offsets are changed manually, then the Centering field is automatically set to None . If the scale is changed manually, the Centering field is automatically set to None and the x and y offsets is automatically set to zero.
Rotate	Rotates the plot ninety degrees. The long axis of the page is considered the x-axis and is referred to as landscape mode. The default rotation is zero degrees. When the Rotate button is checked, the short axis of the page becomes the x-axis referred to as portrait mode.
Mirror	When checked, mirrors the plot about the y-axis.
Scale factor	Sets the scale of the plot. Values for scaling may range from .0001 to 999.9999.
Offset	Sets the x and y offset of the current plot. These values are displayed and must be entered in the units specified by the value of the Sizing Units field in the Plotter Setup section.
Plot size	An information-only field that displays the total size of the resulting plot given the current conditions. Specifically, this field displays the IPF extents multiplied by the scale factor in the units specified in the Sizing Units field in the Plotter Setup section.
Total pages	An information-only field that displays the total number of pages needed to plot the IPF file under current conditions. If the current conditions require a greater number of pages than the maximum specified for the current plotter in the .cdsplotinit file, then an error message is displayed and the maximum number of pages for the current plotter is displayed.
OK to plot multipage plots	<p>Indicates what allegro_plot does with plots that require more than one page. See the Total Pages field for the number of pages required for your plot.</p> <ul style="list-style-type: none">• When checked, allegro_plot plots files that require more than one page.• When deselected, allegro_plot does not plot files that require more than one page.

Line/Arc Options Section

The Line/Arc Options section is used to specify parameters dealing with the manner in which the lines and arcs of an IPF file are plotted.

End caps	Specifies the type of end caps to be used when plotting lines and arcs whose width is greater than zero. Octagon or square end caps may be selected.
Arc approximation	<p>Specifies how arcs should be vectorized, either fine or coarse.</p> <ul style="list-style-type: none">• Fine : Plots arcs that appear very smooth.• Coarse : Plots arcs with fewer line segments resulting in a choppy appearance, but smaller output files or faster plot times.
Width	<p>Specifies how lines or arcs with width are to be plotted. You may choose True Width, Center Line, or Use Threshold.</p> <ul style="list-style-type: none">• True Width: Causes all lines and arcs to be plotted at their actual width.• Center Line: Causes all lines and arcs to be plotted as lines with zero width.• Use Threshold: Causes all lines and arcs whose width is below the value specified in the Threshold Width field to be plotted as lines with zero width and all lines whose width is equal to or greater than the value specified in the Threshold Width field to be plotted at their actual width.
Threshold width	Specifies at what minimum width a line or arc is plotted if the Width field is set to Use Threshold. This value is always specified in mils.
Default line weight	Specifies weightings of lines of zero width.

Filled Object Options Section

The *Filled Object Options* section is used to specify how certain objects are filled when they are plotted. In each of the fields in this section, you can specify one of three fill types for the given objects:

<i>Hollow</i>	Indicates that the objects are drawn as an outline only, with no fill.
<i>XHatch</i>	Indicates that the objects are filled with a cross-hatched pattern. The actual cross-hatched pattern used is plotter dependent.
<i>Solid</i>	Indicates that the objects are filled according to the fill pattern specified in the stipples file specified in the Stipples File field of the Plotter Setup section. If no stipples file is specified, then the fill pattern defaults to solid.

 Some plotters do not support a fill option. In these cases, any filled objects are plotted hollow regardless of the fill type specified.

Click the arrow button in each of the Filled Object Options fields to display the filled object options.

<i>Figures & pads</i>	Specifies the fill type to be used for all drill figures and pads in the IPF file.
<i>Lines & text</i>	Specifies the fill type to be used for all lines, arcs, and text elements that have a width greater than zero in the IPF file. If text is vectorized, any text in the IPF file are made up of lines.
<i>Filled rectangles</i>	Specifies the fill type to be used for all filled rectangles, or frectangles, found in the IPF file. If a frectangle is rotated, then it becomes a shape and is no longer considered a filled rectangle.

Form Control Section

The *Form Control* section consists of four buttons described below.

<i>OK</i>	Closes the allegro_plot Options dialog box. If the parameters have been changed since the last parameter file read or save, then you are prompted whether to save the current parameters. If you choose yes , then the parameters are written to the current parameter file name. If you choose no , then the program simply exits.
<i>Cancel</i>	Resets all of the fields in the allegro_plot Options dialog box to the values that existed when the dialog box first displayed, then closes the dialog box. Only those values that can be specified on the Options dialog box are reset. Values such as the Plot Size field are changed by entering an IPF file from the allegro_plot dialog box and are therefore not affected by the Cancel button.
<i>Reset</i>	Sets all of the fields on the allegro_plot Options dialog box to the values that existed when the dialog box first displayed. Only those values that can be specified on the allegro_plot Options dialog box are reset. Values such as the Plot Size field are changed by entering an IPF file from the allegro_plot dialog box and are therefore not affected by the Reset button.
<i>Help</i>	Displays information on how to use the dialog box.

Queue Status Dialog Box

The *Queue Status* dialog box is used to view the contents of the available plotter queues. The three sections comprising the dialog box are the *Status Info* section, the *Queue Viewer* section, and the *Form Control* section.

Status Info Section

The *Status Info* section consists of the following fields and buttons:

<i>Query plotter</i>	Is a pull-down field from which you can choose an available plotting queue to view. When the Queue Status dialog box is brought up by pushing the Queue Status button in the Commands section of the allegro_plot dialog box, the plotter queue is set to the current active plotter as specified in the Current Plotter field in the Plotter Setup field of the allegro_plot Options dialog box. The Queue Plotter field does not change the current plotter. It only changes the plotter queue that is being viewed.
<i>Delete</i>	Removes the corresponding job from the plotting queue. Type the job number to be removed from the queue in the field next to the Delete button. You can only delete jobs for which you have the appropriate permissions.
<i>Every [] seconds</i>	Allows you to specify the interval in seconds between queue status updates. An UPDATING... message flashes each time the dialog box is updated.

Queue Viewer Section

The *Queue Viewer* section displays a screen dump of the UNIX queue status command, set in the query line of the `.cdsplotinit` file. Output is platform and command dependent.

Form Control Section

The *Form Control* section consists of two buttons:

<i>OK</i>	Closes the Queue Status dialog box.
<i>Help</i>	Displays information on how to use the dialog box.

Running the Allegro Plot Command in Graphical Mode


To create plots on a UNIX workstation, perform the steps:

1. At the UNIX system prompt, type:

```
allegro_plot
```

The Allegro Plot dialog box opens.

1. Click *Plot Options*.
The *Allegro Plot Options* dialog box opens.
2. Set the plotter parameters.
3. Click OK to accept the settings and close the dialog box.
4. Set the plotting parameters in the Setup section of the Allegro Plot dialog box.

 Note: If the IPF file changes before you plot, the file name must be reentered in the IPF File Name box to ensure the changes are read.

5. Click Plot to plot the IPF file.
6. Click Queue Status to monitor the status of any files that are being plotted.
The Queue Status dialog box appears.

Examples

```
allegro_plot -b -p params1 plot1
```

Plots the file plot1 using the parameters file params1 .

```
allegro_plot -b -o outupt1 -p params1 plot1
```

Sends the plot data to a file, output1 , instead of to the plotter specified in the parameter file.

Related Topics

- [allegro_plot](#)

allegro_uprev

The `allegro_uprev` batch command takes a design database from its current version to the latest version of the layout editor. You can use this command to uprev one or multiple design databases in a batch environment.

For upreving multiple databases you can provide both databases and directories to the command. If a directory is encountered, uprev will recursively enter that directory and sub-directories until it encounters the nest directory depth limit or a read-only directory.

The command only processes database extensions and directories supported by the application. All other files are ignored.

If no database is provided as an input the command will process all files present in current working directory and all sub-directories.

Syntax

`allegro_uprev`

Layout, drawing, or symbol file name (*.brd):

Output layout, drawing, or symbol file name (*.brd):

input_file	The name of the database you want to uprev. The default is <code>.brd</code> .
output_file	The name of the database after the uprev. Giving an output name that is different than the input name prevents the input database from being destroyed.
-n <nest directory depth>	Specify maximum depth to descend into a directory tree.
-b	Run command for multiple databases. This option is equivalent to <code>allegro_uprev_overwrite</code> command.
-drc	Updates all DRCs in batch mode.
-d	Runs command in debug mode.
-version	Prints the version.

Examples

```
allegro_uprev -d foo.brd out.brd
```

Uprev and perform batch DRC on `foo.brd` and write result into `out.brd`.

```
allegro_uprev foo.brd
```

Uprev `foo.brd` and overwrite it with updated database.

```
allegro_uprev -b
```

Uprev all databases in a current directory and any sub-directories up to a depth of 3.

```
allegro_uprev -b *
```

Uprev all databases in current directory and any child directories up to a depth of 3.

```
allegro_uprev -b *.pad
```

Uprev only padstacks found in current directory.

```
allegro_uprev -b -d -n 1 symbols padstacks
```

Uprev and perform batch DRC on all databases found in symbols and padstacks directories. Do not descend to any sub-directories.

Updating a Design Database

Perform the following steps to update a design database:

1. Run `allegro_uprev` from your operating system command prompt.
If you type the command name without arguments, it displays command help.
2. Enter the appropriate file name and press Return/Enter.
The design is uprevd to the latest layout editor version.

The `allegro_uprev` command will produce a log file `output_db.log` that reports information and any error messages that have been reported.

allegro_uprev_overwrite

The `allegro_uprev_overwrite` batch command takes a design database from its current version to the latest version of the layout editor. You can use this command to uprev multiple design databases in a batch environment.

You can provide both databases and directories to the command. If a directory is encountered, uprev will recursively enter that directory and sub-directories until it encounters the nest directory depth limit or a read-only directory.

The command only processes database extensions and directories supported by the application. All other files are ignored.

If no database is provided as an input the command will process all files present in current working directory and all sub-directories.

Syntax

```
allegro_uprev_overwrite
```

Layout, drawing, or symbol file name (*.brd):

Output layout, drawing, or symbol file name (*.brd):

input_file	The name of the database you want to uprev. The default is .brd.
output_file	The name of the database after the uprev. Giving an output name that is different than the input name prevents the input database from being destroyed.
-n <nest directory depth>	Specify maximum depth to descend into a directory tree.
-drc	Updates all DRCs in batch mode.
-d	Runs command in debug mode.
-version	Prints the version.

Examples

```
allegro_uprev_overwrite *
```

Uprev all databases in current directory and any child directories up to a depth of 3.

```
allegro_uprev_overwrite *.pad
```

Uprev only padstacks found in current directory.

```
allegro_uprev_overwrite -d -n 1 symbols padstacks
```

Uprev and perform batch DRC on all databases found in symbols and padstacks directories. Do not descend to any sub-directories.

altsubclass

The `altsubclass` command changes the *Alternate Subclass* field in the *Options* panel of the Control Panel to the alternate subclass you specify when using *Route – Connect* ([add connect](#) command). The alternate subclass name can only be one recognized as a current alternate subclass of the subclass displayed in the *Options* panel.

Syntax

```
altsubclass[-+] [--] [altsubclass_name]
```

-+	Increments to the next alternate subclass.
--	Decrements to the previous alternate subclass.
altsubclass_name	Specifies the name of the alternate subclass to which you are changing.

anchor 3d view

The `anchor 3d view` command lets you specify an anchor point to define the area that is not affected by bending operations in 3D canvas. Once defined the location of the anchor point is saved in the database. You can, however, redefine the anchor point anytime.

When viewing a flex design in 3D canvas, you need to specify an area of the design that remains stationary. Anchor point is mainly selected in the rigid part of a rigid-flex design, but it can be placed either side of a bend line.

When bending the design in 3D canvas, all design elements that are on the other side of bending line moves, but the area where anchor point is marked remains static.

angle

The `angle` command lets you input an angle value, either an absolute angle (`angle`) or incremental from the current angle (see [iangle](#)).

Use `angle` for rotating elements in any command that allows rotation. For example, `move`, `add pin`, and `add symbol` applications have `Rotate` in pop-up menus. The `angle` command can also be used for applications expecting angular input, where angular dynamics is active and position readout shows an angle value. As a substitute for `Rotate`, `angle` provides the equivalent of selecting the `Rotate` pop-up, spinning the element to the appropriate angle, then clicking to choose that angle. When an application expects an angular input, `angle` provides the equivalent of clicking to choose an angle. For example, `spin` rotates a selected element and expects angular input. You can enter `angle` instead of clicking.

You can enter angle coordinates from the command prompt or bring up a dialog box into which you can enter the coordinates.

Syntax

```
angle [+ -] <
degree value
>
```

[+] indicates counterclockwise (default).

[-] indicates clockwise.

Specifying an Angle Value

You can input angle values by following these steps:

1. Run a command that supports rotation of an element; for example, `move`.
2. Choose the element to affect.
3. At the user interface command console, type `angle` without specifying coordinates.
A dialog box appears.
4. Enter the angle coordinates. `[+]` indicates counterclockwise (default). `[-]` indicates clockwise.
The selected element is rotated to that degree.
5. Choose Done from the right-button pop-up.

From the command prompt:

1. Run a command that supports rotation of an element; for example, `move`.
2. Choose the element to affect.
3. At the user interface command console, type `angle` and the coordinates. `[+]` indicates counterclockwise (default). `[-]` indicates clockwise.
The selected element is rotated to that degree.
4. Choose Done from the right-button pop-up.

Example

```
angle + 225
```

annotation in

The `annotation in` command lets you import an ASCII `.txt` file that contains the MANUFACTURING layer/MARKUP subclass information from a design opened in a different version of the layout editor, for example an Allegro X PCB Editor design opened in the Allegro Free Physical Viewer.

In addition, several people can work on a board, then export the data with the `annotation out` command, letting you then merge the multiple text files into your board.

Using this command you can load multiple annotation text files into one board. The data from the text files gets merged into the current board.

Access Using

- Menu path: *File – Import – Annotations*

The `annotation in` command opens a standard file browser.

Importing an ASCII File with Board Information

Before importing annotations from several sources, make sure they are not all called annotations.txt, or that they are in different directories. Because these are ASCII files you can rename the files. It is recommended that you keep the .txt extension.

1. Run `annotation in`.
The Annotation In file browser appears. The browser automatically looks for a file named annotations.txt in your current working directory. The filter is set to find all files with a .txt filename extension.
2. Check that the settings in the browser are correct for your needs.
3. Click *Open* to import the annotations into your design.
4. Continue importing data from other annotation files as necessary.
The data will be merged in your board.

annotation out

The `annotation out` command lets you export the MANUFACTURING layer/MARKUP subclass information of the current design in the form of an ASCII `.txt` file.

This lets you transfer drawing data from one design to another, or from one version of Allegro PCB, for example, Allegro Free Physical Viewer, to a full version of Allegro X PCB Editor.

This command also lets several people work on a board and export their work. The owner of the board can then use the `annotation in` command to import the data from the text files where it gets merged into the current board.

Access Using

- Menu path: *File – Export – Annotations*

The `annotation out` command opens a standard file browser.


Exporting Board Information in an ASCII File

Follow these steps to export board information to an ASCII file:

1. Run `annotation out`.

The Annotation Out file browser appears. The default settings in the browser are for the current working directory and the filename to be `annotations.txt`.


2. Check that the settings in the browser are correct for your needs.

 Note: If you are exporting information from several boards either ensure your `annotations.txt` files are in different directories or you change the name of the text file.

3. Click *Save* to save your annotations to a `.txt` file.

apd

The `apd` batch command that starts Allegro X Advanced Package Designer (APD) and lets you create and edit package and `.mcm` designs as well as symbols. You can also edit Allegro boards.

 If you do not include a design name and you have previously run this version of the layout editor, the last saved design in the previous session opens, based on information written to the `master.tag` file. If you do not want the layout editor to open the last design, move or delete `master.tag`. A new, unnamed design appears. To locate `master.tag`, open the `.ini` file, located in your `pcbenv` directory. Search `directory` to locate the file.

Syntax

```
apd <args> [-s <script>][-S <script>][-p <startdir>] [-j|-o<journal>][proj<cpm file>][-product<product name>][-option <option name>][-sq][-nographic|-nograph][-mps<XXX>][<design name>][<apd database>][-help][-version][-versionLong]
```

-s<script>	<p>Executes the specified script on startup. Up to 63 scripts are supported. The default extension is <code>.scr</code>. If you do not run the command in the directory where the script resides, you must include the path in the script's file name. The last directory in the <code>allegro.ini</code> file is not used. Multiple <code>-s</code> options may be specified, and they replay sequentially on the command line.</p> <p>Example:</p> <pre>-s script1 -s script2 ... -s scriptN</pre> <p>If no scripts are specified on the command line (<code>-s</code> or <code>-S</code> options) and the environment variable <code>script_startup</code> has a value, then a script named <code><program> <script startup value>.scr</code> in the startup directory replays. If the variable <code>script_startup</code> is <code>foo</code>, then the script is <code>apd_foo.scr</code>.</p>
-S<script>	<p>Executes the specified script file and sets the startup directory to the last directory stored in the <code>allegro.ini</code> file unless you specify <code>-p</code> or board name. Multiple <code>-S</code> options are permitted (identical to those for <code>-s</code>).</p>
-p<start directory>	<p>Specifies the startup directory, ignoring the <code>allegro.ini</code> file.</p> <p>If you run this command with a design file name that includes a path—for example, <code>/home/dkm/pcb/boards/layout123</code>—other files created during processing, such as log files, are created in the directory you specified and not in the directory where the design resides.</p>
-j -o<journal file>	<p>Starts a journal file that records your work session. The <code>.jrl</code> extension appends to the specified file name. The default journal file is <code><prog>.jrl</code>.</p>
-proj<cpm file>	<p>Reads the HDL-indicated <code>.cpm</code> file on startup. The initial starting directory and design name should be specified in the <code>.cpm</code> file.</p>
-product<product name>	<p>Starts the specified product tier of the layout editor for which you are licensed. If you do not specify a tier, the Cadence Product Choices dialog box appears, from which you choose one.</p> <p>The legal values for <code><product name></code> are shown below. This overrides any default set in a <code>.ini</code> file.</p> <p>Use <code>-product help</code> for a list of available products.</p>
-option <option name>	<p>Specifies the options to be run. Used with the product option to specify the product and option required. The option may be specified multiple times. Use <code>-product help</code> for a list of available products and options. License is one of the following. Each line lists the license name, product code, and actual layout editor name.</p>
-sq	<p>Starts SI (SPECCTRAQuest) for APD packaging.</p>
-nographic -nograph	<p>See the description of -nographic.</p>
-mps<XXX>	<p>Standard Cadence MPS argument support (This is not typically required.)</p>
<apd database>	<p>Start editing with this database (ignore <code>.ini</code> file); default extensions are <code>.brd</code>, <code>.mcm</code>, <code>.dra</code>, <code>.mdd</code>.</p>

A Commands


A Commands--apd

<design name>	<p>Opens the specified design file. The default extension is <code>.mcm</code>.</p> <p>If you do not run the command in the directory where the script is located, you must include the path in the script's file name.</p> <p>If you do not include a design name or if the layout editor cannot find the specified file, the layout editor opens a default file called <code>unnamed.mcm</code>.</p>
-help	Prints information about this command. For UNIX systems only.
-version	Prints the program's version and exits. For UNIX systems only.
-versionLong	Prints the program's long version, if available, and exits. For UNIX systems only.

aperture

The `aperture` command displays the Edit aperture Wheels dialog box that you use to generate and/or edit aperture wheel specifications for photoplotting. In vector-based artwork, you generate a list of the apertures that the photoplotter needs to make the artwork film. This list is generated for you in a file called `art_aper.txt`. (Not required for raster formats.)

You specify one or more wheels for apertures in the aperture list, after which you apply the apertures to the wheel. You can use the Automatic Aperture Editor to apply all of the apertures that the photoplotter needs to a wheel, and to display a table of the aperture data. You can edit and manipulate this aperture data. When your edits are complete, you generate generate the `art_aper.txt` file.

 For additional information about generating artwork, see *Preparing Manufacturing Data* in the user guide. For information on setting artwork parameters interactively, see [film param](#).

Related Topics

- [Specifying an Aperture Wheel](#)
- [Editing Aperture Data](#)

Edit Aperture Wheels Dialog Box

Access Using

- Menu path: *Manufacture – Artwork*

Alternately, access this dialog box by typing `aperture` in the command console or by clicking *Apertures* in the Artwork Control Form.

<i>Add</i>	Adds a new wheel to the aperture list.
<i>Undo delete</i>	Undoes the last deletion you made.
<i>Wheel</i>	Change a wheel number by clicking the number and typing in a new number.
<i>Edit</i>	Lets you edit the aperture table for this wheel number.
<i>Delete</i>	Deletes the wheel number the button is next to.

Edit Aperture Stations Dialog Box

Access by clicking *Edit* in the Edit Aperture Wheels dialog box.

<i>Station</i>	Specifies the aperture station.
<i>Geometry</i>	Specifies the geometry type.
<i>Width</i>	Specifies the width of the geometry.
<i>Height</i>	Specifies the height of the geometry.
<i>Rotation</i>	Specifies the rotation of the geometry.
<i>Auto</i>	Starts the automatic aperture editor. Initially, choose either Without Rotation or With Rotation. If you run the automatic aperture editor again, you must use With Rotation.
<i>Add</i>	Adds a new aperture to the table. Choose the relevant geometry option and insert the values in the aperture table. You also can add a flash to the table.
<i>Sort</i>	Sorts the apertures. You can sort by geometry or by the station on the wheel. Sorting by geometry groups all geometry types (for example, lines or circles) in the aperture table. Sorting by station lists apertures by their D-code.
<i>Undo delete</i>	Undoes the last deletion you made.

Related Topics

- [Editing Aperture Data](#)

Specifying an Aperture Wheel

If you want to specify an aperture wheel, perform the following steps:

1. Run `aperture` or click Apertures from the Artwork Control Form dialog box.
The Edit Aperture Wheels dialog box appears.
2. Use the Edit Aperture Wheels dialog box to add an aperture wheel to the aperture list. When the Edit Aperture Wheels dialog box is displayed for the first time, wheel 1 is added to the aperture list.

To add more wheels:

- Click *Add*.

To change a wheel number:

- Click the wheel number to be changed and enter the new number.

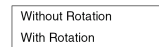
Applying Apertures to a Wheel

- In the Edit Aperture Wheels dialog box, click *Edit* for a wheel.
The Edit Aperture Stations dialog box appears.

You can add apertures to the wheel one at a time, or you can have the Automatic Aperture Editor apply all the apertures that the photoplotter needs to the wheel.


To use the Automatic Aperture Editor:

1. Click *Auto* in the Edit Aperture Stations dialog box.
A pop-up menu appears.



Rotated apertures are needed when flashes are used, and the aperture in the wheel that represents the flash is unsymmetrical.

2. Click one of the options in the pop-up menu.
The Automatic Aperture Editor starts. This editor creates an aperture table in the Edit Aperture Stations dialog box of the apertures it finds. The following figure shows an example of an aperture table produced by the editor.

 The editor does not perform the following tasks:

- Recreating existing apertures on a wheel
- Deleting unnecessary existing apertures
- Examining other wheels to determine whether apertures already exist

Related Topics

- [aperture](#)

Editing Aperture Data

After the editor creates the aperture table, you can perform one of the following tasks to manipulate its contents to meet company or manufacturing needs.

- Sorting Apertures
- Adding an Aperture
- Applying a Specific Wheel Number
- Changing the Units of Measurement
- Rotating Pad Geometries and Flashes

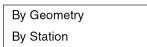
Sorting Apertures

You can sort apertures according to their station on the wheel or by their geometry. Sorting by geometry groups all geometries, such as lines and circles, in the aperture table. Sorting by station lists apertures according to their D-code.

To sort according to the D-code

1. Click *Sort* in the Edit Aperture Stations dialog box.

A pop-up menu appears:



2. Click *By Station*.

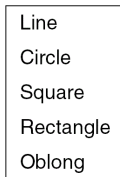
The table changes so that the Station column lists the lowest D-code first.

Adding an Aperture

To add a geometry or a flash to the aperture table, perform these steps:

1. Click *Add* in the Edit Aperture Stations dialog box.

A pop-up menu appears:



2. Click one of the geometry options or on the *Flash* option in the pop-up menu.

The new aperture appears in the aperture table.

3. Enter values for the new aperture.

Enter station and rotation values, plus height and width values for geometries, or the name for a flash.

Applying a Specific Wheel Number

In some cases, you may need specific wheel numbers with particular D-code associations in your aperture list.

Perform these steps to apply a specific wheel number:

1. Run *aperture*. The Edit Aperture Wheels dialog box appears. Then, click the wheel number and enter the appropriate number, or click *Add* in this dialog box to add a wheel.
2. Click *Edit*. The Edit Aperture Stations dialog box for this wheel is displayed. Click *Auto* to run the Automatic Aperture Editor.
3. Use the *Sort* and *Add* buttons to edit the table, or click the fields in the table and change their values. Use the *Delete* button to delete apertures that you do not need.

Changing the Units of Measurement

Click *Inches* at the top of the dialog box to change the units of measurement to inches. Click *Millimeters* to change them to millimeters.

Rotating Pad Geometries and Flashes

You can edit the rotation value of the D-code in the Rotation column.

The following rotation limits apply:

- Rotation for circles and lines is not allowed.
- Rotation for squares is 0 to 89.9 degrees.
- Rotation for rectangles and oblongs is 0 to 179.9 degrees.
- Rotation for flashes is 0 to 359.9 degrees.

To rotate pad geometries and flashes

1. Enter the number of degrees of rotation from 0.000 to 360.000.
2. Click *Auto* and run the Automatic Aperture Editor with the *With Rotation* pop-up menu option.
The Automatic Aperture Editor generates a D-code for each new pin instance on the design. For example, if you rotate the same oblong padstack to two separate rotation angles, the Automatic Aperture Editor generates two unique D-code definitions in the aperture table.

Generating the Aperture Data File

When you have finished with the aperture data, specified all the wheels and applied all the apertures, generate the aperture list. This list is generated in a file called `art_aper.txt`.

To generate the aperture list in the `art_aper.txt` file,

- In the Edit Aperture Wheels dialog box, click *OK*.

If no aperture errors are found when the file is produced, the Edit Aperture Wheel and Edit Aperture Station dialog boxes close. If errors are found, these dialog boxes remain displayed and the Aperture Table Errors window is displayed to show you the errors. You must fix the errors shown in this window before you can close the dialog boxes.

Related Topics

- [aperture](#)
- [Edit Aperture Wheels Dialog Box](#)

apick

The `apick` command, run at the command window prompt, lets you pick points based on polar coordinates, that is, distance and angle. If you do not provide any coordinates, a form appears where you can enter the distance followed by a form for the angle. The picks are absolute values and are not snapped to grid.

Syntax


The format is as follows:

```
apick distance angle
```

Highlighting Objects

Follow these steps to highlight objects in your design:

1. Make sure that you are in command mode, for example, add `connect`.
2. At the command window prompt, type `apick`.
3. Specify the distance and click *OK*.
4. Specify the angle in degrees and click *OK*.

 You can also type the distance and angle on the command line after typing the command name. For example, to specify distance as 1000 and angle as 45:

```
pick 1000 45
```

apick_to_grid

The `apick_to_grid` command is used in scripts to record mouse clicks that must be mapped to the grid. The format is the same as that of the [apick](#) command.

Example

```
pick_to_grid distance angle
```

artwork

The `artwork` batch command that creates photoplot film files (see also `film param`). To generate artwork data files, you must have previously:


- Specified artwork parameters
- Generated aperture lists if using vector-based artwork
- Created artwork film control records
- Saved the layout

The `artwork` program writes each artwork file as a separate ASCII file in the current directory. It writes all status information, warnings, and error messages into the file `photoplot.log`. Be sure to examine the log file carefully after every execution to discover any errors found by the `artwork` command program, correct them, then run the `artwork` program again.

Duplicate warnings, valid for multiple layers are reported once in the log file. To view all the warnings, set `artwork_allwarnings` in the *Manufacture – Artwork* category of the User Preferences Editor or add it to the `env` file in the `pcbenv` directory.

For information on additional aspects of generating artwork, see *Preparing Manufacturing Data* in the user guide.

Access Using

- Toolbar Icon: 

Syntax

```
-artwork [-s -p <-o outline_offset> <-a min_aperture><-f filename1> <-f filename2> <-f filename...> <-s> <-o distance> <-p> [-version] <board>
```

—or—

```
artwork -l <filename>
```

-s	Array outside shapes is not filled on a negative plane
-p	Uses vector pad-type behavior for raster artwork
-o outline_offset	Applies to negative films. Extends the shape boundary of the filled area by adding another outline in all directions around the design outline. Default is "shape bound box" obtained from the film record.
-a min_aperture	Sets the minimum aperture for vector artwork. Default is 3 mils.
-f filename	Films to generate from artwork control record. Default is all films
-l	List films in board, one per film
-version	Prints the version.
board	A <code>.brd</code> or <code>.mcm</code> file

Generating Artwork Data Files

To generate Artwork Data Files From an Operating System Prompt:

- Type either of the following at an operating-system prompt:

```
artwork [-s -p <-o outline_offset> <-a min_aperture><-f filename1> <-f filename2> <-f filename...> <-s> <-o distance> <-p> <board>
```

—Or—

```
artwork -l <filename>
```

If you enter the `artwork` command without specifying arguments, you are prompted for a layout name.

```
artwork
```

```
Existing layout file name (*.brd):
```

Example

The following `artwork` command line generates two artwork data files named `top.art` and `intl.art`,

```
artwork -f top -f intl design.brd
```

You can specify more than one `-f` option at a time on a command line.

If you enter with an `-f` option, an artwork data file name that you did not specify in the Film Control tab of the Artwork Control Form dialog box, the `artwork` program displays a message including all the artwork data file names you specified in that form. The following is an example of how the program responds to a mistake in the `-f` option.

```
artwork -f digl design.brd
```

```
'digl.art' does not exist as a film record in board (ignored)
```

```
List of film records:
```

```
top.art
```

```
sigl.art
```

```
ARTWORK finished
```

The following example shows the truncated contents of a vector artwork data file.

Example Vector Artwork Data File

```
D14*
```

```
X26543Y31496D02*
```

```
X1651D01*
```

```
X63Y-64D01*
```

```
Y-190D01*
```

```
X28D01*
```

```
X417Y508D02*
```

A Commands

A Commands--artwork

X-2413D01*

X508Y127D02*

X1079D01*

X-1969Y1206D03*

X-444Y-2540D03*

X4445Y-4635D03*

M02*

assemrules custom

The `assemrules custom` command will be documented in a future release.

asemrules standard

The `asemrules standard` command lets you perform specific design rule checking of several different rules in a package design. These rules allow you to gauge whether the package, as designed, will meet the physical and spacing requirements necessary for the part to be successfully manufactured and assembled.


Related Topics

- [Setting Up and Running the Assembly Rules Checker](#)




Assembly Design Rule Checks Dialog Box

Access Using

- Menu path: *Manufacture — Assembly Rules Checker*

- Toolbar Icon: 

When you run the `assemrules standard` command, the *Assembly Design Rule Checks* dialog box appears. Here you can select and define a variety of design rules to apply to your design.

Rule Selection	<p>This list shows all the assembly rules that you can apply to your design. The rules are grouped into related categories. If you select the text of a rule, a description of that rule and its corresponding constraints appear. Click the checkbox next to a rule to either enable or disable it for the DRC process. Click the checkbox next to a group folder to enable or disable all the rules in that category.</p> <p> Rules in the Wire group folder for which the selection check boxes are grayed out, indicate the checks that are always performed, and cannot be disabled.</p> <p> The rules under <i>Wire – Wire Online Physical</i>, <i>Wire Online Spacing</i>, and <i>Wire to Wire Online Spacing</i> – are always selected and cannot be changed.</p> <p>To apply all available rules to a design, select the All Rules check box.</p>
Report File Name	Allows you to specify the name of the tab-delimited report file that is generated by the DRC process. The default filename is <code>adrc_report.txt</code> . The report file is saved in the current working directory. (If the <code>ADS_SDRREPORTS</code> variable is set in your user preferences, then the file is saved in the specified subdirectory.)
<Rule Description Field>	Displays a graphical representation and a textual description of the selected rule.
OK	<ul style="list-style-type: none"> If there are changes in the rules to be run, closes the dialog box and starts the batch process to run all the selected checks, generate a report, and display any DRC markers. If there are no changes in the rules to be run, closes the dialog box.
Close	Closes the dialog box, whether or not any rules are selected or checks have been performed.
Apply	Starts the batch process to run all the selected checks, generate the report and display it without closing the dialog box.
Edit Constraints	<p>Launches Constraint Manager, the application used for capturing constraints for each assembly rule selected in the Rule Selection field. The Assembly Rules Constraints are captured in the worksheets in the Assembly tab.</p> <p> When you select a constraint and launch Constraint manager, the constraint opens in the <i>Assembly Constraint Set</i> worksheet. The recommended use model is to create an Assembly Rules Constraints Set (ACSet) and apply this to the design objects.</p>
Help	Invokes context-sensitive Help for this command.
Clear Existing Rule Violations	<p>Removes all Assembly Rules Checker violation markers from the database.</p> <p>As a result, the DRC markers are removed from the physical layout, as well as from the worksheets in the DRC domain in Constraint Manager.</p>

DRC Report

A tab-delimited report file is generated when the Assembly Rules Checker process finishes. The default filename for the report is `adrc_report.txt`. You can load this report file into a spreadsheet editor. You can cross-probe from the report to the design by clicking on the error location coordinates (*Overlap Location*) listed in the report. The rule sections are listed in the report in the order in which they are executed, which matches the order in which they are displayed in the Rule Selection list of the *Assembly Design Rule Checks* dialog box.

A Commands
A Commands--assemrules standard

=====				
Rule:	Cline to Via Overlap			
Constraint:	None			
Info:	0	Clines (not including Bond Wires) on ETCH layers found in design.		
Info:	0	Vias (not including Bond Fingers) on ETCH layers found in design.		
Info:	No vias (excluding Bond Fingers) and/or clines (excluding Bond Wires) on ETCH layers found. No checks can be performed.			
Overlap Location	Cline Layer	Cline Net Name	Via Layer	Via Net Name
(4822 196.18)	M4_SIG	SAW DCSX	M5_SIG	GROUND
(-1166 52 -2417)	M4_SIG	TX_CP	M5_SIG	VDDA
Violations:	2			
=====				

Log File

The log file is a list of which rules were run and the number of violations that were discovered for each rule. The log file also contains any warning or error messages that were generated during the execution of the checks. The default filename for the log is `adrc.log`.


```
Wed May 17 10:20:11 2006: Assembly Design Rule Checks
(-----)
(
(      Assembly Design Rule Checks      )
(
(      Drawing       : New wirebond_start.sip      )
(      Software Version : 15.7b014                )
(      Date/Time      : Wed May 17 10:20:11 2006    )
(
(-----)


Batch will check rule Wire Maximum Angle to Bond Finger
Rule:      Wire Maximum Angle to Bond Finger
Violations:      0
ADRC: Found 0 violations of the selected rule(s).
```

Setting Up and Running the Assembly Rules Checker

To set up and run the Assembly Rules Checker:

1. Run the `assemrules standard` command (*Manufacture – Assembly Rules Checker*).
The *Assembly Design Rule Checks* dialog box appears.
2. Select a rule that you want to run by clicking the check box next to the name of that rule in the *Rule Selection* list. (A check mark in the box indicates that the rule is enabled.)
The rule name appears along with a description of the rule.

 You can enable all the rules in a group by selecting the check box for an entire group.
3. To modify the constraint value, select *Edit Constraints*.
Constraint Manager is launched. Modify the values as required.
4. Repeat Steps 2 and 3 for all other rules that you want to run.
5. Click *OK* to close the dialog box and start the batch check process. (Or, click *Apply* to run the checks without closing the dialog box.)
When the check process is completed, a report file is generated that lists all the violations. DRC markers appear in the design wherever a violation occurs.

 The DRC markers are also listed in the DRC worksheet in Constraint Manager.
6. Browse the design for the DRC markers and determine whether the violation is acceptable, or correct the design accordingly.
7. Repeat the entire process until all DRC violations have been resolved and you are satisfied that the design is acceptable.

Related Topics

- [assemrules standard](#)

assign color

The `assign color` command assigns a color and highlights an element without requiring the use of the *Color* dialog box. Changing the color or highlighting with this command automatically updates the *Nets* section of the *Color* dialog box as well.

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

- Symbols
- Functions
- Nets
- Pins
- Vias
- Fingers
- Clines
- Lines
- Bond Wires
- Shapes
- DRC Errors

Related Topics

- [Assigning a Custom Color or Highlighting an Element](#)

Assign Color Command: Options Panel

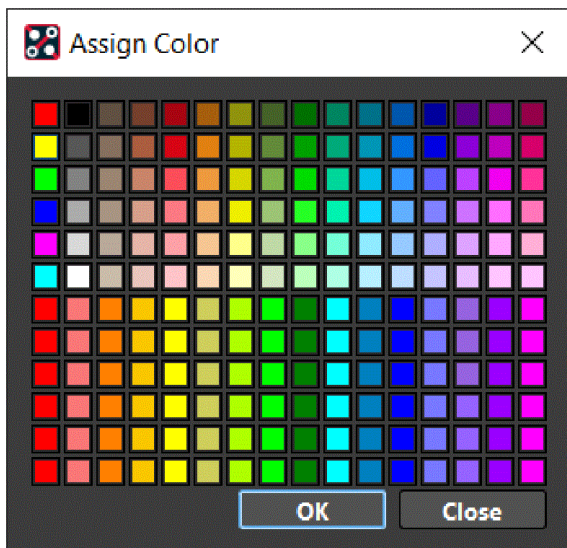
Access Using

- Menu path: *Display – Assign Color*
- Toolbar Icon: 

The following display only when you choose the *Display – Assign Color* menu item.

<i>Selected color</i>	Indicates the currently chosen color. A palette of 24 colors is available with an option of view complete color palette.
<i>More Colors</i>	Displays a palette of 192 colors.
<i>Highlight Pattern</i>	Click to accentuate certain elements with a pattern—or striping—comprising the element’s base subclass color and the temporary highlight color defined in the <i>Display</i> category of the <i>Color</i> dialog box. If the element is a net, it becomes highlighted in the design canvas and its color also displays in the <i>Nets</i> section of the <i>Color</i> dialog box. Striping is only visible when the <code>display_nohilitefont</code> variable is disabled.

In the pre-selection mode, after you right-click and choose *Assign Color*, the following palette displays:



Assigning a Custom Color or Highlighting an Element

Perform the following steps to assign custom colors to an element:

1. Hover your cursor over an element.
2. Right-click and choose *Assign Color* from the pop-up menu.
The color palette displays.
3. Click the color box of the new color for the element. The selected color displays in the bottom right of the palette, and the element's color changes in the design canvas and in the *Nets* section of the *Color* dialog box.
4. Click *Highlight Pattern* to accentuate certain elements with a pattern in the selected color if required.
If the element is a net it becomes highlighted in the design canvas and its color also displays in the *Nets* section of the *Color* dialog box.

Related Topics

- [assign color](#)

assign multi nets

The `assign multi nets` command lets you select a list of source nets to assign to a list of target pins. You can select, filter, and order a list of target pins and a list of source nets. Then, you can assign the selected source nets in order to the target pins.

In a co-design environment, this command assigns floating ports on nets to pins on a co-design component. The logical properties of the assigned ports are moved to the pins and the port names become the pin names.

For additional information about managing net assignments for multi-die packages see *Routing the Design in the user guide*.

Related Topics

- [Assigning Multiple Nets to Pins](#)

Multi-Net Assignment Dialog Box

Access Using

- Menu path: *Logic – Assign Multiple Nets*

When you run the `assign multi nets` command, this dialog box appears. Use it to select the source and target nets, and then assign them. Once you make the multi-net assignments, you can review the results.

<i>Assign selected source nets to selected target pins</i>	Specifies the task you perform when you complete the fields in the Multi-Net Assignment dialog box and commit the changes to the database.
<i>Source Nets Frame</i>	<i>Specifies the left frame of the dialog box where you select and filter nets for the Source Nets list.</i>
<i>Use Existing Nets</i>	Includes only nets existing in the package design database for assignment to pins. When you choose this option, the <i>Select Nets</i> drop-down list is enabled. This is the default selection.
<i>Select Nets</i>	<p>Specifies a drop-down list that includes these selections:</p> <ul style="list-style-type: none"> • <i>All</i> <p>Initially selects all nets in the design for consideration for the source list, and then filters according to the filtering criteria. This is the default selection. When you choose <i>All</i>, the drop-down list to the right of this field is disabled.</p> • <i>by Refdes</i> <p>Enables the alphabetically-sorted, drop-down list to the right of it. Only nets assigned to pins from the chosen component are selected for initial consideration, subject to the filtering criteria. Once you choose a different refdes during the session, toggling the <i>Select Nets</i> drop-down list preserves the last value you selected for the next time you choose the <i>by Refdes</i> selection.</p> • <i>by Class</i> <p>Enables the drop-down list to the right of it. Initially, the class <i>IC</i> appears in the list. Only nets assigned to pins of instances of components of the chosen class are selected for initial consideration, subject to the filtering criteria. Once you choose a different class during the session, toggling the <i>Select Nets</i> list preserves the last value you selected for the next time you choose the <i>by Class</i> selection.</p> • <i>by Device</i> <p>Enables the alphabetically-sorted, drop-down list to the right of it. Only nets assigned to pins of instances of the chosen device type are selected for initial consideration, subject to the filtering criteria. Once you choose a different device type during the session, toggling the <i>Select Nets</i> drop-down list preserves the last value you selected for the next time you choose the <i>by Device</i> selection.</p>
<i>Create New Nets</i>	<p>Creates new nets for assignment to the target pins. When you choose this option, the <i>Pattern</i> field is enabled. Also, the <i>Filters</i>, <i>Select Nets</i>, and <i>Remove Selected Nets</i> sections of the <i>Source Nets</i> frame are disabled, and the <i>Source Nets</i> table becomes read-only. The table will be populated according to the value typed in the <i>Pattern</i> field.</p> <p>If any of the net names listed or generated by the template already exist, then the existing net will be used, rather than a new net being created.</p>
<i>Pattern</i>	<p>Accepts a list of new net names or a template pattern for creation of new net names. You can type in a single name or an ordered list of names separated by spaces, for example, <code>ABC PQR XYZ</code>. You can use the characters <code>'</code>, <code>,</code> and <code>*</code> to form template patterns that generate an ordered list of new net names.</p> <p>For example <code>ADDR[0-15]</code> generates the ordered list of sixteen net names <code>ADDR[0]</code>, <code>ADDR[1]</code> ... <code>ADDR[15]</code>. Similarly, <code>ADDR[0-*]</code> generates as many net names as are needed to match the number of pins in the <i>Target Pins</i> list, starting with <code>ADDR[0]</code> and counting up until the number of pins is matched. You can only use the <code>*</code> immediately after a <code>-</code> to create a net generator pattern. Also, <code>ADDR[1,3,5-7]</code> generates the ordered list of 5 net names <code>ADDR[1]</code>, <code>ADDR[3]</code>, <code>ADDR[5]</code>, <code>ADDR[6]</code>, and <code>ADDR[7]</code>. Note that you must not follow the <code>,</code> by a space character. Space, <code>'</code> and <code>*</code> are not allowed as characters in the net names. You can use <code>\</code> in a net name by preceding it with <code>\</code>. You can combine specific names and net name generator patterns into one string separated by spaces, such as: <code>A0 D7-15 A31</code>.</p>

A Commands

A Commands--assign multi nets

Source Nets table	<p>Displays a sorted table of source nets matching the specified filters. If you choose the <i>Create New Nets</i> option, this is a read-only list that is controlled only by the patterns typed into the <i>Pattern</i> field.</p> <p>If you choose the <i>Use Existing Nets</i> option, then the list is taken from existing nets filtered by the filtering options. Some nets may be missing from all those matching the filters because you removed them by clicking the <i>Remove Selected Net</i> button.</p> <p>When initially created, the table is sorted in ascending order by net name. You can toggle the sort order and direction by right-clicking on one of the column headers: <i>Net</i> or <i>Pin</i>. A pop-up menu appears with two options: <i>Sort Ascending</i> and <i>Sort Descending</i>. Choosing one of these sorts the table on the selected column in the chosen direction.</p> <p>When you click the <i>Apply</i> button, if the <i>Target Pins</i> table and the <i>Source Nets</i> table have the same number of items, then the nets in the <i>Source Nets</i> table are assigned to the pins in the <i>Target Pins</i> table in order from top to bottom. You can select individual nets from the table, and then remove them from the list by clicking the <i>Remove Selected Net</i> button.</p> <p>If you change any of the filter options, any nets removed from the list are restored if they match the new filter settings. You can also change the ordering of the source list by right-clicking on any row of the table. A pop-up menu appears with commands <i>Top</i>, <i>Bottom</i>, <i>Up</i>, and <i>Down</i>. The <i>Up</i> and <i>Down</i> selections move the net in the row up or down one row of the table. The <i>Top</i> and <i>Bottom</i> selections move the net to the top or bottom of the table, respectively.</p>
Filter	<p>Specifies the fields in which you can enter regular expressions for filtering. Depending on the column in which you type the filter string (<i>Pin</i> or <i>Net</i>), only those nets which match the regular expression for that column are selected in the source nets list.</p> <p>The regular expression language definition is a slightly simplified version of the Cadence regular expression language for Pattern Matching Functions described in the document <i>SKILL Reference Manual: Language Fundamentals</i>. This is the language used by the SKILL function <code>rexCompile</code>. The simplification for the <code>assign multi nets</code> command filters is that <code>*</code> matches 0 or more occurrences of any character. In the standard SKILL regular expression language, <code>*</code> matches 0 or more occurrences of the preceding regular expression form. The regular expression form <code>'.'</code> matches any character. So, the filter value <code>A*</code>, which matches any name starting with A, is equivalent to the SKILL regular expression <code>A.*</code>. For the <i>Pin Name</i> column, the layout editor filters on pin name only. This makes it easy to filter on pin name without having to include the component reference designators in the filter string. Even so, all pin names appear in the table prefixed with their component reference designator and a <code>'.'</code> character. So, for example, if the pin name filter is <code>AD*</code>, both pins <code>U1.AD0</code> and <code>U1.AD1</code> appear in the list. You can use the <i>Source Nets</i> drop-down lists to control which component pins' nets appear in the table.</p>
Reset	Restores any removed nets to the <i>Target Pins</i> table so it again matches the specified <i>Target Pins</i> filters. This button is disabled if you click the <i>Create New Nets</i> radio button.
Remove Selected Net button	If any nets in the <i>Source Nets</i> table are currently selected, it removes them from the table. This button is disabled if you clicked the <i>Create New Nets</i> radio button.
Number of Selected Nets	Shows the total count of the number of source nets in the list.
Target Pins Frame	Specifies the right frame of the dialog box where you select and filter pins for the <i>Target Pins</i> list.
Select Pins	<p>Specifies a drop-down list that includes these selections:</p> <ul style="list-style-type: none"> <i>All</i> - Initially selects all pins in the design for the <i>Target Pins</i> list and then filters according to the filtering criteria. This is the default selection. When you choose <i>All</i>, the drop-down list to the right of this field is disabled. <i>by Refdes</i> - Enables the alphabetically-sorted, drop-down list to the right of this field. Only pins from the chosen component are selected for initial consideration, subject to the filtering criteria. Once you choose a different refdes during the session, toggling the <i>Select Pins</i> drop-down list preserves the last value selected for the next time you choose the <i>by Refdes</i> selection. <i>by Class</i> - Enables the drop-down list to the right of this field. Initially, class <i>IC</i> appears in the list. Only pins from instances of components of the chosen class are selected for initial consideration, subject to the filtering criteria. Once you select a different component class during the session, toggling the <i>Select Pins</i> drop-down list preserves the last value selected for the next time you choose the <i>by Class</i> selection. If you remove pins and then change the settings, the removed pins appear again in the table. <i>by Device</i> - Enables the alphabetically-sorted, drop-down list to the right of this field. Only pins from instances of the chosen device type are selected for initial consideration, subject to the filtering criteria. Once you choose a different device type during the session, toggling the <i>Select Pins</i> drop-down list preserves the last value selected for the next time you choose the <i>by Device</i> selection.
Net reassignment allowed	Allows you to reassign nets. If you do not check this box, then you cannot assign any destination pin that already has a net assignment. Therefore, all such pins that would otherwise appear in the <i>Target Pins</i> table are removed from the table. With this box set, the <i>Net</i> column of the <i>Target Pins</i> table is blank. The default setting of the box is checked. Checking this box, when it had been unchecked, restores to the <i>Target Pins</i> table any pins matching the filter that were removed because they had a net assignment. Checking the box also restores any pins explicitly removed when you clicked the <i>Remove</i> button.

A Commands

A Commands--assign multi nets

<i>Target Pins table</i>	<p>Displays a sorted table of target pins matching the specified filters. Some pins may be missing from all those matching the filters if you clicked <i>Remove Selected Pin</i>.</p> <p>When initially created, the table is sorted in ascending order by pin name. You can toggle the sort order and direction by right-clicking on one of the column headers: <i>Pin</i>, <i>Pin Use</i>, or <i>Net</i>. A pop-up menu appears with two options: <i>Sort Ascending</i> and <i>Sort Descending</i>. Choosing one of these sorts the table on the selected column in the chosen direction.</p> <p>When you click <i>Apply</i>, if the <i>Target Pins</i> table and the <i>Source Nets</i> table have the same number of items, then the <i>Nets</i> in the <i>Source Nets</i> table are assigned to the <i>Pins</i> in the <i>Target Pins</i> table in order from top to bottom. You can also select individual pins from the table and remove them by clicking <i>Remove Selected Pin</i>.</p> <p>You can change the filter options. Any pins removed from the table are restored if they match the new filter settings. You can also reorder the <i>Target List</i> by right-clicking on any row of the table. A pop-up menu appears with commands: <i>Top</i>, <i>Bottom</i>, <i>Up</i>, and <i>Down</i>. The <i>Up</i> and <i>Down</i> options move the pin in the row up or down one row in the table. The <i>Top</i> and <i>Bottom</i> selections move the pin to the top or bottom of the table, respectively.</p> <p>You can also populate by pick, window select, or choosing <i>Temp Group</i> from the pop-up menu in the graphical window. Only pins are enabled for selection. The set of selected pins is filtered by the filter settings and those that match are populated into the table replacing the previous contents of the table. If the filter settings are changed subsequently, all pins matching the filter settings are restored to the table, thus replacing any previous graphical selection of pins.</p>
<i>Filter</i>	<p>The first row of the <i>Target Pins</i> table is a row of type-in fields in which you can enter regular expressions.</p> <p>The regular expression language definition is a slightly simplified version of the Cadence regular expression language for Pattern Matching Functions described in the document <i>SKILL Reference Manual: Language Fundamentals</i>. This is the language used by the SKILL function <code>rexCompile</code>. The simplification for the <code>assign multi nets</code> command filters is that <code>*</code> matches 0 or more occurrences of any character. In the standard SKILL regular expression language, <code>*</code> matches 0 or more occurrences of the preceding regular expression form. The regular expression form <code>'</code> matches any character. So, the filter value <code>A*</code>, which matches any name starting with <code>A</code>, is equivalent to the SKILL regular expression <code>A.*</code>.</p> <p>Depending on the column in which you type the filter string (<i>Pin</i>, <i>Pin Use</i>, or <i>Net</i>), only those pins, which match the regular expression for that column are selected in the <i>Target Pins</i> list. The layout editor filters only on the pin name in the <i>Pin</i> column. This makes it easy to filter on the pin name without having to include the component reference designators in the filter string. Even so, all pin names appear in the table prefixed with their component reference designator and a character. For example, if the pin name filter is <code>AD*</code>, both pins <code>U1.AD0</code> and <code>U1.AD1</code> appear in the list. You can use the <i>Select Pins</i> lists to control which components' pins appear in the table.</p>
<i>Reset</i>	Restores any removed pins to the <i>Target Pins</i> table so it again matches the specified <i>Target Pins</i> filters. Clicking <i>Reset</i> also restores all pins in the design that match the filters to the table. Thus any previous graphical selection of pin is ignored.
<i>Remove Selected Pin</i>	If any pins in the <i>Target Pins</i> table are currently selected, it removes them.
<i>Number of Selected Pins</i>	Shows a total of the number of target pins in the list.
<i>Assign</i>	Assigns the first <i>source net</i> to the first <i>target pin</i> , the second <i>source net</i> to the second <i>target pin</i> , and so on until all of the source nets are assigned in order to the target pins. This occurs only if the same number of nets are in the <i>Source Nets</i> table and number of pins in the <i>Target Pins</i> table. If the number of items in each table differs, you are unable to use the <i>Assign</i> button.
<i>Undo</i>	Lets you undo the most recent assignment change you made when you clicked the <i>Assign</i> button. This occurs only if you have not committed the change by clicking <i>Apply</i> . If there are no uncommitted assignments, then the <i>Undo</i> button is disabled. Clicking <i>Apply</i> or <i>Undo</i> disables the <i>Undo</i> button. Clicking <i>Assign</i> re-enables the <i>Undo</i> button.
<i>OK</i>	Commits all previously made changes and exits from the command.
<i>Apply</i>	Commits all assignment changes you made when you clicked the <i>Assign</i> button. This button becomes enabled once you use the <i>Assign</i> button to make assignments. The <i>Apply</i> button then becomes disabled until you make more assignments using the <i>Assign</i> button. Once you commit the assignment changes using the <i>Apply</i> button, you cannot undo them using the <i>Cancel</i> button.
<i>Cancel</i>	Lets you undo all previously made changes, except for those committed when you clicked the <i>Apply</i> button, and dismisses the dialog box.
<i>Help</i>	Displays the user documentation for the <code>assign multi nets</code> command in a separate help tool window.

You can undo one level of assignment made by the *Assign* button using the *Undo* button (or *Oops from the pop-up menu*). Once you are satisfied with the results, you can permanently commit the assignments to the database with the *Apply* button (or *Apply from the pop-up menu*). Clicking *Apply* commits all assignments performed using the *Assign* button since the last *Apply*. The *Cancel* button (or *Cancel from the pop-up menu*) discards any assignments made with the *Assign* button since the last *Apply*, and exits the command. The *OK* button (or *Done from the pop-up menu*), commits all uncommitted *Assigns* in the session and exits from the command.

Assign Multiple Nets Pop-up Menu


This pop-up menu appears when you right-click in the Design Window. It contains the following options:

<i>Done</i>	Exits the <code>assign multi net</code> command and commits all the assignments that have been made. This is the same as clicking <i>OK</i> in the Multi-Net Assignment dialog box.
<i>Apply</i>	Commits any assignments made by the <i>Assign</i> command since the last <i>Apply</i> . This is the same as clicking <i>Apply</i> in the Multi-Net Assignment dialog box.
<i>Cancel</i>	Cancels the command without committing any of the net assignments performed during the command. This is the same as clicking the <i>Cancel</i> button in the Multi-Net Assignment dialog box.
<i>Assign</i>	Performs the assignment if the currently selected list of source nets can legally be assigned to the currently selected list of target pins. This is the same as clicking <i>Assign</i> in the Multi-Net Assignment dialog box. If the number of items in each table differs, then you are unable to use the <i>Assign</i> button.
<i>Oops</i>	Lets you undo the most recent net assignment. This is the same as clicking <i>Undo</i> in the Multi-Net Assignment dialog box.
<i>Temp Group</i>	Lets you select the objects graphically. Then choose <i>Complete</i> from the pop-up menu and the original pop-up menu re-appears to allow the command sequence to continue.

Assigning Multiple Nets to Pins

This procedure is based on an early I/O feasibility study that is being considered for a multi-die package with at least two co-design dies.

1. Import the standard dies that already have some existing die pin layout through DEF (*Add – Standard Die – DEF*), die text files (*Add – Standard Die – Die-Text-In Wizard*), or DIE (*Add – Standard Die – D.I.E. Format*) files.
This may actually import some existing net names into the package database.
2. Place the dies in some appropriate configuration in the package using the *Edit – Move* (`move` command).
3. Create the co-design dies, die pin patterns, and determine placement location using the *Add – Co-Design Die* (`add codesign die` command).
At this point several unconnected dies exist in the package. It is necessary to establish the connectivity between them.
4. Do one of the following:

 If this is a package-driven flow, see Step 6 before performing this step.

- a. Choose *Logic – Assign Multiple Nets* (`assign multi net` command) from the menu bar. In the Multi-Nets Assignment dialog box, select lists of pins from some die and then assign them to lists of nets, probably assigned to pins of other dies.
- b. If there are no appropriate existing nets to assign to the pins, choose *Logic – Auto Create Net* (`auto create net` command) to select the list of pins and create a list of nets to assign to them or click *Create New Nets* in the Multi-Nets Assignment dialog box.

To select the groups of pins for assignment requires that you assign an order, either alpha-numerically or by explicit listed order in a spreadsheet table. Assign the nets from the ordered list of source nets to the ordered list of target pins starting with the first net and pin in each list, and continuing in parallel order down the two lists.

5. For wire bonded dies, you must create the wire bonds to establish the order of the pin escapes onto the package substrate. Use the *Route – Wire Bond* toolset in the menu bar. Also, wire bond die to die to verify direct die to die net assignment and connection.
6. Use both the *Logic – Assign Multiple Nets* (`assign multi nets`) and *Logic – Auto Create Net* (`auto assign net`) commands to make net assignments between the die pins and package pins.
In a package-driven flow, it is possible that the package netlist, that is, the nets connected to the package pins, existed before you added the dies to the package. In this case, you have to perform this step before step 4. You should add the dies as co-design dies to preserve any IC net names and the mapping between them and package nets. Then, assign package pins to die pins using the *Logic – Assign Multiple Nets* (`assign multi net`) command, before assigning die pin to die pin. Doing the package-pin to die-pin assignments first ensures that all the die pins connected to nets on package pins preserve the package net names, which is essential for a package-driven netlist.
7. Complete the remainder of package planning and mock-up of the I/O cells and die pins on each IC as per existing flows.
8. Using DEF, export the co-design die to the IC tool.

Related Topics

- [assign multi nets](#)

assign net

The `assign net` command assigns pins to an existing net. You choose the net and then the pin to be assigned to the selected net.

❗ When you backannotate this design in Allegro Design Entry HDL XL or System Connectivity Manager, the logic is not updated.

In a co-design environment, this command assigns floating ports on the selected net to a pin on the selected co-design component. The logical properties on the port are moved to the pin and the port name becomes the pin name.


Assigning pins to a net is part of the flow of sequences you perform when manually defining connectivity. For additional details about connections and routing, *Routing the Design in the user guide*.

Related Topics

- [Assigning Pins to an Existing Net](#)

Assign Net Command: Options Panel

Access Using

- Menu path: *Logic – Assign Net*
- Toolbar Icon: 

<i>Re-assign Pin Allowed</i>	Lets you use previously assigned pins during net assignment or reassign the nets.
<i>Propagate to connected items</i>	Lets you assign all objects on the same branch as the selected pin or shape to the new net.

Assigning Pins to an Existing Net

To assign pins to an existing net, perform these steps:

1. Run `assign net`.

You are prompted to enter a selection point (on the net).

⚠ In a co-design environment, the `logic_edit_enabled` environmental variable in the Logic category of the User Preferences Editor must be set to be able to assign logical ports.

2. If you want to reassign previously assigned pins, click the *Re-assign pin allowed* button in the *Options* panel.
3. Identify the net to which pins will be assigned by selecting a point on the net, or use the Find Filter and Find by Name feature to choose the appropriate net.
The layout editor highlights the selected net, identifies the net name in the command line, and asks you to choose a pin to be assigned.
4. Choose the pin to be assigned to the net selected in step 3.

✓ You can select the *Propagate to connected items* to easily select a branch and assign all objects in the branch to the new net.

The layout editor highlights the pin, adds it to the net, and displays the ratsnest line for the net. If the selected pin is currently assigned and pin reassignment is allowed (as indicated in the *Options* panel), the pin from the old net is removed and added to the new net.

⚠ If the reassigned pin had existing connections, DRC errors may occur.

If you choose a currently assigned pin, and pin reassignment is not allowed, the layout editor does the following:

- Tells you the pin cannot be reassigned
 - Deselects the pin
 - Asks you to make another pin selection
5. Continue selecting additional pins you want to assign to the net.
 6. Click right to display the pop-up menu and do one of the following:
 - To undo the last selection, choose *Oops*.
 - To cancel all selections and end the net assignment session, choose *Cancel*.
 - To complete the net assignment and end the net assignment session, choose *Done*.

The layout editor dehighlights the selected net and pins and exits the command.

Related Topics

- [assign net](#)

assign plating layer

The `assign plating layer` command lets you assign a plating bar layer to pins and nets. You make plating bar connections from the outermost point of a net on the plating layer, relative to the center of the package design.

Related Topics

- [Assigning a Plating Layer to Pins and Nets](#)

Assign Plating Layer Command: Options Panel

Access Using

- Menu path: *Route – Plating Layer Assign*

Use these controls to configure the plating bar parameters for pins and nets in your design.

<i>User advanced selection filtering</i>	Lets you filter out selected nets and pins from the plating bar layer assignment. Selecting the option displays the Advanced Selection Filtering dialog box when you choose nets or pins for plating bar layer assignment.
<i>Reassign allowed</i>	When checked, lets you replace the existing ASSIGN_PLATING_LAYER property on the selected pin/net with updated values.
<i>Assign power and ground nets</i>	When checked, the layout editor processes these nets as well as the signal nets.
<i>Derive from physical</i>	When checked, the selections you make get fixed to specific layers, based upon their physical connections; that is, the ending points of their via structures. Because the layer assignment is determined by the command in this mode, the assignment types and layer selections are disabled. Status messages display pin-to-layer assignments.
<i>Assignment Type:</i>	<p>This option lets you choose the type of layer assignment for your selection.</p> <p>Select Fixed in conjunction with the available plating bar layers. When you choose this assignment type, you also must choose a plating layer.</p> <p>Select Free to allow the routing layout editor to route the selections to the layer that best ensures a successful connection.</p> <p>Select Remove to delete the ASSIGN_PLATING_LAYER property from the selection.</p>
<i>Available Routing Layers:</i>	The list of available plating bar layers is based on your layer stack-up. You must choose one plating layer when your selection assignment type is Fixed. (To change a layer selection, you must deselect the current active layer first.)


Related Topics

- Advanced Selection Filtering Dialog Box

Assigning a Plating Layer to Pins and Nets

You can assign plating bar layers to pins and nets by following these steps:

1. Choose *Route – Plating Layer Assign*.
The *Options* panel of the user interface is reconfigured for the command.
2. Set the Find filter to choose pins, nets, or both.
3. Set the parameters in the *Options* panel for your first selection, as described in the section above.
4. Choose the design element to which you want to assign the ASSIGN_PLATING_LAYER property.

 To choose a group of elements with the same parameters:

- a. Right-click before making a selection.
- b. Choose *Temp Group* from the pop-up menu.
- c. Make your selections.
- d. When you have completed the selection process, click right again.
- e. Choose *Complete* from the pop-up menu. (Cancel terminates the command and returns the layout editor to an idle state.)

If you enabled the advanced selection filtering option in the *Options* panel, the Advanced Selection Filtering dialog box appears with a listing of your selected nets and pins.

5. Right-click to display the pop-up menu.
6. Choose *Done*.
The layout editor returns to an idle state.
Verify the results by running `property edit` and clicking on the assigned pins/nets.

Related Topics

- Advanced Selection Filtering Dialog Box
- [assign plating layer](#)

assign port

An internal Cadence engineering command.

assign portgroup

An internal Cadence engineering command.

assign power

An internal Cadence engineering command.

assign refdes

The `assign refdes` command assigns reference designators to package symbols. It creates a reference designator for each component. Reference designators are expected to consist of a prefix and a series-number, for example, U6, C128, or RP09. The prefix is one or more alphabetic characters. The `assign refdes` command chooses the next higher number in that prefix series for the next reference designator. For example, if the last in the RP* series was RP9, the next is RP10.

You can specify the reference designator prefix for the component containing a function in either of two ways:

- Attach to that function the REF_DES_FOR_ASSIGN property with the required prefix value.
Then execute the `assign refdes` command.
- Specify the reference designator prefix for the component by adding the prefix to the component package symbol.
For example, if you add the prefix ABC*, assign assigns the sequences of reference designators ABC1, ABC2, and so on, to those components.

Related Topics

- [Assigning Reference Designators to Package Symbols](#)

Assign Refdes Command: Options Panel

Access Using

- Menu path: *Logic – Assign Refdes*

<i>Refdes</i>	Lets you type in a reference designator or click <i>Browse</i> to choose one from the object browser.
<i>Refdes increment</i>	Lets you provide a reference designator value

Assigning Reference Designators to Package Symbols

To assign reference designators to package symbols, perform the following steps:

1. Run `assign refdes`.
2. In the *Options* panel, provide a reference designator in the Refdes field and a value in the Refdes increment box if you are assigning a group of reference designators to symbols.
3. Press Enter on the keyboard.
4. Click to choose the package/part symbol to which you are assigning the reference designator.
5. If applicable, choose the next symbol to which you are applying a reference designator.
The editor uses your reference designator definition, plus the specified increment, to assign the reference designator.
6. Repeat step 5 as needed.
7. Click right to display the pop-up menu and choose *Done*.

Related Topics

- [assign refdes](#)

assign region

The `assign region` command lets you select multiple shapes and assigns single region to them. This command displays the *Assign to Region* dialog box to create a new region or select an existing region for assigning to selected shapes.


The `assign region` command is available in the *General edit* application mode. The command functions in a pre-selection use model, in which you choose an element first, then right-click and execute the command.

Assigning Region to Multiple Shapes

Perform the following steps to assign region to multiple shapes:

1. Select multiple shapes. You can select multiple shapes with a single pick, window drag, Select by Polygon selection modes.
2. Right-click and choose *Assign to region* from the pop-up menu or run the `assign region` command.
The *Assign to Region* dialog box displays.
3. Enter a new name in the *Enter new region name* or select an existing region from the *pick region to assign to shape(s)* list.
4. Click *OK* to assign the region to selected shapes.

You can choose *Clear* to clear the existing region assignment.

 If you try to assign a region to a shape which does not exist on the Constraint Region class, the following message appears in the console window:

E- Shape is not on the Constraint Region layer, cannot be assigned to a region.

assign route layer

The `assign route layer` command lets you assign a routing layer to pins and nets.

Related Topics

- [Assigning A Routing Layer to Pins and Nets](#)

Assign Route Layer Command: Options Panel

Access Using

- Menu path: *Route—Routing Layer Assign*

Use these controls to configure the routing parameters for pins and nets in your design.

<i>User advanced selection filtering</i>	This option lets you filter out selected nets and/or pins from the routing layer assignment. Selecting the option displays the Advanced Selection Filtering dialog box when you choose nets or pins for routing layer assignment.
<i>Reassign allowed</i>	When checked, lets you replace the existing ASSIGN_ROUTE_LAYER property on the selected pin/net with updated values.
<i>Assign power and ground nets</i>	This option lets you filter out previously assigned power and ground nets from the routing layer assignment. The default condition of this feature is On (checked).
<i>Derive from physical</i>	When checked, the selections you make get fixed to specific layers, based upon their physical connections; that is, the ending points of their via structures. Because the layer assignment is determined by the command in this mode, the assignment types and layer selections are disabled. Status messages display pin-to-layer assignments.
<i>Assignment Type:</i>	Lets you choose the type of layer assignment for your selection. Fixed is used in conjunction with the available routing layers. When you choose this assignment type, you must also choose a routing layer. Free allows the routing layout editor to route the selection(s) to the layer than best ensures a successful connection. Remove deletes the ASSIGN_ROUTE_LAYER property from the selection.
<i>Available Routing Layers:</i>	The list of available routing layers is based on your layer stack-up. You must choose one routing layer when your selection assignment type is Fixed. (To change a layer selection, you must deselect the current active layer first.)
<i>Number of selected pins:</i>	Displays the number of pins selected for assignment.


Related Topics

- Advanced Selection Filtering Dialog Box

Assigning A Routing Layer to Pins and Nets

You can assign routing layer to pins and nets by following these steps:

1. Run `assign route layer`.
The *Options* panel of the user interface is reconfigured for the command.
2. Set the Find filter to choose pins, nets, or both.
3. Set the parameters in the *Options* panel for your first selection, as described in the section above.
4. Choose the design element you want to assign the ASSIGN_ROUTE_LAYER property to.

 To choose a group of elements with the same routing parameters:

- a. Click right before making a selection.
- b. Choose *Temp Group* from the pop-up menu.
- c. Make your selections.
- d. When you have completed the selection process, click right again.
- e. Choose *Complete* from the pop-up menu. (Cancel terminates the command and returns the layout editor to an idle state.)

If you enabled the advanced selection filtering option in the *Options* panel, the Advanced Selection Filtering dialog box is displayed with a listing of your selected nets and pins.

5. Click right to display the pop-up menu.
6. Choose *Done*.
The layout editor returns to an idle state.
Verify the results by running `property edit` and clicking on the assigned pins/nets.

Related Topics

- Advanced Selection Filtering Dialog Box
- [assign route layer](#)


auto_connect

An internal Cadence engineering command.

auto_pin_renumber

The `auto_pin_renumber` command lets you easily renumber pins in a design when it becomes necessary to alter their positions in a symbol drawing (.dra).

By setting parameters in the *Pin Renumbering Options* dialog box, you can renumber pins according to your preferences. The renumbering scheme can follow any direction of your preference, and can also be *Alphabetical* or *Alphanumeric*.

 This command is available only with Allegro X PCB Editor. In Allegro X Advanced Package Designer (APD), run the command `rpn` to perform similar pin renumbering actions.

Related Topics

- [rpn](#)
- [Renumbering Pins Automatically](#)

Pin Renumbering Options Dialog Box

Access using:

- Menu path: *Layout – Renumber Pins*

<i>Numbering Scheme</i>	Drop-down menu gives you options to choose between <i>Numerical</i> , <i>Alphabetical</i> , and <i>Alphanumerical</i> numbering schemes for the selected pins.
<i>Letters to Omit</i>	This field lets you exclude specific letters from the pin names when they are automatically renumbered.
<i>Leading Zeroes</i>	By enabling this option, all pin numbers will include a zero at the beginning when renumbered automatically.
<i>Vertical Direction</i>	Define the vertical direction of increasing pin numbers. Choose whether the pin numbers will increase in the 'Top to Bottom' direction, or the 'Bottom to Top' direction.
<i>Horizontal Direction</i>	Define the vertical direction of increasing pin numbers. Choose whether the pin numbers will increase in the 'Left to Right' direction, or the 'Right to Left' direction.
<i>Letter Assignment</i>	Choose whether rows or columns get prefixed with alpha characters
Starting Pin Position	Enter your choice for where you want the pin numbering scheme to begin from. The options for the origin of pin numbers are 'Left-Top' and 'Middle Top'
Directions for Letters or Numbers	You can specify the overall direction of increasing pin numbers. By this setting, the general trend can either be increasing pin numbers in the <i>Horizontal</i> , <i>Vertical</i> , or <i>Circular</i> direction.
OK	Click <i>OK</i> to accept the changes in pin numbering and dismiss the dialog box.
Apply	Accepts the changes in pin numbering without dismissing the dialog box.
Cancel	Exits Advanced Package Router without making any changes.
Help	Displays help for Advanced Package Router.

Renumbering Pins Automatically

Perform the following steps to automatically renumber pins:

1. Access the command through the main menu (Layout – Renumber Pins), and the *Pin Renumbering Options* dialog box opens up.
2. Set your renumbering preferences such as direction of increasing pin numbers, the position of the starting pin, whether the numbering scheme should include alphabetical characters or not, and so on.
3. When all the renumbering settings are in place, click OK to apply changes and close the dialog box.
4. The footprint represented in the .dra file gets updated, and its pins are renumbered in accordance with your renumbering settings.


Related Topics


- [rpn](#)
- [auto_pin_renumber](#)

auto_route

The `auto_route` command lets you automatically route all or parts of your .brd or .mcm file through the *Automatic Router* dialog box. This command performs mainstream routing for wholly automatic routing of designs that do not require interactive routing.

You can route automatically with other commands, too: [specctra](#), [specctra_out](#), and [route_by_pick](#).

 Prior to autorouting, execute all pre-routing procedures.

 The `auto_route` command does not automatically protect existing etch/conductor when routing. If you want to do this, you must either enable the *Protect existing routes* option in the Automatic Router dialog box (Router Setup tab), or apply the FIXED property to any nets that you do not want modified during routing.

Related Topics

- [Routing the Board Automatically](#)
- [Routing the Design](#)

Automatic Router Dialog Box

The dialog box opens to the tab that co

Access Using

- Menu path (Allegro X PCB Editor, Allegro SI): *Route – PCB Router – Route Automatic*
- Menu Path (APD with the *SiP Layout* option): *Route – Router – Route Automatic*

You define parameters for automatic routing through the Automatic Router dialog box, a tabbed form. This dialog box appears when you run `auto_route`. Each tab in the Automatic Router dialog box lets you configure specific routing parameters. Common buttons along the right side of the dialog box perform the following functions:

A Commands

A Commands--auto_route

<i>Close</i>	Closes the dialog box and terminates the <code>auto_route</code> command. Parameters that you have set are saved in the database and used as the initial settings the next time you open the dialog box in the same design file.
<i>Run Checks</i>	Available only when you run the standalone program, <code>spif</code> , from your operating system prompt or executable icon. Runs a pre-route check of the current design to identify conditions that could result in routing failure.
<i>Route</i>	Shows the progress of the routing process. The router provides feedback to you during the routing process, based upon the parameters in the dialog box. You can halt the route at any time using the <i>Stop</i> button in the Progress dialog box. When routing is complete, the new data is loaded into the design's database.
<i>Undo</i>	Returns the design to its pre-routed state, otherwise this button remains inactive.
<i>Results</i>	Displays the results of the routing passes you have performed during the current command session.
<i>Script</i>	Available only when you run the standalone program, <code>spif</code> , from your operating system prompt or executable icon.

The automatic routing dialog box contains four tabs. Each tab contains specific parameter settings.

Router Setup Tab

You can choose from the following high-level strategies for routing your design.

<i>Specify routing passes</i>	Uses the parameter set in the <i>Routing Passes</i> tab.
<i>Use smart router</i>	Uses the parameters set in the <i>Smart Router</i> tab.
<i>Do file</i>	Lets you enter the name of—or browse for—the name of the <code>.do</code> file you want to execute when you route your design. When you choose this option, settings in the <i>Routing Passes</i> and <i>Smart Router</i> tabs are inactivated.
<i>Options</i>	
<i>Limit via creation</i>	Lets you route on the active layer only and avoids creating vias on other layers.
<i>Enable diagonal routing</i>	Lets you use diagonals on all selected layers during route and clean passes.
<i>Turbo stagger</i>	Use to optimize router performance and efficiency within non 45 degree staggered connector pin or via fields. Typical designs might be backpanels or motherboards where large quantities of diff pairs require routing throughout the pin fields. If unset, the router may route around the pins, resulting in longer trace runs. Setting <code>turbo_stagger</code> on may degrade overall performance, so use this option on specific nets or classes rather than globally. The default is off.
<i>Limit wraparounds</i>	Lets you route by avoiding (wherever possible) a wire that routes around a pin to get to another pin.
<i>Protect existing routes</i>	Lets you protect existing wiring such as fanouts from being ripped up during routing.
<i>Post-route smooth</i>	Available in APD only. Automatically runs the <code>custom smooth</code> command on the cline returned from the router.
<i>(Parameter button)</i>	Available in APD only. Opens the Automatic Router Parameter dialog box. You can set custom smooth parameters, among other settings.
<i>Wire grid</i>	Lets you set the X, Y wire grid spacing and the offset from where the grid originates. Values are in user-defined units.
<i>Via grid</i>	Lets you set the X, Y via grid spacing and the offset from where the grid originates. Values are in user-defined units.
<i>Routing Subclass/ Routing Direction</i>	Displays a list of etch subclasses of Etch/Conductor type on which you can perform routing. You can enable or disable each etch/conductor layer for routing. When enabled, you set the <i>Routing Direction</i> for that layer to horizontal, vertical, or both (orthogonal). If orthogonal routing is enabled, you can also choose either positive or negative diagonal routing, or both.

Protect	Causes all clines on the specified layer to be fixed so they cannot be ripped up during routing.
---------	--

The current layer setup is initialized with the data saved from the previous routing session. Routing layers that did not exist or were disabled during previous sessions use the default settings, *Horizontal* or *Vertical*.

Routing Passes Tab

Parameters in this tab are active only when Specify routing passes in the *Router Setup* tab is checked.


<i>Preroute and route</i>	Lets you specify the routing actions which you want performed, in a specific sequence. The arrow buttons to the left of each row indicate the order of routing passes. You can modify the sequence by right-clicking on a button to insert a new action or to delete the action. Each action can be enabled or disabled using the checkbox. For example, to prevent fanout during auto routing, disable the checkbox for the actions for which <i>Pass type</i> is set to <i>Fanout</i> . Disabling an action does not remove it from the list. Values set in prior sessions are the defaults for the current session.
<i>Pass Type</i>	Determines the routing action that is performed. Choose an action pass type by clicking on the arrow button.
<i>Passes</i>	Set the number of passes for each valid action (fanout, route, and clean).
<i>Start</i>	The starting pass is available only for route passes.
<i>Params</i>	Lets you set additional parameters for various action types in the Automatic Router Dialog Box. The dialog box opens to the tab that corresponds to the item you have highlighted in the <i>Preroute and route</i> section—for example, the <i>Bus Routing</i> tab.
<i>Clear</i>	Removes all <i>Preroute and route</i> entries.
<i>Post Route</i>	This section contains a set of items for controlling post-route actions. The items are active only if you selected the <i>Specify routing passes</i> strategy in the <i>Router Setup</i> tab. Checked items are run from top to bottom. Additional parameters can be set for <i>Spread wires</i> and <i>Miter corners</i> .

Smart Router Tab

The items in this tab are active only when you choose the Use smart router strategy in the *Router Setup* tab.

<i>Grid</i>	Minimum via grid defaults to <i>1</i> . Minimum wire grid defaults to <i>1</i> .
<i>Fanout</i>	Fanout if appropriate, when activated, allows fanout routing. Via sharing allows fanouts to share vias on the same net. Pin sharing allows fanouts to escape to through-pins on the same net.
<i>Generate Testpoints</i>	Off, Top, Bottom, Both indicate where you want to generate testpoints. Use grid indicates which grid to use. This field is disabled if you choose Off.
<i>Miter after route</i>	Allows mitering after routing.

Selections Tab

 This tab is not available through the `route_by_pick` command.

<i>Objects to route</i>	Lets you choose which mode to use in routing your design.
<i>Entire design</i>	Processes all nets.
<i>All selected</i>	Activates the <i>Available objects</i> list, and lets you choose specific nets and/or components for routing.
<i>All but selected</i>	Works in the opposite manner as <i>All selected</i> . Nets and/or components that you choose are not routed when you choose this option.

A Commands

A Commands--auto_route

<i>Available objects</i>	Lets you select, by way of the Object type field, nets and/or components to route or to keep from being routed. The Filter field lets you limit the objects in the list by displaying only objects of the selected type.
<i>Select all in list</i>	Moves all the listed objects to the <i>Selected Objects</i> section.
<i>Selected Objects</i>	Displays objects selected from the Available objects section, displayed as Comp or Net.
<i>Deselect all</i>	Removes all items from this list.

Routing the Board Automatically

To automatically route the boards, perform the following steps:

1. Run `auto_route` (*Route – Route Automatic*).

When you run this command, the following actions occur:

- The layout editor writes a design, rules, and forget file from the current database.
- The router is launched in the background (that is, the router interface does not appear).
- The generated design file is read into the automatic router.
- The Automatic Router dialog box opens.

2. Set the parameters in the dialog box.

3. Press *Route*.

The design routes in the background

The routing results—but not the wires themselves—are read into your design and displayed in the Automatic Router Progress dialog box.

When routing is complete, the Progress dialog box closes and the Automatic Router dialog box reappears.

4. To end the routing session and close the dialog box, click *Close*.

Related Topics

- [auto_route](#)
- [Automatic Router Dialog Box](#)

auto assign net

The `auto assign net` command lets you facilitate routing by creating and assigning routing conditions between your die, package, and plating bar. The automatic net assignment functionality uses your design constraints, package layout design, and routing layer assignments to determine routing solutions between pins, nets, and components. Solutions made by `auto assign net` are based on pin use codes.


Related Topics

- [APD: Connections](#)
- [Assigning Nets to Design](#)

Automatic Net Assignment Dialog Box

Access Using

- Menu path: *Logic – Auto Assign Net*

- Toolbar icon: 

The following table describes the controls in the dialog box.

<i>Use advanced selection filtering</i>	Lets you filter out selected nets or pins, or both, from the routing layer assignment. Selecting the option displays the Advanced Selection Filtering dialog box when you choose nets or pins for routing layer assignment.
<i>Source</i>	The type (COMPONENT CLASS) of pins from which to assign nets. Options are die (the default selection), package, or plating bar. Your choice determines the available destinations.
<i>Number selected</i>	The number of active pins assigned from your selection, advanced filtering, power/ground, net assign, and create net flag settings.
<i>Destination</i>	Destination type for the pin assignment. Options are die, package, or plating bar. Type availability is dependent on your <i>Source</i> selection.
<i>Number selected</i>	The number of active pins assigned to your selection, advanced filtering, power/ground, net assign, and create net flag settings.
<i>Assign power and ground nets</i>	<p>Routes power and ground pins along with signal pins. This option should remain unchecked if your design contains power and ground planes.</p> <p><i>Important note:</i> A power or ground pin is defined as either assigned to a power/ground net (a net with a voltage property attachment) or assigned to a dummy net but having a pin code of power or ground. Therefore, a pin with a pin use code of BIDIRECTIONAL that is attached to a power net is viewed as a power pin, and the pin use is effectively overwritten by the net assignment.</p> <p>If you do not want the pin use code considered for pins that are on dummy nets, we recommend the following procedure:</p> <ol style="list-style-type: none"> Run <i>Edit–Properties</i>. Set the pin use on all pins to be either UNSPECIFIED or BIDIRECTIONAL.
<i>Assign new connections</i>	Click this button to assign new connections for a selected group of pins.
<i>Net reassignment allowed</i>	Lets you change pre-existing assignments to allow better results for a selected group of pins.
<i>Create nets for unassigned pins</i>	<p>Allows automatic net creation for pins in the Source selection; otherwise, these pins are ignored. New nets are created in one of two formats:</p> <p><i>SIG_<SourcePin>_TO_<DestinationPin></i> when a net mate is located</p> <p><i>SIG_>SourcePin>_TO_NONE</i> when one is not found.</p>
<i>Assign one pin only for multi-pin nets</i>	Lets you assign only one destination for multi-pin source nets. The count of pins to be assigned is updated in the dialog box.
<i>Exclude pre-routed pins</i>	Removes pins in the source set that are routed to any other pin and removes pins in the destination set that are routed to any pin in the source set.
<i>Optimize existing assignments</i>	<p>Click this button to optimize the existing net assignments, ensuring that the assignment matches the logical connectivity of the pin.</p> <p>For information about how this option works, see the <i>Connections</i> chapter in the <i>Routing the Design User Guide</i>.</p>
<i>Algorithm</i>	<p>Determines the mode used to auto assign the nets. Options are:</p> <ul style="list-style-type: none"> – Nearest Match (the default for all others) – Constraint Driven <p>For information on automatically assigning nets, see the <i>Routing the Design</i> in the user guide.</p>

<i>Strategy</i>	The strategy file contains specific instructions for the auto net assignment command. You can select the default strategy file from the pulldown, click the <i>View</i> button, and review the options and instructions that are part of it.
<i>Help</i>	Displays help for this command.

Related Topics

- [Advanced Selection Filtering Dialog Box](#)
- [property edit](#)

Assigning Nets to Design

If your design is a wire bond, you must wire bond it before running the `auto assign net` command, otherwise the design is treated as a flip-chip.

Check the command console for prompts, messages, and warnings/errors during the selection phase of the procedure and after running the auto assignment.

1. Choose *Logic – Auto Assign Net*.
The Automatic Net Assignment dialog box is displayed.
2. Set the parameters of the auto net assignment as described above.
3. Choose the source pins you want to assign. You can choose pins individually, by window, or by using the right button pop-up selection *Temp Group*.
4. If you enabled the advanced selection filtering option, the Advanced Selection Filtering dialog box is displayed with a listing of your selected nets or pins.
5. Choose the destination pins you want to assign (individually, by window, or by *Temp Group*).
6. When you have completed your selection, click *Assign* in the dialog box or from the right button pop-up.
Net assignment runs, based on your settings. If any selected source pins were left unassigned, an unassigned pin list is displayed. You can also view the log file, *Auto_assign_net.log*, for results, in the current working directory.
7. To accept the net assignments, click *Ok* to exit the command. –or– Modify the parameters and/or pin selections, and click *Assign* to run a new assignment.

Related Topics

- [Advanced Selection Filtering Dialog Box](#)
- [APD: Connections](#)
- [auto assign net](#)

auto assign pinuse

The `auto assign pinuse` command lets you set your pin use codes on component pins, based on their netlist connections to other components in the same design. You can map existing pin assignments from one component to another as follows:

- Die to die
- Die to package
- Package to die
- Package to plating bar
- Plating bar to package

Related Topics

- [Setting Pin Use Codes on Component Pins](#)

Auto Pin Use Assignment Dialog Box

Access Using

- Menu path: *Logic – Auto Assign Pin Use*

The following table describes the controls in the dialog box.

<i>Use advanced selection filtering</i>	If checked, you can filter nets and pins from the set you initially selected. The layout editor displays the Advanced Selection Filtering dialog box once you choose nets or pins for pin use assignment.
<i>Source</i>	Specifies the component class type from which to copy the pin use codes. The default setting is <i>Die</i> .
<i>Destination</i>	Specifies the component class type on which to set the pin use codes. The default setting is <i>Package</i> .
Override pin use on selected instance only	Specifies that only the selected component instance should be updated. By default, this is not checked and will update all instances. Checking this enables front-to-back flow with SCM where power/ground pins are assigned to signal nets in the co-design flow.
<i>Assign power and ground pin uses</i>	<p>If checked, power and ground pin-use code assignments are propagated. Otherwise, only the signal pin use types are mapped to the destination pins.</p> <p>This provides an easy way to filter these particular pin uses without using the <i>advanced filtering</i> mechanism or <i>Temp Group</i>. By default, this option is disabled.</p>
<i>Reassignment allowed</i>	If you uncheck this box, only destination pins that are currently set to a pin use of <i>UNSPECIFIED</i> are updated. If you check this box, then all pins are updated, except for power and ground, which are determined by the <i>Assign power and ground pin uses</i> check box. By default, this option is enabled.
<i>Source/Destination Table</i>	<p>Allows you to specify the mapping for each individual pin use code from the source pins to one use on the destination pins. For example, <i>IN</i> on one Die might be <i>OUT</i> on a second die for a bus that runs between the two components in a multi-chip module. The default mappings are:</p> <ul style="list-style-type: none"> • Unspecified -> Unspecified • Power -> Power • Ground -> Ground • No Connect -> No Connect • In -> Out • Out -> In • Bidirectional -> Bidirectional • Tristate -> In • OCA -> In • OCL -> In
<i>OK</i>	Commits any changes made and exits the command.
<i>Cancel</i>	Exits the current command, undoing the last change.
<i>Help</i>	Displays the online help for the dialog box.

Related Topics

- [Advanced Selection Filtering Dialog Box](#)

Setting Pin Use Codes on Component Pins

Before you run this command, be sure that:

- At least one component in your design has the proper pin use codes set on its pins.
- You have already assigned nets between components.

To set up pin use codes o component pins, follow these steps:

1. Run the `auto assign pinuse` command.
The Auto Pin Use Assignment dialog box appears.
2. Complete the dialog box with the source and destination class, filter settings, and mappings.
If you check the *Use advanced selection filtering* box, you can filter nets and pins from the set you initially selected. The layout editor displays the Advanced Selection Filter dialog box once you choose nets or pins for pin use assignment (Step 4).
3. Set the Find filter to the appropriate element type.
4. Pick the elements, either by picking the source or destination elements.
If you set the *Source* and *Destination* fields in the Auto Pin Use Assignment dialog box to different component classes, then you need to select either the *Source* or *Destination* pins. The layout editor automatically detects the other pins from the existing net assignments. If you set the *Source* and *Destination* fields to the same component class, then you must pick both the source and destination pins for command operation as the layout editor cannot automatically detect whether the selected items are intended as the source or destination.
The layout editor takes each element from the selected set and, for source pins, finds the destination class pins connected on the same net. For destination pins, the layout editor finds the source class pin on the same net. It then uses the source pin's pin use and maps it to the specified destination pin use and sets this value on the destination pin. Thus, the mapping is by net assignment, not by matching pin numbers.
5. Repeat Step 4 until you have set all pin use codes on all components.
The `auto_assign_pinuse.log` file is generated detailing which pins are changed, their previous pin use, and their new pin use. The log shows the same messages that appear at the console window prompt.

Related Topics

- [Advanced Selection Filtering Dialog Box](#)
- [auto assign pinuse](#)

autobundle

The `autobundle` command uses the GRE route engine to automatically bundle rats in the design associated with one or more selected objects. If no objects are selected, then all rats in the design are considered for autobundling. When you bundle rats automatically, the GRE route engine determines which rats to combine into bundles. It uses autobundling criteria that you specify by setting design parameters prior to running the command. As part of the autobundling process, it may delete certain system controlled bundles (ones that it created previously). However, user-defined bundles are delete protected.

Access Using

- Menu path: *FlowPlan– Auto Bundle*
- Right Mouse Button Option: *Auto Bundle*
- Toolbar Icon: 

Bundling Rats Associated with Selected Objects Automatically

You can automatically bundle rats associated with selected objects by following these steps:

1. In IFP application mode, select one or more objects associated with the route plan (components, nets, pins, or rats).

✔ Design density may make object selection difficult. You can limit the find criteria to just one specific object type by right-clicking in the Design window, then choosing *Super filter – <object_type>* from the menu.

The selected objects highlight and also appear in the *WorldView* window.

2. With your cursor on a selected object, right-click and choose *Auto Bundle* from the menu.
The rats associated with the selected objects are automatically bundled by the GRE route engine.
3. Repeat steps 1 and 2 to automatically bundle rats associated with other objects as needed.

⚠ In cases where rats have not been bundled to your satisfaction, choose *Setup – Design Parameters – Flow Planning*, review and edit the settings of your Auto Bundle parameters, re-select the objects, and run the command again.

Automatically bundle all rats in the design:

- Choose *FlowPlan – Auto Bundle* from the menu.
All rats in the design are automatically bundled by the GRE route engine.

⚠ In cases where rats have not been bundled to your satisfaction, choose *Setup – Design Parameters – Flow Planning*, review and edit the settings of the Auto Bundle parameters, and run the command again.

auto create net

The `auto create net` command creates and assigns unique net names to component pins that you choose in your design. Each net name is made up of a prefix (optional) and a pin number (required). The component's reference designator is used as the default prefix; for example, D1_2.

Assigned pins are preserved unless you choose the option, *Reassign Pins Allowed*.

The `auto create net` command also allows you to select a specific set of pins for which new nets are created. You enable selection by choosing *Pins* in the Find Filter. The new nets are assigned to the pins in order based on (x, y) coordinates.

Related Topics

- [Creating Nets Automatically](#)

Auto Create Net Dialog Box

Access Using

- Menu path: *Logic – Auto Create Net*

<i>Netname Prefix</i>	Lets you specify a prefix to be added to the pin number for each selected pin when creating unique net names. You can select <i>Component Refides</i> , choose not to have a prefix (<i>No Prefix</i>), or choose to define a prefix (<i>User Defined</i>). If you select <i>User Defined</i> , edit the field to specify the prefix.
Net name suffix	Lets you specify a suffix from the list. The suffix can be any of: <i>Mixed-Case Pin Number</i> , <i>Mixed-Case Net Name</i> , <i>Pin Name</i> , <i>Pin Number</i> , or <i>Verilog Port Name</i> .
Use pin number if default value not available	Check this option to use the pin number as a suffix in case the suffix type specified Net name suffix is not available.
<i>Re-assign Pins allowed</i>	Re-assigns pins which are already assigned to a real net to new nets based on the settings you choose in this dialog box. If this field is disabled, only pins current on dummy nets will have nets created and assigned.
<i>Propagate assignment to connected pins and shapes</i>	Extends the reassignment to all associated conductor elements.
Create nets for floating function pins	Assigns nets to floating function pins of the selected component based on the prefix and suffix specified. Available only if a component is selected and either <i>Pin Name</i> or <i>Verilog Port Name</i> is specified in the <i>Net name suffix</i> field. If Verilog Port Name is the suffix but does not exist, Pin Name is used.

Creating Nets Automatically

You can automatically create nets and assign unique net names to component pins that you choose in your design:

1. Run the `auto create net` command.

The following message appears in the console window:

Please select components or pins for which nets are to be automatically created.

2. Do one of the following:
 - a. Choose *Comps* in the Find Filter.
 - b. Uncheck *Comps* in the Find Filter and choose *Pins*.

The Auto Create Net dialog box appears.

If you choose *Comps* in the Find Filter, a net is created for each pin of the selected component. The default net name prefix is the component's reference designator followed by an underscore, for example, U1_.

If you choose *Pins* in the Find Filter, the set of selected pins has new nets created and assigned to them. The default net name prefix is NET_. Selection by pins allows you to select a set of pins that belong to more than one component. Also, the selected pins may be a subset of the pins of one or more components.

3. Configure the controls in the Auto Create Net dialog box.
4. Click *OK* to complete the command and close the dialog box.

Related Topics

- [auto create net](#)

auto define bbvia

The `auto define bbvia` command creates multiple blind or buried vias between a range of etch/conductor layers in your design.

Use this command to save time creating many vias for different etch/conductor layer combinations. A padstack for each pair of etch/conductor layers in your design is created.

Related Topics

- [Creating Blind or Buried Vias Automatically](#)

Create bbvia Dialog Box

Access Using

- Menu path: *Setup – B/B Via Definitions – Auto Define B/B Via*

<i>Input Pad Name</i>	Indicates the padstack whose pads the editor copies when it creates vias. The button to the right of the Input Pad Name field displays a data browser containing a list of available database/library padstacks.
<i>Add Prefix</i>	Indicates a prefix that is attached to the names of vias that the editor creates. To create more than one set of vias for a design, revise information in this option.
<i>Select Start Layer</i>	Indicates the subclass that specifies the layer that starts the range of layers between which the editor creates vias.
<i>Select End Layer</i>	Indicates the subclass that specifies the layer that ends the range of layers which the editor creates vias.

Layers

<i>Use All Layers</i>	Creates bbvias between every layer pair combination.Example: Your stackup is composed of: TOP VCC INT1 INT2 GND BOTTOM Use All Layers creates vias for TOP-VCC, TOP-INT1, TOP-INT2, TOP-GND, TOP-BOTTOM; VCC-INT1, VCC-INT2, VCC-GND, VCC-BOTTOM; INT1-INT2, INT1-GND, INT-BOTTOM; INT2-GND, INT2-BOTTOM; AND GND-BOTTOM.
<i>Use Only Adjacent Layers</i>	Creates a bbvia for each pair of adjacent layers.Example: Your stackup is composed of: TOP VCC INT1 INT2 GND BOTTOM Use Only Adjacent Layers creates vias for TOP-VCC, VCC-INT1, INT1-INT2, INT2-GND, AND GND-BOTTOM.
<i>Set number of Layers</i>	This creates B/B vias between every layer pair as long as the via doesn't span more than the number of layers specified by the fill-in box. Example: a value of 2 creates the same vias as the Use only Adjacent Layers option. A value of 3 gives you every via that spans either 2 or 3 layers. It would create the following vias: TOP-VCC, TOP-INT1, VCC-INT1, VCC-INT2, INT1-INT2, INT1-GND, INT2-GND, INT2-BOTTOM, and GND-BOTTOM.
<i>Use Only External Layers</i>	Indicates that the blind via must start or stop on an external layer.
<i>Use Wire Bond Layers</i>	Indicates layers designated as wire bond layers.

<i>Use Top Pad</i>	<p>Specifies that the top pad from the template via is used as the top pad in each of the bbvias that you create.</p> <p>Example: Your template padstack has this configuration:</p> <ul style="list-style-type: none">• LAYER - PAD• TOP20 - circle• INT125 - square• INT230x40 - oblong• BOTTOM35x50 - rectangle <p>The following two padstacks created with the option on assumes these configurations:</p> <p>Padstack INT1 - INT2</p> <ul style="list-style-type: none">◦ LAYER - PAD◦ TOP - none◦ INT120 - circle◦ INT230x40 - oblong◦ BOTTOM - none <p>Padstack INT2 - BOTTOM</p> <ul style="list-style-type: none">◦ LAYER - PAD◦ TOP - none◦ INT1 - none◦ INT220 - circle◦ BOTTOM35-50 - rectangle
--------------------	---

Rule Sets

<i>Available</i>	Lists the available physical constraints defined for the design.
<i>Selected</i>	Lists the selected rule sets to which the editor adds the BBvias.

Other Options

<i>Close</i>	Closes the dialog box and ignores all input.
<i>Run</i>	Runs the Auto Define B/B Via program.

Creating Blind or Buried Vias Automatically

You can automatically create multiple blind or buried vias between a range of etch/conductor layers in your design by performing these steps by following these steps:

1. Run `auto define bbvia` to display the Create bbvia dialog box.
2. Enter a name for *Input Pad Name*.
3. If you are creating more than one set of vias for your design, click *Add Prefix* and enter the prefix in the box to the right of this option.
4. Choose the Subclass you want to start on from the *Select Start Layer* list box.
5. Choose the Subclass you want to finish on from the *Select End Layer* list box.
6. In the *Layers* section of the dialog box choose the required option.
7. Choose the Rule Sets you want to add the generated Blind/Buried vias from the *Available* list box. Click *ALL>>* to choose all the rule sets.
8. Click *Run*.

⚠ Click any rule set in the *Selected* list box to deselect it. Click *<<ALL* to deselect all rule sets.

9. If you are going to use the via on a net for routing, run `cmgr_phys` and move the newly defined padstack into the *Current via* list, located in the Physical worksheet of Constraint Manager, under the Vias column heading.

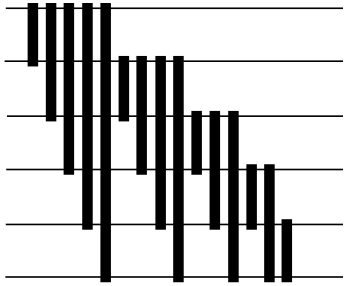
Example

The `auto bbvia` command creates in a design layout multiple BBvias between a range of layers in your design. It creates vias between class ETCH/CONDUCTOR subclasses of type Conductor or Plane.

Following figure shows the subclasses of ETCH/CONDUCTOR in an example design and the vias you create with the following `auto` command (shown in batch mode):

Figure 1.3: Example Command `bbvia via_padstack top bottom mlc_drawing_1`

ETCH Subclass Name	ETCH Subclass Type	
TOP/SURFACE	Wire Bond	
BOND_PADS	Conductor	
D1	Dielectric	
VCC	Plane	
D2	Dielectric	
SIGX	Conductor	
D3	Dielectric	
SIGY	Conductor	
D4	Dielectric	
GND	Plane	
D5	Dielectric	



The `bbvia` command uses the names of the connected ETCH/CONDUCTOR subclasses to name the vias it creates. The BBvias shown here have the following names:

```

BOND_PADS-VCC
BOND_PADS-SIGX
BOND_PADS-SIGY
BOND_PADS-GND
BOND_PADS-BOTTOM/BASE
VCC-SIGX
VCC-SIGY
VCC-GND
VCC-BOTTOM/BASE

```

SIGX-SIGY
SIGX-GND
SIGX-BOTTOM/BASE
SIGY-GND
SIGY-BOTTOM/BASE
GND-BOTTOM/BASE

⚠ The bbvia command also created the via named BOND_PADS-BOTTOM/BASE, a through via. The bbvia command creates through vias if the range of layers specified by the < startlayer > and < endlayer > arguments on the bbvia command line include all the etch/conductor layers in the design (all the ETCH/CONDUCTOR subclasses including TOP/SURFACE and BOTTOM/BASE).

In this example, the subclass TOP/SURFACE is not type conductor or plane, so the bbvia command does not create any vias that start on TOP/SURFACE subclass.

Related Topics

- [auto define bbvia](#)

awb2therm

The `awb2therm` batch command passes the power predictions of AWB smoke-alarm analysis into MAX_POWER_DISS properties on components for more accurate thermostat temperature predictions.

Syntax

```
awb2therm
```

Existing layout file name (*.brd):

axlmark

An internal Cadence engineering command.

