



P 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	14
P Commands	14
package_height	18
Package Height Command: Options Panel	19
Adding Height Specifications to Package Keepout Areas	20
Adding Height Specifications to Place Bound Rectangles	21
package integrity	22
Package Design Integrity Checks Dialog Box	23
Running Design Integrity Checks	24
packagepi	25
The Package PI Flow	26
Package PI Dialog Boxes and Options	27
Power Integrity Main Dialog Box	28
Power Integrity Dialog Box, General Tab	29
Ground Nets List Editor Dialog Box	30
Target Impedance Editor Dialog Box	31
3-D Modeling Parameters Dialog Box	32
Power Supply Editor Dialog Box	36
Cadence - Standard VRM Editor Dialog Box	37
VRM Input Inductance Calculation Dialog Box	38
Power Supplier Subckt Connection Dialog Box	39
Excitation Editor Dialog Box	40
Decoupling Capacitor Model Dialog Box	42
Decoupling Capacitor Model Editor Dialog Box	43
Power Integrity Dialog Box, Port Information Tab	44
Port Group Dialog Box	44
Die Subcircuit Connection Dialog Box	46
Power Integrity Main Dialog Box and Common Controls	47
Power Integrity Simulation Options Parameters	48
Power Integrity Decoupling Capacitor Reporter Dialog Box	49
Right Mouse Button Functionality for Package PI	50
Package PI Flow Tasks	52
Setting Up Power and Ground Net Information	53

Setting Up Component and Port Information	54
Selecting and Placing Decoupling Capacitors	55
Extracting and Analyzing Circuits	56
Reporting and Exporting Decap Information	57
package symbol wizard	58
Package Symbol Wizard Dialog Boxes	59
Creating a Package Symbol	63
pateditcur	64
pateditdb	65
Pad Edit DB Command: Options Panel	66
Editing Padstack Definitions in Your Current Design	67
Editing Padstack Instances in Your Current Design	68
pateditlib	69
Pad Edit Lib Dialog Boxes	70
Modifying a Padstack	71
pads in	72
PADS IN Dialog Box	73
Creating a PADS ASCII Database File	74
Importing a PADS Database in Interactive Mode	75
Importing a PADS Database in Batch Mode	76
PADS to Allegro Translation Options Dialog Box	77
Changing an Element Mapping	78
Editing the Database after Importing PADS Data	79
pads lib in	81
PADS Library Translator Dialog Box	82
PADS Layout Library to Allegro Translator Options Dialog Box	83
PADS to Allegro Layer Mapping	85
Importing a PADS Layout Library in Interactive Mode	86
Importing a PADS Layout Library in Batch Mode	87
padstack_editor	88
Padstack Editor	90
Padstack Editor Tabs and Panels	91
Padstack Editor: Menu Commands	103
Prerequisites to Defining New Library Padstacks	104
Padstack Definition Guidelines	105
Padstack Editor Tasks	106
Starting Padstack Editor	107

Defining Padstacks	108
Displaying Derived Padstack Names	109
Modifying Padstacks in a Library	110
Modifying Padstacks in the Design	111
Purging Unused Padstacks	112
Recording and Replaying Padstack Scripts	113
Viewing Padstack Instances	114
Creating or Modifying Padstacks from XML	115
Exporting Padstack Definition to XML file	116
pad to shape	117
Pad to Shape Command: Options Panel	118
Copying Selected Pads to a Different Layer	119
panel links	120
panel setup	121
parallel	122
Analyzing Interconnection Parallelism	125
param in	126
Import Parameter Dialog Box	127
Select Parameter File browser	128
Importing a Database Parameter File	129
param out	130
Exporting Database Parameters	131
parasitics	132
Calculating Capacitance	133
partition	134
Partition Command: Options Panel	135
Partition Notes Dialog Box	137
Creating Partitions	138
Refreshing Partitions	139
Previewing Partitions Prior to Export	140
Appending a Note	141
partlogic	142
Parts List Dialog Box	143
Component Browser	144
Adding Parts to the Parts List	146
Adding Models from SI Libraries	147
Adding New Instances of Existing Parts	148

Adding Packages from Package Libraries	149
Adding Parts from Component Libraries	150
Creating Temporary Devices	151
Deleting Parts from the Parts List	152
Modifying Parts	153
paste	154
Paste Command: Options Panel	155
Pasting Elements to Multiple Destinations in Rectangular Patterns using Post-select Mode	157
Pasting Elements to Multiple Destinations in Polar Patterns using Post-select Mode	158
Pasting Elements using Pre-select Mode	159
pause	160
Suspending Processing During Script Playing	161
pbar check	162
Plating-Bar Selection Dialog Box	163
Plating Bar Check Dialog Box	164
Reporting Plating Bar Connectivity Errors	167
pbar create	168
Create Plating Bar Dialog Box	169
Creating a Plating Bar Component	170
pbar delete	171
Delete Plating Bar Dialog Box	172
Deleting the Plating Bar in Your Design	173
pcad in	174
PCAD IN Dialog Box	175
Creating a PCAD PDIF Database File	176
Importing Information into Board/substrate Databases	177
Editing the Database	178
pdf out	179
Allegro PDF Publisher Dialog Box	180
Exporting Physical Design Information	182
phase_tune	183
Phase Tune Command: Options Panel	184
Adding Phase Bumps to Differential Pairs	185
pick	186
Pick Dialog Box	187
Highlighting Objects	188
pick_origin	189

pick_to_grid	190
pin_delay in	191
Pin Delay Import Dialog Box	192
Importing Pin Delays into a Design	193
Updating Pin Delays in a Design using Constraint Manager	194
pin_delay out	195
Pin Delay Export Dialog Box	196
Exporting Pin Delays	197
pipe	198
pkg ic overlay export	199
The Write Package Overlay for IC Dialog Box	200
Using Package Overlay Export	201
Viewing Package Overlay in EDI	202
place area design	203
Running the Place Area Design Command	204
place area list	205
place area room	206
Placing Components in a Room	207
place area window	208
Defining a Window and Placing Components	209
place auto bottomgrid	210
Generating a Grid on the Bottom Subclass of Your Design	211
place auto topgrid	212
Generating a Grid on the Top Subclass of Your Design	213
place execute	214
Performing Automatic Placement in Automatic Mode	215
place interactive	216
Performing Automatic Placement in Interactive Mode	217
place manual	218
Place Manual Command: Options Panel	219
Placement Dialog Box	220
Placing Components	223
Placing Symbols	224
Placing Alternate Symbols	225
Placing Modules	226
Rotating During Placement	227
Mirroring During Placement	228

placement	229
Executing Automatic Placement in a Unix Window	230
placementedit	231
Accessing Command Help for Right Mouse Button Options within an Application Mode	232
place param	233
Automatic Placement Dialog Box	234
Setting Parameters for and Running Automatic Placement in Automatic Mode	237
Examples of the Place Param Command	238
place replicate apply	240
Place Replicate Unmatched Component Interface Dialog Box	241
Place Replicate Component Swap Interface Dialog Box	242
Place Replicate Layer Mapping Dialog Box	243
Running the Circuit Replication Flow Command	244
Resolving Unmatched Components between the Seed and Targeted Replicated Circuits	245
place replicate create	247
Placement Replicate Create Dialog Box	248
Creating a Place Replicate Module File (.mdd) from a Seed Circuit	249
place replicate update	250
Modifying Replicated Circuits	251
place set bottomgrid	252
Generating Grids on a BOARD GEOMETRY/SUBSTRATE GEOMETRY Class, PLACE_GRID_BOTTOM Subclass	253
place set topgrid	253
Generating Grids on a BOARD GEOMETRY/SUBSTRATE GEOMETRY Class, PLACE_GRID_TOP Subclass	254
place structure	254
Place Structure Dialog Box	255
Placing a Structure	256
plane generator	257
Power Delivery Solution Generator Dialog Box	258
plc chglyr	259
plctxt	260
Running the Plctxt Command	261
plctxt in	262
Import Placement Dialog Box	263
Importing Component Placement Information	264
plctxt out	265

Export Placement Dialog Box	266
Exporting Component Placement Information	267
plot	268
Print Dialog Box	269
Plotting Prerequisites on a Unix Workstation	270
Plotting Your Design on Unix	271
Plotting Your Design on Windows	272
plot preview	273
Previewing an Active Design	273
plot setup	273
Plot Setup Dialog Box	274
Setting Parameters for Plotting a Design	276
plotwint	277
polar	278
Entering Selection Points Specified by Polar Coordinates	279
pop add	280
pop addrect	281
pop align	282
Using the Pop Align Command	283
pop alt module	284
pop alt symbol	285
Substituting an Alternate Package Symbol	286
pop apply	287
pop autovoid	288
pop bbdrl	289
Adding a Blind/Buried Via to the Design	290
pop change layer	291
pop controltrace	292
pop cut	293
Using the Pop Cut Command	294
pop datum	295
Creating a New Datum Point	296
pop delete all voids	297
pop delete vertex	298
Deleting a Vertex	299
pop dimension value	300
pop drc window	301

pop drill	302
Adding a Through Via Hole to the Design	303
pop dyn_option_select	304
pop finish	305
pop fix	306
pop flip	307
Using the Pop Flip Command	308
pop in	309
pop jinput	310
pop lasso	311
pop mirror	312
Mirroring a Selected Component	313
pop mirror geometry	314
pop move	315
pop neck	316
pop net list	317
pop net name	318
pop new target	319
pop no target	320
pop out	321
pop path	322
pop pickgroup	323
pop prmed	324
pop properties	325
pop readfile	326
pop refdes list	327
pop refdes name	328
pop region list	329
pop route_from_target	330
pop routespace	331
pop scale	332
pop shape change type	333
pop shape copy	334
pop shape copy layers	335
pop shape defer dyn fill	336
Deferring Dynamic Copper Fill for a Single Dynamic Shape	337
pop shape delete vertex	338

pop shape edit boundary	339
Changing a Shape or Void Outline	340
pop shape move	341
Moving an existing shape or void	342
pop shape param	343
pop shape raise priority	344
Assigning a Higher Voiding Priority to a Dynamic Shape	345
pop shape report	346
pop shape select	347
pop shape update	348
pop show	349
pop skip	350
pop swap	351
Swapping the Active Layer for an Alternate Layer	352
pop swap pin assignment	353
Swapping an Active Pin for the Alternate Pin	354
pop unfix	355
pop unfixall	356
Deleting the FIXED Property from All Objects	357
pop window	358
Using the Pop Window Command	359
pop wirebond add	360
pop wirebond add guide	361
pop wirebond add jumper	362
pop wirebond add route	363
pop wirebond adjust drc	364
pop wirebond auto bond	365
pop wirebond best fit path	366
pop wirebond center	367
pop wirebond chg group	368
pop wirebond copy finger left	369
pop wirebond copy finger right	370
pop wirebond copy guide	371
pop wirebond create ring	372
pop wirebond create unwired	373
pop wirebond cut shape	374
pop wirebond default	375

pop wirebond delete	376
pop wirebond delete wires	377
pop wirebond del route	378
pop wirebond edit guide	379
pop wirebond heal shape	380
pop wirebond merge fingers	381
pop wirebond move	382
pop wirebond move guide	383
pop wirebond move wires	384
pop wirebond net assign	385
pop wirebond pause	386
pop wirebond popul guide	387
pop wirebond recon	388
pop wirebond settings	389
pop wirebond space	390
pop wirebond split fingers	391
pop wirebond split guide	392
pop wirebond swap	393
pop wirebond swap wires	394
power integrity	395
Preparing your Design to Satisfy Power Integrity Setup Requirements	396
preferences	398
prepopup	399
pring wizard	400
Power/Ground Ring Wizard Dialog Boxes	401
Ring Count and Placement Dialog Box	402
First Ring/Die Flag Definition Dialog Box	403
Next Ring Definition Dialog Box	405
Result Verification Dialog Box	406
Creating a Set of Split Rings Around a Complex Wire Bond Die	407
print	409
Printing a File from the Command Line	410
printform	411
prmed	412
Design Parameter Editor Dialog Box	413
Changing Display Parameters	414
Changing Design Parameters	415

Changing Text Parameters	416
Changing Shapes Parameters	417
Changing Flow Planning Parameters	418
Changing Route Parameters	419
Changing Manufacturing Applications Parameters	420
prompt	421
property edit	422
Edit Property Dialog Box	423
Assigning a Property to a Design Element	424
Editing a Property on a Design Element	425
Deleting a Property from a Design Element	426
Attaching a Property to a Room Boundary	427
Creating an Inherited Property	428
pulse_commit	429
pulse_copy_as	430
pulse_designs	431
pulse_getrevision	432
pulse_lock	433
pulse_reuse_module_mgr	434
Pulse Module Manager Dialog Box	435
pulse_share	435
pulse_unlock	437
pulse_version_control	438
purge unplaced comps	440
Purge Unplaced Components Dialog Box	441
Removing Unplaced Components from the Design Database	442
purge unused nets	443
Purge Unused Nets Dialog Box	444
Removing Unused Nets from the Design	445
push connectivity	446
Push Connectivity: Options Panel	447
Advanced Selection Filtering Dialog Box	448
pwd	449

P Commands

package_height	package integrity	packagepi
package symbol wizard	paddeditcur	paddeditdb
paddeditlib	pads in	pads lib in
padstack_editor	Padstack Editor	pad to shape
panel links	panel setup	parallel
param in	param out	parasitics
partition	partlogic	paste
pause	pbar check	pbar create
pbar delete	pcad in	pdf out
phase_tune	pick	pick_origin
pick_to_grid	pin_delay in	pin_delay out
pipe	pkg ic overlay export	place area design
place area list	place area room	place area window
place auto bottomgrid	place auto topgrid	place execute
place interactive	place manual	placement
placementedit	place param	place replicate apply
place replicate create	place replicate update	place set bottomgrid
place set topgrid	place structure	plane generator
plc chglyr	plctxt	plctxt in
plctxt out	plot	plot preview
plot setup	plotwint	polar
pop add	pop addrect	pop align
pop alt module	pop alt symbol	pop apply
pop autovoid	pop bbdrl	pop change layer
pop controltrace	pop cut	pop datum

P Commands

P Commands

pop delete all voids	pop delete vertex	pop dimension value
pop drc window	pop drill	pop dyn_option_select
pop finish	pop fix	pop flip
pop in	pop jinput	pop lasso
pop mirror	pop mirror geometry	pop move
pop neck	pop net list	pop net name
pop new target	pop no target	pop out
pop path	pop pickgroup	pop prmed
pop properties	pop readfile	pop refdes list
pop refdes name	pop region list	pop route_from_target
pop rout esp ace	pop scale	pop shape change type
pop shape copy	pop shape copy layers	pop shape defer dyn fill
pop shape delete vertex	pop shape edit boundary	pop shape move
pop shape param	pop shape raise priority	pop shape report
pop shape select	pop shape update	pop show
pop skip	pop swap	pop swap pin assignment
pop unfix	pop unfixall	pop window
pop wirebond add	pop wirebond add guide	pop wirebond add jumper
pop wirebond add route	pop wirebond adjust drc	pop wirebond auto bond
pop wirebond best fit path	pop wirebond center	pop wirebond chg group
pop wirebond copy finger left	pop wirebond copy finger right	pop wirebond copy guide
pop wirebond create ring	pop wirebond create unwired	pop wirebond cut shape
pop wirebond default	pop wirebond delete	pop wirebond delete wires
pop wirebond del route	pop wirebond edit guide	pop wirebond heal shape
pop wirebond merge fingers	pop wirebond move	pop wirebond move guide
pop wirebond move wires	pop wirebond net assign	pop wirebond pause
pop wirebond popul guide	pop wirebond recon	pop wirebond settings
pop wirebond space	pop wirebond split fingers	pop wirebond split guide

P Commands

P Commands

pop wirebond swap	pop wirebond swap wires	power integrity
preferences	prepopup	pring wizard
print	printfm	prmed
prompt	property edit	pulse_commit
pulse_copy_as	pulse_designs	pulse_getrevision
pulse_lock	pulse_reuse_module_mgr	pulse_share
pulse_unlock	pulse_version_control	purge unplaced comps
purge unused nets	push connectivity	pwd

package_height

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

Related Topics

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

Package Height Command: Options Panel

Access Using

- *Menu Path: Setup – Areas – Package Height (Allegro X PCB Editor)*

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

Related Topics

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

Adding Height Specifications to Package Keepout Areas

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

 You can also add package heights using the `property edit` command.

Related Topics

- [package_height](#)
- [prmed](#)

Adding Height Specifications to Place Bound Rectangles

1. Run `package_height`.
2. In the Options section of the mini-status form, make sure the active class and subclass is set to PACKAGE GEOMETRY/PLACE_BOUND_TOP or PLACE_BOUND_BOTTOM.
3. Choose a package/part shape, or choose *Add Rectangle* or *Add Shape* from the right-button pop-up to add one.
The *Options* panel displays for the package height command. If you chose an existing shape, the current minimum and maximum heights are displayed, as well as a graphic view that illustrates how the editor treats the height values. (Note that the graphic is not dynamic; that is, it does not change according to the specific height values that you enter.)
4. If the place boundary element is new, enter the minimum and maximum height information in the text fields of the *Options* panel.
The heights that you enter are expressed as a range in user units (mils, inches, or millimeters) in the Design tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters*. This defines the vertical dimension of the shape area from minimum value to maximum value. The minimum must be greater than or equal to 0. The maximum can be anything up to infinity (the default).
5. When you have attached the restrictions you want, click *Done*.

Related Topics

- [package_height](#)
- [Package Height Command: Options Panel](#)

package integrity

The `package integrity` command lets you run integrity checks to ensure that your database is configured correctly. With this command, you can diagnose your own design problems. You can also customize the tool to look for problems or deviations from your company requirements that the standard design tool may not consider errors.

Individual checks are product-dependent. Any checks that are not related to the currently running tool will be removed from the user interface when the command is run. For example, any checks related to a feature not in the specified family of products, are not included.

Related Topics

- [Running Design Integrity Checks](#)

Package Design Integrity Checks Dialog Box

The Package Design Integrity Checks dialog box includes all the categories and checks currently registered for the active running product. You can enable all the categories and checks or enable only the ones that you want to run. If you select the name of a rule or category but do not check the boxes, the right panel displays the detailed description, but does not change its active state. The default setting for all the rules is off.

The right panel provides a detailed description of the category and check, including the nature of the problem, reason for the problem, probable cause or origin of the problem, preventative measures, and the solution if this command will not fix it.

Access Using

- Menu Path: *Tools – Package Design Integrity*

All On	Click this button to have the package design integrity tool include all defined integrity checks in all categories for the test.
All Off	Click this button to have the package design integrity tool exclude all the defined integrity checks in all categories for the test.
General	This category includes checks that may impact a wider range of commands within your package substrate layout tool, and therefore do not lend themselves to inclusion in one of the more specific categories. Correcting issues reported in this category may improve results obtained with multiple commands in the system. If you are running checks from one or more categories, it is recommended that you also consider running these supplementary scans.
Manufacturing	This category includes checks for issues most likely to cause problems when you generate manufacturing output from the tools, such as generating DXF, stream, or artwork data. Correcting issues reported in this category helps minimize cycles with your package foundry. You can correct minor inconsistencies in the database prior to the creation of final mask data.
Signal Integrity	This category includes checks for items that are most likely to cause problems when performing signal integrity analysis using either a 2D or 3D field solver. Correcting issues reported in this category helps minimize problems running the solvers, while also ensuring that the tools give the most accurate results possible.
Wire Bonding	This category includes checks for issues related to adding and manipulating the wire bonds in your design. Correcting issues reported in this category helps to ensure that you do not encounter unforeseen problems when trying to adjust your wire bonds later in your design flow. In many cases, these checks may also help if you are experiencing problems with the display of bond wires in the Cadence 3D Design Viewer or extracting them for the 3D field solver.
Reporting Options	
Log file	Check this box to write a log file of problems found. The default setting is on. This file is written to the location specified by the <code>ADS_SDLOG</code> user preferences variable.
Write descriptions to log (verbose mode)	Check this box to have the log file include the description of the checks as well as the issues found. The default setting is on.
Fix errors automatically (where possible)	Check this box to specify that the tool fix errors in the design when possible. The ability to self-correct the errors varies for each rule. The default setting is on.
External DRC markers	Check this box to specify that the tool add external DRC markers at the locations where violations appear. These markers use the name of the defined rule as their description element, therefore making them easier to locate. The default setting is off.
Clear DRCs	Click this button to remove any DRC violations found by this command during this or previous runs of the tool.
OK	Click this button to close the dialog box. If you do not click <i>Apply</i> before closing the dialog box, the checks do not occur.
Cancel	Click this button to close the dialog box without running any checks. The tool rolls back any changes that you have made.
Close	Click this button to exit the tool, leaving the design in its present state.
Apply	Click this button to perform the configured checks, but leave the command active and the dialog box displayed.
Help	Click this button to display context-sensitive help for this command.

Running Design Integrity Checks

Use this command prior to performing certain milestone events, such as performing a full 3D SI model extraction on your database. You can also run the `package integrity` command with the related checks when you have problems getting a specific command to work correctly and before you call Cadence Technical Support.

The tool helps you overcome problems in your design by correcting them or advising you how to correct them.

1. Run the `package integrity` command.
2. Check boxes for the rules that you want to run.
3. Click *Apply* once you configure the appropriate checks and the reporting options.

A progress meter appears.

If any errors are found in the database, which the tool cannot automatically correct, it is recommended that you correct those errors and re-run this command prior to continuing with your design flow. This ensures that you do not encounter further problems or compound the difficulties later on.

Related Topics

- [package integrity](#)

packagepi

The `packagepi` command lets you analyze power delivery to power nets in your packaging and silicon-in-package designs. The correct analytical results are derived from the appropriate configuration of field solver and 3D modeling parameters, target impedance, port group assignments, group and excitation settings, placement of decoupling capacitors (decaps), and extraction of equivalent circuits. The Package PI functionality lets you configure all required and optional settings by way of graphical user interfaces, then analyze the results.

Related Topics

- [The Package PI Flow](#)
- [Package PI Flow Tasks](#)

The Package PI Flow

The following list of commands represents the recommended flow for performing power integrity analysis for your package designs, including required preliminary steps.

Preliminary configuration

- [Assign power and ground net voltages](#)
- [Ball and bump parameters](#)
- [Set up bond wire profiles](#)
- [Set up field solver parameters](#)

Power/Ground Net Configuration

- [Select a power net for analysis](#)
- [Set return ground nets](#)
- [Set target impedance](#)
- [Set power supply for board side](#)

Die and Port Configuration

- [Import/export port information](#)
- [Add excitation sources/current profile](#)
- [Set on-die equivalent sub-circuit](#)
- [Group die pads and BGA pins](#)
- [Select port type and reference port](#)

Decap Selection and Placement

- [Add/Import decoupling capacitors](#)
- [Place virtual decaps](#)
- [Place optional probe port](#)

Extraction and Analysis

- [Select voltage ripple/impedance analysis](#)
- [Extract/reuse equivalent circuit](#)
- [Check analysis results in SigWave](#)

Report and Export

- [Report and export decaps in design](#)

Related Topics

- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Package PI Dialog Boxes and Options

The Package PI functionality supports numerous dialog boxes. The purpose of each dialog and its controls are described in this section.

- [Power Integrity Dialog Box, General Tab](#)
- [Ground Nets List Editor Dialog Box](#)
- [Target Impedance Editor Dialog Box](#)
- [3-D Modeling Parameters Dialog Box](#)
- [Power Supply Editor Dialog Box](#)
- [Cadence - Standard VRM Editor Dialog Box](#)
- [VRM Input Inductance Calculation Dialog Box](#)
- [Power Supplier Subckt Connection Dialog Box](#)
- [Excitation Editor Dialog Box](#)
- [Decoupling Capacitor Model Dialog Box](#)
- [Decoupling Capacitor Model Editor Dialog Box](#)
- [Power Integrity Dialog Box, Port Information Tab](#)
- [Port Group Dialog Box](#)
- [Die Subcircuit Connection Dialog Box](#)
- [Power Integrity Main Dialog Box and Common Controls](#)
- [Power Integrity Simulation Options Parameters](#)
- [Power Integrity Decoupling Capacitor Reporter Dialog Box](#)

Related Topics

- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Power Integrity Main Dialog Box

The Power Integrity main dialog opens when you invoke the Package PI command from the menu bar or from the command console. The dialog box contains two tabs:

- The [Power Integrity Dialog Box, Port Information Tab](#) contains controls for selecting and/or configuring power and ground nets, setting target impedance, ball and bump information, board-level VRMs, and sink current excitation sources.
- The [Power Integrity Dialog Box, Port Information Tab](#) lets you set up/configure die and port information.

Access Using

- Menu Path: *Analyze – Power Integrity*

Power Integrity Dialog Box, General Tab

Use the controls in this tab to specify a power or ground net for analysis.

<i>Net Information section</i>	
<i>Power Net</i>	The power net selected for analysis.
<i>Ground Nets</i>	The return ground nets associated with the selected power net, used to identify which nets decoupling capacitors connect to when placed in the design. By default, all ground nets associated with the selected power net are selected.
<i>Ground Nets Edit button.</i>	Opens the Ground Nets List Editor Dialog Box for adding or deleting ground nets.
<i>Voltage</i>	The voltage of the selected power net.
<i>Ripple Tolerance</i>	The maximum voltage drop or spike that the design can tolerate (expressed as a percentage of voltage between 1 and 5%). You can set the value from the field drop-down menu or in the Target Impedance Editor.
<i>Max. Delta Current</i>	The maximum amount of current that the design can tolerate in a three-phase circuit. You can set the value in the field or in the Target Impedance Editor.
<i>Target Impedance</i>	The powerplane target impedance for all frequencies, based on the bias voltage, ripple tolerance, and worst-case dynamic current.
<i>Target Impedance Edit button</i>	Opens the Target Impedance Editor Dialog Box .
<i>Parameters button</i>	Opens the 3-D Modeling Parameters dialog box that lets you set the parameters used by the 3D field solver.
<i>Power Supply button</i>	Opens the Power Supply Editor Dialog Box that lets you add VRM models and board-level power supply sub circuits.
<i>Excitation Source button</i>	Opens the Excitation Editor Dialog Box that lets you manage your excitation sources.
<i>Decoupling Capacitors section</i>	
<i>Decap worksheet</i>	<p>The decoupling capacitor worksheet lists all the decaps in the design and provides the following information:</p> <ul style="list-style-type: none"> • <i>Model Name</i>: Name of the decap model • <i>Capacitance</i>: Capacitance value as specified in the signal model • <i>ESR</i>: Equivalent series resistance as specified in the signal model • <i>ESL</i>: Equivalent series inductance as specified in the signal model • <i>Frequency</i>: Computed resonant frequency (in Hertz for ESpice and S-Parameter models) • <i>Virtual/Instant</i>: Number of virtual and/or actual decaps placed in the design. The text indicator displays in red for placed virtual decaps. When you select a row in the worksheet and right-click, the context pop-up menu lets you choose the following operations: <ul style="list-style-type: none"> • <i>Place Virtual</i> lets you place a virtual component. When you select this operation, the <i>Options</i> panel in the product UI is reconfigured to display decap information; specifically: <ul style="list-style-type: none"> ◦ RefDes will display the reference designator of the decap. Instant decap reference designators will begin with C, as in Cxxx; virtual decap designators will begin with VC. ◦ Decap Model will display the decap model name. ◦ Frequency will display the resonant frequency. <p>If you change the placement settings, those changes will apply to all subsequent decaps that you place, during the active session. The product default settings will be re-established during the next placement session.</p> <p>Package PI will not take into account any physical or spacing violations as a result of decap placement.</p> <ul style="list-style-type: none"> • <i>Component List</i> opens a text window containing information about the decoupling capacitor. • <i>Graph Response</i> displays the waveform of the impedance for the decoupling capacitor in SigWave.

Add Model button	Opens a window that lets you add a decap to the worksheet. This can be a new model or one selected from the Model Browser.
Edit Model button	Opens the Decoupling Capacitor Model Editor Dialog Box that lets you view or modify the parameters of the selected decap.
Delete Model button	Deletes the selected decoupling capacitor.
Assign Model button	Opens the Signal Model Assignment dialog box that lets you assign a model to the selected decap. (For information on this dialog box, see signal_model .)

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Ground Nets List Editor Dialog Box

Use this dialog box to configure your ground nets. Package PI automatically lists all the ground nets associated with your selected power net. To override this default condition, the Ground Nets List Editor lets you add or delete associated ground nets.

Power Net	The power net selected for analysis
Available Gnd List	Lists all the ground nets available to be selected. Double-click the net name to select it or highlight the name and click the -> button.
Selected Gnd List	Lists the selected ground nets. Double-click the net name to deselect it or highlight the name and click the <-> button.
Reset	Restores the default list of ground nets for the selected power net.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Target Impedance Editor Dialog Box

Use this dialog box to set a target impedance level. The power distribution system should provide the components with a required/suitable impedance path across the entire frequency spectrum to the power and ground supplies in order for the system to work properly. The required impedance—called target impedance—is expressed as the following equation:

$$Z_{target} = \frac{(power_supply_voltage) \times (allowed_ripple)}{current}$$

The Target Impedance Editor lets you modify these factors to define target impedance at specified frequency ranges. (For a detailed account of target impedance, see the Best Practices document, *Impedance Distribution in Power Delivery Systems*.)

Power Net	The power net selected for analysis
Voltage	The voltage of the power net
Ripple Tolerance	The maximum voltage drop or spike that the design can tolerate, expressed as a percentage of voltage.
Max. Delta Current	The maximum amount of current that the power net will potentially suffer in a worst-case scenario.
Corner Frequency	The frequency up to which the target impedance is constant.
Slope (dB/Decade)	The ramp-up of corner impedance above corner frequency
Lower Frequency	The low frequency range of the analysis.
Upper Frequency	The high frequency range of the analysis.
Target Impedance	The target impedance value based on the supply voltage, ripple tolerance, and maximum delta current parameters.
Plot	Displays a wave form of the target impedance profile in SigWave.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

3-D Modeling Parameters Dialog Box

Use this dialog box to set up the modeling parameters for your 3D field solver. This dialog is composed of five tabs:

- [General Tab](#)
- [Bump Tab](#)
- [Ball Tab](#)
- [External Ground Tab](#)
- [SI Ignore Layers Tab](#)

General Tab

Option	Description	
<i>Solder Ball Location</i>	<p>Specifies how the package is oriented relative to the PCB.</p> <p>⚠ If the die and ball pins are on the same side of the package substrate, the 3D Field Solver determines that it is located on the bottom.</p> <p>Options are:</p>	
	<i>Auto Detect</i>	Specifies that the package be determined by the software.
		⚠ Currently, Auto Detect covers most design cases, but not all. In cases where the software cannot derive the package position from the design, you are prompted to explicitly set the type yourself by selecting one of the two remaining options.
	<i>Bottom</i>	Specifies a package position on the bottom of the PCB.
	<i>Top</i>	Specifies a package position on the top of the PCB.
<i>Design Unit</i>	Shows the current unit used for the design.	
<i>Frequency</i>	Specifies the frequency at which the narrowband circuit model is generated.	
<i>Number of coupling nets</i>	<p>Specifies the number of coupling nets to model. A value of <i>1</i> indicates a single line. A value of <i>2</i> indicates a single neighbor net. Note:</p> <ul style="list-style-type: none"> ◦ For crosstalk or comprehensive types of analyses, the geometry window / min coupled length / min neighbor capacitance is ignored if the 3D Field Solver is used. The number of neighbor nets is set by <i>Number of coupling nets</i>. ◦ However, for differential nets (reflection analysis), both the inverted and non-inverted nets are always extracted regardless of <i>Number of coupling nets</i>, 	
<i>Minimum via diameter</i>	<p>Specifies a minimum via diameter. The default is either <i>2mil</i> or <i>50um</i> (depending upon the drawing unit type).</p> <p>⚠ If the via diameter is less than the default, 90% of the diameter for the smallest pad in the design is used.</p>	
<i>Ignore void diameter</i>	The maximum boundary extension in both the x and y dimensions of a void to ignore. Set this to an appropriate value to have small voids ignored to speed up the simulation. The default is <i>0</i> , meaning no voids are ignored.	
<i>RL mesh density (resistance/inductance)</i>	Specifies the density (cell size) of the RL mesh used for finite element package modeling and defines how the RL accuracy should asymptotically converge. Click the arrow to choose <i>Coarse</i> , <i>Fine</i> , or <i>Finest</i> from the drop-down menu. The value you choose determines the following 3D modeling performance / accuracy trade-offs:	
	<i>Coarse</i>	Fastest, but least accurate.
	<i>Fine</i>	Default, with good accuracy.
	<i>Finest</i>	Slowest, but most accurate.
<i>CG mesh density (capacitance/conductance)</i>	Specifies the density of the CG mesh used for finite element package modeling. Click the arrow to choose <i>Coarse</i> , <i>Fine</i> , or <i>Finest</i> from the drop-down menu. The value you choose determines 3D modeling performance / accuracy trade-offs.	

P Commands

P Commands--packagepi

<i>CG planar boundary box</i>	Specifies the size of the boundary box in the x and y dimensions used to enclose the package area that includes all chosen nets to be modeled. Click the arrow to choose <i>Small</i> , <i>Medium</i> , or <i>Large</i> . The default is <i>Medium</i> .
<i>CG z-directional boundary box</i>	Specifies the size of the boundary box in the z dimension used to enclose the package area that includes all chosen nets to be modeled. Click the arrow to choose <i>Small</i> , <i>Medium</i> , or <i>Large</i> . The default is <i>Medium</i> .
<i>Enable Multiport</i>	When YES (the default selection), specifies that multi-pin circuits will generate an equivalent lumped circuit representing all ports in the circuit in the post-processed model. When NO, a multiport solution is generated for all ports; however, the post-processed model will be collapsed into a two node (input node and output node) lumped model. Note: The multi-port option is intended to model signal nets with 3 ports. While you can use this feature to help in the extraction of models of power or ground nets, it requires significant computing time and resources due to the typically large number of pin ports in power/ground nets. We recommend you exercise caution in using this feature when modeling power/ground nets.
<i>Controlled sources in model</i>	When YES (the default selection), instructs the field solver to produce a multi-port DC port models with controlled sources. (This control is inactive if the <i>Multiport</i> option is not enabled.) Note: Set this option to NO if you are using Cadence's VoltageStorm power-grid verification to analyze IR-drop.
<i>Number of subcircuit segments</i>	Indicates the number of distributed subcircuits generated for a narrowband model transmission line. The default value is 5. Higher numbers of segments will yield more accurate models, but may increase computation time.
<i>Start frequency</i>	Enter a value to specify the start frequency. The default is <i>0Hz</i> . This is also the recommended start frequency
<i>Number of frequency points</i>	Enter the number of points in the frequency range. The default is 2048 points. This value should be a power of 2, with a frequency step of about 10MHz.
<i>Frequency sweep scale</i>	Select a frequency sweeping type from the pulldown menu. The default is <i>Linear</i> .
<i>Reference impedance</i>	Enter the impedance for the generated output. The default is 50ohm.

Bump Tab

Option	Description
<i>Die Component</i>	Allows you to select from the drop-down menu one die to be modeled from the correct design. You then enter the parameters for the selected die in the succeeding fields. Note: You can leave this field blank if you do not want to specify an individual die.
<i>Dmax</i>	Specifies the maximum diameter for the solder bumps. (If the value of <i>Dmax</i> is set to 0, solder bumps will not be modeled). Using a value that is too large risks solder bump overlap. A value of zero for <i>Dmax</i> indicates that the bumps are not modeled.
<i>D1</i>	Specifies the bottom diameter of the solder bumps. This value must be less than or equal to <i>Dmax</i> .
<i>D2</i>	Specifies the top diameter of the solder bumps. This value must be less than or equal to <i>Dmax</i> .
<i>HT</i>	Specifies the height of the bumps.
<i>Conductivity</i>	Specifies the conductivity for the solder bumps.

Bump information that you configure for individual dies in the *Bump* tab controls are stored in the *.abf* file in your current working directory (If you do not define a specific die component, bump data defaults to the *.agf* file.) The following shows the syntax for an *.abf* file and an example:

```
die_comp_name Dmax D1 D2 HT conductivity direction_flag
```

```
P1    45    40    40    40    6897    dieup
```

A value of zero for *Dmax* or *HT* indicates that the bumps are not modeled.

Ball Tab

Option	Description
<i>Dmax</i>	Specifies the maximum diameter for the solder balls.
<i>D1</i>	Specifies the bottom diameter of the solder balls. ⚠ This value must be less than or equal to <i>Dmax</i> .
<i>D2</i>	Specifies the top diameter of the solder balls. ⚠ This value must be less than or equal to <i>Dmax</i> .
<i>HT</i>	Specifies the height of the balls.
<i>Conductivity</i>	Specifies the conductivity for the solder balls.
<i>WireBond Profile</i>	Opens the Wire Bond Profile Editor. (For information on using the editor, see wirebond settings .)

⚠ A value of zero for *Dmax* or *HT* indicates that the balls are not modeled.

External Ground Tab

Option	Description
<i>Include PCB plane (Ground #1)</i>	Specifies whether a PCB plane is to be used. When enabled (checked), click on either <i>Ground</i> or <i>Float</i> to choose the plane type
<i>h1</i>	Specifies the distance between the package bottom layer and the PCB ground plane layer (Ground #1).
<i>Under fill dielectric constant</i>	Specifies the dielectric constant of the under fill. ⚠ The under fill is the material between the bottom layer of the package and the PCB top layer (not including the solder ball material).
<i>PCB dielectric constant</i>	Specifies the dielectric constant of the PCB.
<i>Include top plane (Ground #2)</i>	Specifies whether to use a top plane. When enabled (checked), click on either <i>Ground</i> or <i>Float</i> to choose the plane type.
<i>h2</i>	Specifies the distance between the package top layer and the top plane layer (Ground #2). ⚠ This value can be zero.
<i>Top fill dielectric constant</i>	Specifies the constant of the top fill dielectric.

SI Ignore Layers Tab

All Layers	Lists of all layers in the design. Click on a layer to display it in the <i>SI Ignore Layers</i> list and have it excluded from the 3D model.
SI Ignore Layers	<p>Lists layers in the design that are currently excluded from 3D modeling.</p> <p>⚠ The number of remaining conductor layers must be the same as the number of actual metal layers in the package.</p> <p>Click on a layer to remove it from the list and have it included in the 3D model.</p>
All	Simultaneously removes all layers in the <i>SI Ignore Layers</i> list and includes all layers in the 3D model.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Power Supply Editor Dialog Box

Use this dialog box to add a new or existing VRM models to your design as well as a power supply sub-circuit that acts as a conduit for current from the VRM to the package by way of the package pins. The Package PI functionality includes delivery of power from voltage regulator modules (VRMs) to board-level designs as part of a complete co-design flow. The VRM that you select is not added to your package design; rather, the Package PI functionality automatically connects it to the package.

Cadence provides a default VRM, `pg_vrm`, that you can access from the Model Browser, or you can load user-defined VRMs or create new ones.

Power Net	The selected power net
<i>VRM Model</i>	Opens the Add Model dialog that lets you add one VRM from the Model Browser or lets you create a new one. You can also unselect the VRM.
<i>Supply SubCkt</i>	Lets you add one power supplier sub-circuit, consisting of a sub-circuit definition file and a pin-mapping file. If pin-mapping information for the sub-circuit is not available through the file browser, you must create the pin-to-port mapping by way of the Power Supplier Subckt Connection Dialog Box form.
<i>Text Edit</i>	Opens a text editor for modifying the model content.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Cadence - Standard VRM Editor Dialog Box

Use this dialog box to set the parameters of your VRMs. To open the VRM Editor, double-click the VRM model name in the Power Supply Editor.

<i>Slew Inductance</i>	Rate at which the VRM can react to changes in current. Example: a VRM may take 15 microseconds to slew the current from 8- to 20-amps.
<i>Calculate</i>	Displays the VRM Input Inductance Calculation Dialog Box dialog box.
<i>Flat Resistance</i>	Equivalent series resistance of the capacitor that is associated with the VRM.
<i>Output Inductance</i>	Cable or pin parasitic inductance of connecting the VRM to the board.
<i>Output Resistance</i>	Sense resistance between the VRM and the load.
<i>Text Edit</i>	Opens a text editor for modifying the model content.
<i>Graph Response</i>	Displays the impedance curve of the VRM in SigWave.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

VRM Input Inductance Calculation Dialog Box

Use this dialog box to calculate input inductance for the 4-element SPICE model.

<i>Voltage</i>	The potential difference between the plane pairs.
<i>Ripple Tolerance</i>	The maximum voltage droop or spike noise that the design can tolerate (expressed as a percentage of voltage).
<i>Ramp Time</i>	The maximum time for the VRM to react to a transient current.
<i>Ramp Current</i>	The maximum transient current. This value is usually the same as the delta current value that you specify in the Power Integrity Design and Analysis dialog box.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Power Supplier Subckt Connection Dialog Box

Use this dialog box to map board-level pins to ports for your selected supply sub-circuit. If your sub-circuit already contains mapping information, you can modify it by double-clicking on the sub-circuit name in the Power Supply Editor to bring up this dialog.

Power Net	The selected power net
SubCkt Name	The selected sub-circuit
Text Edit	Opens a text editor for modifying the model content.
Supplier Component	Optional information specifying where the sub-circuit comes from
Package <--> Board Mapping grid	Lets you map package pins to board pins and subcircuit ports.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Excitation Editor Dialog Box

Use this dialog box to introduce an excitation source into your design that will act as the sink current in a co-design flow of board to package to die. The supported formats for excitation sources are Gaussian, Pulse, and Current Profile. The excitation sources you introduce to your design are configured and managed through the Excitation Editor.

Excitation Tab

Use this tab to add and configure Gaussian and pulse excitation sources.

Excitation Name	The user-defined name of the excitation signal that you add to your design. The name must start with an alphabetic character.
<i>Add</i>	Opens a pop-up window for entering an excitation source.
<i>Remove</i>	Deletes the highlighted excitation source from the editor.
<i>Import</i>	Lets you import an existing excitation source, formatted as a text file, from an external location.
<i>Export</i>	Lets you export a selected excitation source as a text file to an external location.
The following fields lets you configure Gaussian function configuration values	
<i>Amplitude</i>	The pulse amplitude
<i>Delay Time</i>	The pulse delay time
<i>Width</i>	The pulse width
The following fields lets you configure pulse configuration values	
<i>Init Value</i>	The initial pulse value
<i>Final Value</i>	The pulse amplitude value
<i>Delay Time</i>	The pulse delay time
<i>Rise Time</i>	The pulse rise time
<i>Fall Time</i>	The pulse fall time
<i>Width</i>	The pulse width
<i>Period Time</i>	The pulse period time

Current Profile Tab

Use this tab to add one or more existing current profiles as excitation sources.

Excitation Name	The user-defined name of the current profile excitation source that you add to your design.
<i>Pin Num</i>	Displays the total pin number of the highlighted current profile.
<i>Max. Current</i>	The maximum current value for the highlighted pin number.
<i>Add</i>	Opens a file browser for choosing an existing current profile, formatted as a text file. (Typically derived from the IC manufacturer or from Cadence's VoltageStorm extractor.) When you select a file, a pop-up window is displayed in which you must enter a new name for the excitation source.
<i>Remove</i>	Deletes the highlighted excitation source from the editor.
<i>Import</i>	Lets you import an existing excitation source, formatted as a text file, from an external location.
<i>Export</i>	Lets you export a selected excitation source as a text file to an external location.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Decoupling Capacitor Model Dialog Box

Use this dialog box to add a decap to the worksheet. This can be a new model or one selected from the Model Browser.

Model Name	The name of the selected decap model
Browse	Opens the Model Browser from where you can select a decap model to add to the worksheet.
Model Edit	Opens the Decoupling Capacitor Model Editor from where you can modify the parameters of the decap model.
OK	Adds the model to the worksheet and closes the form.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Decoupling Capacitor Model Editor Dialog Box

Use this dialog box to view or modify the parameters of the selected decoupling capacitor.

Any changes that you make in the Decoupling Capacitor Editor are saved to the component model file that is associated with the selected capacitor. All capacitors that reference the device model, in turn, are refreshed with the new values.

<i>Model Name</i>	Name of the device file that is associated with the capacitor.
<i>Capacitance</i>	Default nominal capacitance value as specified in the signal model.
<i>Intrinsic inductance (ESL)</i>	Estimated Intrinsic inductance of a surface mounted capacitor computed from its height parameters. Intrinsic inductance computations do not account for the capacitor's mounting characteristics. This field can be left blank or populated with an estimated value if the capacitance for your model is not available, as might be the case with an n-terminal or S-Parameter model.
<i>Estimate button</i>	Opens the Intrinsic Inductance Estimator form. This form lets you estimate the intrinsic inductance of a capacitor based on the thickness (height) value that you supply. The intrinsic inductance estimation does not consider the capacitor's mounting characteristics.
<i>Intrinsic Resistance (ESR)</i>	Signal model's impedance value at resonance. This field can be left blank or populated with an estimated value if the capacitance for your model is not available, as might be the case with an n-terminal or S-Parameter model.
<i>Resonant frequency</i>	Computed resonant frequency based on capacitance and intrinsic inductance values. This field can be left blank or populated with an estimated value if the capacitance for your model is not available, as might be the case with an n-terminal or S-Parameter model. For S-Parameter models, the resonant frequency point can be extracted at the lowest value for S11 or S22
<i>Text Edit button</i>	Opens a text editor for modifying the model content.
<i>Graph Response button</i>	Displays a single capacitor response wave in SigWave.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Power Integrity Dialog Box, Port Information Tab

Use the controls in this tab to configure on-die and port information.

<i>Filter</i>	Lets you tailor the list of components in the selected net using alphabetic and wildcard combinations; for example, A*.
<i>Select All</i> <i>button</i>	Highlights all the components in the selected net.
<i>Port Group</i> <i>button.</i>	Opens the Port Group Dialog Box that lets you group source pins and sink pins in a multiport net.
<i>Die Information section</i>	
<i>Current Profile</i>	Lets you select from the drop-down list a current profile, a text file that defines a switching current profile. You would typically derive this file from Cadence's VoltageStorm extractor or from the IC manufacturer. The profiles in the list are determined by those added in the Excitation Editor. When you add a profile, the sum of the current is distributed equally to all the pins in the selected component.
<i>None</i>	Lets you configure the on-die information without entering subcircuit capacitance and resistance values.
<i>Capacitance</i>	Lets you enter an on-die capacitance value.
<i>Resistance</i>	Lets you enter an on-die resistance value. You can use both this value and the capacitance value as a simplified die circuit
<i>SubCkt</i>	Lets you add/import an Espice die subcircuit by way of the Die Subcircuit Connection dialog box. You can import the subcircuit model either through a device library or an external source.
<i>Edit button</i>	Opens the Die Subcircuit Connection Dialog Box .
<i>Ports section</i>	
<i>Pin Name</i>	Lists all the pins in the selected power net.
<i>Port Type</i>	<p>Lists the port types for each pin.</p> <ul style="list-style-type: none"> If you have not selected a current profile, all port types default to <i>Sink</i> for die components and can be edited. If a BGA is the selected component, all port types default to <i>Source</i> and cannot be edited. If a current profile is selected, it is listed as the excitation source for all pins and cannot be edited.
<i>Excitation</i>	<p>Lists the excitation source or a constant current value for each pin.</p> <ul style="list-style-type: none"> If your component selection is a die and you have selected a current profile, the excitation source for all pins defaults to the profile and cannot be edited. If you do not select a current profile and you select a port type of <i>Sink</i> for any pin, the <i>Excitation</i> field becomes active and lets you select a Gaussian or pulse excitation source from the drop-down or lets you enter a constant current value. If your component selection is a BGA, no excitation source or current profile is selectable.
<i>Group</i>	Lets you group the sink and source pins on your packages and dies as a multi-port net to improve extraction and simulation performance. You group sink and source pins in the Port Group dialog box.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Port Group Dialog Box

Use this dialog box to group source pins and sink pins in a multiport net. Port grouping gives you the capability of setting up a partition-based extraction by enclosing ports of source and sink pins in a specified portion of your design. This eliminates the limitation of having to extract the entire design with each pin identified.

⚠ For purposes of simulation, you must designate at least one source pin and one sink pin to each net. Other than that requirement, you can designate any pin (port) as either source or sink. You can also include source and sink pins in a single group. In every instance, Open pins are ignored during simulation.

<i>Selection Area</i>	
<i>Net Select</i>	Displays a list of the nets in your design. Selection of a net (either from the list in the dialog box or from the design on the canvas) displays information associated with all the pins in that net.
<i>Comp Select</i>	Displays a list of the components in your design. Selection of a component (either from the list in the dialog box or from the design on the canvas) displays information associated with all the pins in that net.
<i>Port Group Assignment Area</i>	
	⚠ This section of the dialog box lets you select pins for grouping and assigning. But you can also select individual pins directly from the design canvas. A selection that you make from the canvas has the effect of moving that pin name from the list box it is presently in, to the other list box. Selecting the pin again moves the pin name back to the other list.
<i>Group/Type Filter</i>	Determines which pins of a specified type are displayed in the list box. Choices are * (all), reference, open, sink, source, and unspecified. If you do not select one group of pins as a reference group, the highest group of pins acts as the reference group.
<i>New Group/Type</i>	Determines the group type the selected pins will be converted to.
<i>Clear</i>	Deletes the display of pins in the list boxes

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Die Subcircuit Connection Dialog Box

Use this dialog box to model the die on a selected power net from an ESpice sub-circuit model on an IC. The IC sub-circuit is typically extracted by the IC manufacturer or from the Cadence VoltageStorm extractor.

The IC sub-circuit consists of a sub-circuit definition file and a pin mapping file (similar to that of a power supply sub-circuit). After loading the two separate files, you need to specify the mapping from the die pins to chip pads and ports. Package PI will attempt to do this automatically, based on the pin number, pad and port names.

Equivalent Subcircuit for Die	Displays the selected die name
<i>Power Net</i>	Displays the selected power net
<i>SubCkt</i>	The IC sub-circuit name from the selected die. The drop-down list will contain all the sub-circuits in your design.
<i>Browse</i>	Lets you import an external sub-circuit definition file and pin-mapping file into your design.
<i>Text Edit</i>	Opens a text editor to edit the selected sub-circuit.
<i>Die <-> Chip Mapping grid</i>	Lets you match the die pins to the IC pads and ports

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Power Integrity Main Dialog Box and Common Controls

The controls common to both tabs in the Power Integrity main dialog box are:

Analyze button	Opens the Power Integrity Simulation Options Parameters form. Invoke this form to perform voltage ripple analysis in the time domain or impedance analysis in the frequency domain, or to extract a DML model.
Report button	Opens the Power Integrity Decoupling Capacitor Reporter Dialog Box .
Hide button	Hides the Power Integrity dialog box from view. To restore it to view, right-click in the design canvas and select <i>Show Form</i> from the pop-up menu.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Power Integrity Simulation Options Parameters

Use this dialog box to set the type of simulation you want to run, along with their associated parameters. These options are typically set after you have selected and configured your field solver, added your optional VRMs, added your optional board/die sub-circuit model, grouped your source and sink pins, and placed all required virtual decoupling capacitors into your design. When you have done this and clicked **OK** in the dialog, Package PI will call the selected field solver to extract the equivalent sub-circuit for the selected power and reference nets, establish a full circuit for Tlsm in the time or frequency domain, and display the analysis results in SigWave.

Simulation Type section	Lets you select a simulation type <ul style="list-style-type: none">• Impedance analysis in the frequency domain• Voltage ripple analysis in the time domain• DML model extraction
Simulation Parameters section	Parameters in this section are dependant on the type of simulation you have selected <ul style="list-style-type: none">• Impedance analysis – Upper and lower frequency range of the simulation and the type of sweep scale, either <i>Linear</i> or <i>Log</i>• Voltage Ripple analysis – Duration and resolution time
DML Model Option section	In instances when you have run a previous simulation, you can choose to not re-extract the DML model and, instead, reuse the previously extracted model from its stored location. Note that the location used to store the DML model must be readable. You can also modify the previously extracted model as a different model type, either <i>DML Wideband</i> , <i>Narrow Band</i> , or <i>S-Parameter</i> . Note that if you select the Narrow Band option, the center frequency is the frequency value set in the 3-D Modeling Parameters dialog box.
OK	Initiates the selected field solver and analysis run.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Power Integrity Decoupling Capacitor Reporter Dialog Box

Use this dialog box to display a report of the decoupling capacitor characteristics in your design on either a selected power net or on all of them. Highlighted decaps in the report can be saved to a text file. Note that in the case of virtual decaps, the right button pop-up menu contains a *VDecap Report* item, that will bring up a text window displaying similar characteristics of the decoupling capacitors in your design.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Right Mouse Button Functionality for Package PI

Package PI supports numerous context-specific features that you access by way of the right mouse button. The items that populate the right button pop-up menus are dependant on which areas of the user interface your cursor is focused on when you right-click.

Decoupling Capacitor Worksheet in the Package PI Main Form – General Tab

When you right-click on a highlighted decap in the Decoupling Capacitor worksheet, the following functions are displayed in the pop-up menu:

PlaceVirtual	<ul style="list-style-type: none"> • Lets you place a virtual component. When you select this operation, the Options panel in the product UI is reconfigured to display decap information; specifically: <ul style="list-style-type: none"> ◦ RefDes will display the reference designator of the decap. Instant decap reference designators will begin with C, as in Cxxx; virtual decap designators will begin with VC. ◦ Decap Model will display the decap model name. ◦ Frequency will display the resonant frequency. <p>If you change the placement settings, those changes will apply to all subsequent decaps that you place during the active session. The product default settings will be re-established during the next placement session.</p> <p>Package PI will not take into account any physical or spacing violations as a result of decap placement.</p>
Component List	Opens a text window containing information about the decoupling capacitor.
Graph Response	Displays the waveform of the impedance for the decoupling capacitor in SigWave.

Ports Information Worksheet in the Package PI Main Form – Port Information Tab

When you right-click on the column headers in the Ports worksheet, the following functions are displayed in the pop-up menu:

All	<p><i>Sort Ascending/Sort Descending:</i> Lists the column entries in ascending or descending alphabetical order <i>Find:</i> Opens a window in which you can type in a pin name which will be highlighted in the Pin Name column. <i>Import/Export:</i> Opens a file browser that lets you import or export port information in text file format from or to an external source. The files are formatted as in the following example:</p> <pre>;; Title: PDN port information ;; Date: Jun 19 14:37:06 2008 ;; Design: D:/Cadence/packagepi/multiport.mcm ;; Power Nets: VDD (("Excitation" "pl" Width 1e-009 DelayTime 1e-009 Amplitude 1.0 Type Gaussian) ("Pin" PortGroup "3" SinkCurrent nil PortType "SINK" PinName "DIE.201"))</pre>
Port Type	<i>Change All:</i> In this column, opens the Source window that lets you select a port type that will propagate to all editable port types for the selected component.
Excitation	<i>Change All:</i> In this column, opens the Port Sink Current window that lets you select an available excitation source from those previously created in the Excitation Editor. The source you select will propagate to all Sink port types for the selected component.
Group	<i>Group:</i> Opens the Port Group Dialog Box for changing one or more pins to different groups.

Design Canvas

When you are running the `packagepi` command and position your cursor in the design canvas, the following functions are displayed in the right-button pop-up menu:

Done	Terminates the Package PI session and returns the tool to an idle state.
<i>Show Form</i>	Redisplays the Power Integrity main dialog box, if hidden.
<i>Show Element</i>	Displays information related to the selected element.
<i>Show Result</i>	Opens a file browser from where you can select the simulation (.sim) file containing the analysis results on your design.
<i>VDcap Clear All</i>	Deletes all the virtual decaps on all power nets.
<i>VDcap Report</i>	Displays the decoupling capacitor report which contains all the virtual decaps in the design and their locations.
<i>VPort Find</i>	Opens a window in which you can enter a virtual probe port. Package PI zooms in to the selected items and highlights them.
<i>VPort Place</i>	Lets you place a virtual probe port at a location on the power/ground net. A probe port is a specific one-pin component that lets you output a voltage ripple and impedance value from that point during circuit extraction.
<i>VPort Clear All</i>	Deletes all the virtual probe ports on all power nets.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Package PI Flow Tasks

The tasks associated with Package PI are similar to those presented in [The Package PI Flow](#). Each procedure assumes you have previously performed the following tasks:

- a. Identified the power and ground net voltages and assigned the voltage property to the nets.
- b. Set up the ball and bump parameters.
- c. Set up the bond wire profiles.
- d. Set the selected field solver parameters.

The following procedures are performed during a `packagepi` active session.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Setting Up Power and Ground Net Information

1. Select a power net from the drop-down list from the Power Net drop-down in the [Package PI main dialog box](#).
2. Select or add ground nets in preparation for placing decoupling capacitors, using the [Ground Nets List Editor Dialog Box](#).
3. Set your target impedance: voltage, tolerance maximum delta current, etc. using the [Target Impedance Editor Dialog Box](#).
4. Select a VRM model and board power supplier sub circuit, using the [Power Supply Editor Dialog Box](#). (This step is optional.)

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Selecting and Placing Decoupling Capacitors](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Setting Up Component and Port Information

Package PI automatically identifies all die and package components connected to the power net you select for analysis.

1. If port information (pin names, group, type, excitation source) was previously configured and saved to an external file, you can import that information to Package PI, using the *Export/Import* option of the Power Integrity Main dialog box, otherwise proceed to step 2.
2. Set a die current profile from the Current Profile drop-down in the Package PI main dialog box or an excitation source from the Excitation Editor Dialog Box. (This step is optional.)
3. Set on-die capacitance and resistance in the Die Information section of the Package PI main dialog box, or a die sub-circuit, from the Die Subcircuit Connection Dialog Box. (This step is optional.)
4. Set port group information for die and package components, using the Port Group Dialog Box. (This is an optional step.)
5. Set port type and excitation sources in the Ports worksheet of the Power Integrity Main dialog box.
6. Export the port information to an external file using the *Export/Import* option of the Power Integrity Main dialog box. (This would typically be done when you have analyzed the power net to your satisfaction.)

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Extracting and Analyzing Circuits](#)
- [Reporting and Exporting Decap Information](#)

Selecting and Placing Decoupling Capacitors

Perform the following tasks iteratively until you achieve satisfactory simulation results.

1. Select the appropriate decoupling capacitor (decap) DML model, using the *Add Model* button in the Package PI main dialog box. If necessary, edit the ESpice model file with the Decoupling Capacitor Model Editor Dialog Box.
2. Place the virtual decoupling capacitors into the design. Select the decap model in the Decoupling Capacitor worksheet in the Package PI main dialog box and select *Place Virtual* from the right-button pop-up menu. The virtual decaps that you place during the `packagepi` session are deleted from the design when you terminate the command but are restored when a new session of `packagepi` is started.

When you have determined the optimum location for reducing the impedance profile, you can save and print off the location information in the virtual decoupling capacitor report (*VDecap Report* right-button pop-up option) and clear the virtual decaps (*VDecap Clear All*) from your design. When you quit the Package PI session, you can then place actual decaps in the locations specified in the decap report.

 If the model that you are placing is to be used *only* by a virtual decap, the model will not be saved in the design database. Thus, if the design is sent to others, you will need to transfer both the design and the `devices.dml` file associated with it.

3. Add probe ports to output the voltage ripple and impedance value from specific points in the design by selecting the *Vport Place option* in the right-button pop-up menu on the design canvas. The virtual probe ports that you place during the `packagepi` session are deleted from the design when you terminate the command but are restored when a new session of `packagepi` is started.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Reporting and Exporting Decap Information](#)

Extracting and Analyzing Circuits

These steps describe how to perform voltage ripple analysis in the time domain or impedance analysis in the frequency domain, then review the results in SigWave. Perform the tasks iteratively until you achieve satisfactory simulation results.

1. Select the appropriate analysis type and associated options in [Power Integrity Simulation Options Parameters](#) by clicking the *Analyze* button in the Package PI main dialog box.
2. Check the results of the analysis in SigWave when Package PI automatically launches the tool when analysis is complete. Points on the package that exceed your pre-set targets for impedance value or voltage ripple can be addressed by modifying the placement/setup of the virtual decaps.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)

Reporting and Exporting Decap Information

1. To view a detailed report of the decoupling capacitors used in your design, click the *Report* button in the Package PI main dialog box to open the [Power Integrity Decoupling Capacitor Reporter Dialog Box](#). You can export the highlighted file, in ASCII format, to an external source by clicking the *Save* button in the Reporter.

Related Topics

- [package integrity](#)
- [Package PI Dialog Boxes and Options](#)
- [Right Mouse Button Functionality for Package PI](#)
- [Package PI Flow Tasks](#)
- [Setting Up Power and Ground Net Information](#)
- [Setting Up Component and Port Information](#)
- [Selecting and Placing Decoupling Capacitors](#)

package symbol wizard

The `package symbol wizard` command is available when you open a new design (`new`). You can also access the symbol wizard by selecting the drawing type *Package symbol (wizard)* in the New Drawing dialog box.

The Package Symbol Wizard provides an easy way for you to create a package symbol. The wizard is designed either to help beginning users create a simple package symbol, or for experienced designers who want a quick way to create a base package symbol that they can modify into a more complex symbol.

Following the instructions on the dialog boxes, you enter data and see visual representations of the package symbol. When you have finished creating your package symbol, the corresponding symbol drawing is automatically opened in the Symbol editor. If you cancel out of the wizard during the process, a new drawing, with the name you specified in the New Drawing dialog box, opens in the Symbol editor.

At all times, you can move backward and forward through the wizard screens to modify your package.

During the Package Symbol Wizard process, you are expected to specify padstacks to use with your symbol. When you choose a padstack, the data browser lists all the padstacks available in your current PADSTACK path (PADPATH). If you want to create your own padstacks for use with the package symbol, you can use the Padstack Designer to do this before using the Package Symbol Wizard. You can also use the Padstack Designer to determine padstacks already in the Padstack Library. For details, see the *Defining and Developing Libraries* user guide in your documentation set.

Related Topics

- [Creating a Package Symbol](#)

Package Symbol Wizard Dialog Boxes

The package symbol wizard is composed of a series of dialog boxes:

Package Selection

When you choose the package symbol, the thumbnail graphic shows you what your symbol looks like. You choose the package symbol from the following list of basic symbol types:

- DIP (Dual Sided Through Hole Package)

The DIP Setup page for dual sided through-hole package type allows you to customize the following package attributes:

- Number of pins in the package (N). The default is 14 pins.
- Lead pitch (e). This is an editable field with a pop-up containing some industry-standard values. The default is 100 mils.
- Spacing between the two rows (e1). This is an editable field with a pop-up containing some industry-standard values. The default is 300 mils.
- Package width (E). The default is 250 mils.
- Package length (D). the default is 800 mils.

- SOIC (Small Outline Package)

The SOIC Setup page for small outline package type allows you to customize the following package attributes:

- Number of pins in the package (N). The default is 14 pins.
- Lead pitch (e). This is an editable field with a pop-up containing some industry-standard values. The default is 50 mils.
- Spacing between the two rows (e1). This is an editable field with a pop-up containing some industry-standard values. The default is 225 mils.
- Package width (E). The default is 175 mils.
- Package length (D). the default is 395 mils.

- PLCC/QFP(Quad Flat Package)

The first PLCC/QFP Setup page for PLCC and QFP package type allows you to customize the following package attributes:

- Number of pins in the package along the vertical boundary (Nv).
- Number of pins in the package along the horizontal boundary (Nh).
- The location of pin number 1 may be chosen from:
 - The top left corner of the package
 - The middle of the top row of pins (the default selection)
- Lead pitch (e). This is an editable field with a pop-up containing some industry-standard values. The default is 31.50 mils (0.8 mm).
- Spacing between the two rows (e1). This is an editable field with a pop-up containing some industry-standard values. The default is 225 mils.

The second PLCC/QFP Setup page for PLCC and QFP package type allows you to customize the following package attributes:

- Spacing between the two terminal columns (e1). This is an editable field. The default is 425 mils (10.8 mm).
- Spacing between the two terminal rows (e2). This is an editable field. The default is 425 mils (10.8 mm).
- Package width (E). The default is 394 mils (10 mm).
- Package length (D). the default is 394 mils (10 mm).

- PGA/BGA (Pin Grid Array Package)

The first PGA/BGA Setup page for PGA package type allows you to customize the following package attributes:

- Maximum matrix size:
 - Pin count along vertical boundary (MD)
 - Pin count along horizontal boundary (ME)
- Pin arrangement
- Full matrix perimeter
- Matrix arrangement:
 - Outer rows
 - Core rows
- Staggered pin (on or off)

The second PGA/BGA Setup page for PGA package type allows you to customize the following package attributes:

- Pin numbering scheme:
- Number right – letter down
- Number right – letter up
- Number left – letter down
- Number left – letter up
- Letter right – number down
- Letter right – number up
- Letter left – number down
- Letter left – number up
- Number horizontally
- Number vertically
- Letter horizontally
- Letter vertically
- JEDEC standard
- Pad numbered with zeros

The third PGA/BGA Setup page for PGA package type allows you to customize the following package attributes:

- Lead pitch along vertical columns (ev). This is an editable field with a pop-up containing some industry-standard values. The default is 100 mils.
- Lead pitch along horizontal rows (eh). This is an editable field. This is an editable field with a pop-up containing some industry-standard values. The default is 100 mils.
- Package width (E). The default is 1575 mils.
- Package length (D). the default is 1575 mils.

- TH (Through Hole) Discrete

The THD setup page for Through Hole Discrete Package type allows you to customize following package attributes:

- Spacing between the two terminal pins (e1). This is an editable field with a pop-up containing some industry-standard values. The default is 1000 mils.
- Package Width (E). This is an editable field. The default is 750 mils.
- Package Length (D). This is an editable field. The default is 300 mils.

- SMD (Surface Mount Discrete)

The SMD page for SMD Discrete Package type allows you to customize following package attributes:

- Spacing between the two terminal pins (e1). This is an editable field with a pop-up containing industry-standard values. The default is 195 mils.
- Package Width (E). This is an editable field. The default is 145 mils.
- Package Length (D). This is an editable field. The default is 145 mils.

- SIP (Single Inline Package)

The SIP Setup page for Single Inline Package type allows you to customize following package attributes:

- Number of pins in the package (N). The default is 8 pins.
- Lead Pitch (e). This is an editable field with a pop-up containing some industry-standard values. The default is 100 mils.
- Package Width (E). This is an editable field. The default is 110 mils.
- Package Length (D). This is an editable field. The default is 750 mils.

- ZIP (Zig-zag Inline Package)

The ZIP Setup page for Zig-zag Inline Package type allows you to customize following package attributes:

- Number of pins in the package (N). The default is 14 pins.
- Lead Pitch (e). This is an editable field with a pop-up containing some industry standard values. The default is 50 mils.
- Spacing between the two terminal rows (e1). This is an editable field with a pop-up containing some industry standard values. The default is 100 mils.

- Package Width (E). This is an editable field. The default is 110 mils.
- Package Length (D). This is an editable field. The default is 750 mils.

When you have chosen the package type, you must complete the Template and General Setup screens. After you complete this information the wizard presents specific screens that request information about your package choice. Default values are inserted in fields where relevant, and industry standard choices are available where applicable.

You choose the package symbol you want to create and then customize it. When you are satisfied with your choices, the package symbol is created with the following objects:

<i>Pins</i>	Pins are added to the symbol at locations depending on the basic symbol you choose and the parameters you set for the symbol. You define a unique padstack for pin 1 and another padstack for all the other pins.
<i>Component Outline</i>	The Component Outline is a rectangle drawn to your specifications. This outline is added to class PACKAGE_GEOMETRY, subclasses ASSEMBLY_TOP and SILKSCREEN_TOP.
<i>Labels</i>	The Reference Designator you specify is added at default locations on class REF DES, subclass SILKSCREEN_TOP. It is also added to class REF DES, subclass ASSEMBLY_TOP.
<i>Constraint Areas</i>	The package keepout area is automatically generated from the component outline and pin information you provide.

Template Selection

A template is a .dra file that contains basic information for the package symbol. Cadence supplies a default template, or you can create your own template that contains basic information on colors, text sizes, or documentation for your symbol. Cadence recommends that if you are a new user of the editor, you should use the default template.

 The location of the default template is

```
<cdsroot>/share pcb pcb.lib symbols/template/sym_template.dra
```

The default file is replaced each time you apply a hotfix or QSR. If you edit the file, you should store a copy outside the Cadence hierarchy.

Customized Templates

You should use a customized template only after you are familiar with the Package Symbol wizard. When you use a customized template you can add preexisting data or common data to your drawing. This data can include your units and accuracy, company logo, specific colors, text block definitions, and any other layer-specific data that you want to use.

Make sure your customized template is available before starting the Package Symbol wizard. The browser lets you navigate to your custom template.

 There is no path variable available to find customized template .dra files. You must enter the path in the query field.

- When you prepare your custom template note the following guidelines:

The custom template should not contain any data that might interfere with the symbol generation process. In particular, it should have no data on the following classes:

PIN ETCH

PACKAGE GEOMETRY PACKAGE KEEPIN

PACKAGE KEEPOUT REF DES

ROUTE KEEPIN ROUTE KEEPOUT

VIA CLASS VIA KEEPOUT

- The symbol wizard uses Text Block 3 for the Reference Designator Prefix text size.
If you put more than one reference designator prefix in a template, the package symbol wizard only recognizes the first reference designator. This does not preclude you from changing the reference designator prefix in the wizard.
- The symbol wizard uses Text Block 1 for Pin Number text size.

General Setup

The General Setup screen lets you set the units for the package symbol, the accuracy for the drawing, and the Reference Designator Prefix.

At all times you can choose options from drop-down list boxes.

 The default reference designator is U*. If you forget to append an asterisk (*) to your reference designator, the wizard appends one for you.

Padstacks

Generally you specify one padstack for Pin 1 and another padstack for the rest of the pins in your package.

 When you choose the padstack for Pin 1, it automatically appears in the list box on the screen for the other padstacks. Use the Browse button to choose a different padstack for the other pins.

Compiled Symbol Generation

The Compiled Symbol Generation screen lets you specify whether or not you want to create the compiled symbol. In order to use a symbol within a board it must be compiled. The compile step checks the symbol for errors and produces a compact file.

If you choose to compile the symbol, the wizard creates both a `.dra` file and a `.psm` file in your current working directory.

If you do not compile the symbol, only the symbol drawing (`.dra`) is created. You can open the symbol drawing and continue editing it in the editor. When you finish editing the symbol use the *File – Create Symbol* command to create the compiled symbol (`.psm`) file.

Creating a Package Symbol

1. Run `new`.
The New Drawing dialog box appears
2. Enter a name for the drawing in the Drawing Name box.
If you enter a name of a drawing that already exists, a message appears warning you that the current file exists and its contents are overwritten.
3. Choose *Package symbol (Wizard)* in the Drawing Type box.
4. Click *OK*
The Package Symbol Wizard appears.
5. Follow the Package Symbol Wizard instructions to create your package symbol.

Related Topics

- [package symbol wizard](#)

pateditcur

An internal Cadence engineering command.

padeditdb

The `padeditdb` command lets you choose a padstack definition or instances from your design and modify it. To modify padstacks on the disk, use [padeditlib](#).

You set the padstack parameters in the *Options* panel of the Control Panel or click on the padstack that you want to modify in your design. You then open the Padstack Designer to edit the padstack.

If you edit padstacks in your design, you also can purge any unused padstacks. However, if you replace a padstack on pins or vias that are part of a symbol, the change occurs to the symbol instance and not to the symbol definition. To purge the padstack from the board, you must update the symbol's `.dra` file, recompile it to create the `.psm` file, and then refresh the symbol in the database to remove the symbol definition's reference(s) to the padstack.

If you edit a padstack definition, you can update your design with the changed definition from within the Padstack Designer.

You cannot modify layout padstacks within the Symbol Editor because they never propagate to a design. They are not saved to the `.psm` file.

For more details, see the *Defining and Developing Libraries* user guide in your documentation set.

Related Topics

- [Editing Padstack Definitions in Your Current Design](#)
- [Editing Padstack Instances in Your Current Design](#)
- [Purging Unused Padstacks](#)

Pad Edit DB Command: Options Panel

Access Using

- *Menu Path: Tools – Padstack – Modify Design Padstack*

<i>Instance</i>	Lets you modify a padstack instance.
<i>Definition</i>	Lets you modify a padstack definition.
<i>List box</i>	Displays the padstacks available for modification.
<i>Name</i>	Lets you enter the name of a padstack instance or definition.
<i>Symbol</i>	Lets you specify the symbol you want to attach the padstack to. If you place an asterisk (*) in this field, the <i>Refdes</i> field automatically
<i>Pin</i>	Lets you specify the pin you want to attach the padstack to. If you specify an asterisk (*) in this field, then all pins of the specified symbol (or reference designator) get the new padstack. This field is not active when you choose a padstack definition.
<i>Refdes</i>	Lets you specify the reference designator you want to attach the padstack to. This field is not active when you choose a padstack definition.
<i>New name</i>	Lets you assign a new name to the chosen padstack. This field is not active when you choose a padstack definition.
<i>Edit</i>	Opens the Padstack Editor which lets you create and edit padstacks and save them to your design, to a library, or to both at once.
<i>Reset</i>	Clears all the fields in the <i>Options</i> panel.
<i>Purge</i>	Lets you purge padstacks (all or derived) from your design.

Related Topics

- [Editing Padstack Instances in Your Current Design](#)
- [Purging Unused Padstacks](#)

Editing Padstack Definitions in Your Current Design

Perform the following steps to edit padstack definitions in your design:

1. Run `pateditdb`.
The Options panel is configured for the command.
2. Enable *Definition*.
3. Choose the padstack definition that you want to edit from the list of available padstacks or select a pin referencing the padstack in your design.
The padstack information appears in the *Name* field.
4. Click *Edit*. (You can also choose *Edit* by right-clicking in the design window and selecting it from a pop-up menu.)
The Padstack Designer opens with the padstack definition loaded.
5. Specify the padstack parameters and layers as appropriate. For details, see [Padstack Editor](#).
6. As needed, check the padstack definition by running the Padstack Summary report. Choose *Reports – Padstack Summary* from the Padstack Designer.
7. Choose *Update To Design* from the toolbar or the *File* menu to update all of the padstacks in the design that have the current name.

A DRC check occurs on all of the changes to your padstacks in the layout. All of the padstacks in the design refresh unless errors exist in the padstack definition, in which case, a warning message appears and you can correct the errors in the Padstack Designer. If the padstack exists in the design (that is, you have not renamed the padstack), you are prompted to overwrite the existing padstack.

You can also choose *File – Save* from the Padstack Designer to save your padstack data to a library.

Related Topics

- [pateditdb](#)
- [Purging Unused Padstacks](#)

Editing Padstack Instances in Your Current Design

To edit padstack instances, follow these steps:

1. Run [pateditdb](#).
2. Click on the *Options* panel of the Control Panel.
3. Click *Instance*.
4. Specify a symbol name, pin name, or reference designator in the appropriate fields. You can also click on the padstack instance in the design window that you want to modify. For example:
 - To edit instances of pins of a part symbol, that reference the same padstack, type the symbol name in the Symbol field and the pin number specification in the Pin field.
 - To edit a specific pin number on a specific reference designator, type the pin number in the Pin field and the reference designator in the Ref Des field.For example, to edit pin 1 on Z18, enter 1 in the Pin field and Z18 in the Ref Des field.

Notes:

If you specify an asterisk (*) in the field, then all pins of the specified symbol (or refdes) get the new padstack.

If you specify an asterisk (*) in the Symbol field, the Ref Des field automatically changes to an asterisk (*) and all of the pins that you specify in the Pins field get the new padstack.

If you specify an asterisk (*) in the Ref Des field, the Symbol field automatically changes to an asterisk (*), and all of the pins that you specify in the Pins field get the new padstack.

If you are modifying a design padstack in the Symbol Editor, you do not see the Ref Des or Symbol fields in the *Options* panel.

1. Click *Edit*. You can also choose *Edit* by right-clicking in the design window and selecting it from a pop-up menu. The padstack designer opens with the padstack instance loaded. The banner on the Padstack Designer says

"Editing Design Instance <name>".

The tool finds the pins that match the information in the Symbol, Pin, and Ref Des boxes, and then finds the padstack name that is used by those pins. If the combination specifies pins that have more than one padstack name among them, the following error message displays:

E - More than one pad stack specified

Enter a different combination and click OK again.

1. Modify the padstack parameters and pad layers as appropriate.
2. Choose *Update to Design* from the toolbar or the *File* menu to update the padstacks of all the pins in the design that were specified. DRC executes on all of the changes to the padstacks in the layout. Padstacks in the design refresh unless errors exist in the padstack definition, in which case, a warning message appears, and you can correct the errors in the Padstack Designer.

You can also choose *File – Save* from the Padstack Designer to save your padstack data to a library.

 A derived padstack name is automatically generated if there are pins referencing the padstack that were not included in the selection set. The derived padstack name is displayed in the new name field. You can edit this name.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

padeditlib

The `padeditlib` command lets you modify a padstack from your library. To modify padstacks within the design, use `padeditdb`.

This command lets you choose a padstack from the library browser, edit its definition, and update the padstack to your library or add/update it to your current design.

For more details, see the *Defining and Developing Libraries* user guide in your documentation set.

Related Topics

- [Modifying a Padstack](#)
- [padeditdb](#)

Pad Edit Lib Dialog Boxes

Access Using

- *Menu Path: Tools – Padstack – Modify Library Padstack*

Select Library Padstack

This standard element browser lets you choose a library padstack for editing. All objects appear in alphabetical order.

To choose an element, type the name in the search field, or highlight it in the list box. To narrow the list, enter a search string in the search field and click OK. The asterisk (*) displays the complete list. For example, a search string of MTG* returns all objects beginning with MTG. Your last search is remembered.

 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.

If an element in the database has the same name as an element in the library but contains different content, the database element takes precedence in the data browser—that is, the database element is chosen.

Related Topics

- [Padstack Editor](#)

Modifying a Padstack

Follow these steps to modify a padstack:

1. Run `pateditlib`.
2. Choose the library padstack that you want to modify from the file browser and click *OK*.
The Padstack Editor dialog box appears with the padstack definition loaded. The banner of the Padstack Editor lists the name and library location of the padstack that you are modifying.
3. Edit the padstack parameters and layers as appropriate.
4. To save the padstack to a different library or to overwrite the current library padstack definition, choose *File – Save As* in the Padstack Editor.
To save the padstack in the local library, choose *File – Save*. The padstack definition is saved in the local directory and not in the padstack library.
To load the padstack into your design, choose *Update To Design* from the toolbar or the *File* menu. This option is available only if you invoke the Padstack Editor from a current design instead of using the Padstack Editor as a standalone program.

Related Topics

- [Padstack Editor](#)
- [pateditlib](#)

pads in

The `pads in` command imports information from Mentor PowerPCB and Pads Layout 2005 ASCII database files into Allegro X PCB Editor board databases. It is assumed that the PADS databases being translated are completed (placed and routed).

Familiarity with Mentor PowerPCB and Pads Layout 2005 is assumed. For additional information, refer to Mentor PowerPCB and Pads Layout 2005 documentation or contact the vendor.

The PowerPCB and Pads Layout 2005 translator reads Mentor PowerPCB and Pads Layout 2005 database files and writes an Allegro X PCB Editor board database. Due to format differences, other types of input files cannot be read. PowerPCB and Pads Layout 2005 can be used to convert an ASCII database file to a version 4 or 6 file. For further information on how to do this, see the Mentor PowerPCB and Pads Layout 2005 documentation.

Before running the PowerPCB and Pads Layout 2005 translator, you must create an ASCII version of a PADS job file, which contains all decal, part type, part, signal, route, and graphic data.

During translation, the *Pads to Allegro Translator* dialog box displays information about the translation progress. End the translation by clicking *Cancel* on this dialog box. All generated files write to your output directory, and you can use these temporary files for reference. You need the board file (`.brd`) file to edit the design.

When the translation finishes, the status dialog box closes. Use *File – Viewlog* or *File – File Viewer* to open the `pads_in.log` file and review any errors. Also examine `netin.log` file for any warnings or errors.

Syntax

To translate several databases using DOS batch files, you run the PADS translator in batch mode by specifying all required information on the command line.

⚠ Before attempting to run `pads_in`, export a PADS job file to a PADS ASCII database that can be read by the translator, as explained in [Creating a PADS ASCII Database File](#).

`pads_in <input_file> <output_directory> <options_file>`

PADS ASCII input file	Specifies the full path and name of the PADS ASCII database file.
<i>Output Directory</i>	Specifies the full path and name of the output directory.
<i>Options File</i>	Specifies the full path and name of an options file. If this does not exist, it is created.

If you run `pads_in` (note the underscore) at the operating system prompt without specifying arguments, a dialog box appears, prompting you for the data listed above. For details, see [Importing a PADS Database in Batch Mode](#).

Related Topics

- [Creating a PADS ASCII Database File](#)
- [Importing a PADS Database in Interactive Mode](#)
- [Importing a PADS Database in Batch Mode](#)
- [PADS to Allegro Translation Options Dialog Box](#)
- [Changing an Element Mapping](#)
- [Editing the Database after Importing PADS Data](#)

PADS IN Dialog Box

Access Using

- *Menu Path: File – Import – CAD Translators – PADS*

Use this dialog box to convert PADS database information.

PADS ASCII input file	Specifies the full path and name of the PADS ASCII database file.
<i>Options File</i>	Specifies the full path and name of an options file. If this does not exist, it is created.
<i>Output Design</i>	Specifies the full path and name of the output directory.
<i>Show options dialog</i>	Shows set options dialog box of translator. If this checkbox is not selected translator takes options from .ini file.

Related Topics

- [Importing a PADS Database in Interactive Mode](#)
- [Importing a PADS Database in Batch Mode](#)
- [PADS to Allegro Translation Options Dialog Box](#)
- [Changing an Element Mapping](#)
- [Editing the Database after Importing PADS Data](#)

Creating a PADS ASCII Database File

The PADS job file must be converted to a PADS ASCII database that the translator can read. This file contains all decal, part type, part, signal, route, and graphic data from the job file. An ASCII database is self-contained and does not require any information from PADS library files. To create a PADS ASCII database file:

1. Run PADS - Perform and load the JOB database file to translate.
2. Re-pour all copper pour areas (if any).
3. Click *In/Out*, then *ASCII OUT* to display the ASCII OUT form.
4. Choose the *All* option to output all data in the database. Choose this option only if you want to translate the complete database.
5. Choose the *Convert miters to lines* option to convert any route miters to lines (or arcs). If you do not choose this option, miters are not converted.
6. Specify the output file name and click *OK*.

See the PADS-Perform documentation for more information on this subject.

Related Topics

- [pads in](#)
- [Importing a PADS Database in Batch Mode](#)
- [PADS to Allegro Translation Options Dialog Box](#)
- [Changing an Element Mapping](#)
- [Editing the Database after Importing PADS Data](#)

Importing a PADS Database in Interactive Mode

Perform the following steps to import a PADS database to your design in interactive mode:

1. Run `pads in` at the command prompt.
The *PADS IN* dialog box displays.
2. Enter the PADS ASCII file to translate in the PADS ASCII Input File box. Use the *Browse* button to locate the file if necessary.
3. Enter the full path and name of an options (`.ini`) file in the *Options File* box. The translator saves all options in this file, which can be used for later translations of the same or different ASCII files.

 If you enter a file name that does not exist in the *Options File* box, the *PADS to Allegro Translation Options* dialog box is displayed during translation process to map layers between PADS Layout layer stack and Allegro class/subclass pairs.

4. Enter the output board file name and path in the *Output Design* box. By default, this box contains the path to your current working directory, where the output board file is created.
5. Select the checkbox *Show options dialog* to open the Options dialog box.
6. Click *Run*.
The *PADS to Allegro Translator* dialog box displays the translation status. On successful run, a new board file with the name provided is created in the *Output Design* location.
7. When the translation is completed, click *Close* to dismiss the PADS IN dialog box.

 Click *Cancel* to stop the translation.

Related Topics

- [pads in](#)
- [PADS IN Dialog Box](#)
- [PADS to Allegro Translation Options Dialog Box](#)
- [Changing an Element Mapping](#)
- [Editing the Database after Importing PADS Data](#)

Importing a PADS Database in Batch Mode

Perform the following steps to import a PADS database to your design in batch mode:

This section describes the procedure for running `pads_in` from the operating system prompt when you do not qualify the command with arguments.

1. Export a PADS job file to a PADS ASCII database that the translator can read, as explained in [Creating a PADS ASCII Database File](#).
2. At a command line prompt, type:

`pads_in`

The File Names dialog box appears.

3. Enter the name (and full path) of the `.ascii` file being input in the *PADS ACSII Input File* field. Use the *Browse* button to locate the file if necessary.
4. Enter a directory to which to write the output file in the *Output Directory* field. Use the *Browse* button to locate the directory if necessary.
5. Enter the full path to an options file in the *Options File* field. Use the *Browse* button to locate the file if necessary.
6. Click OK to continue.

`pads_in` reads the input file and determines the number of etch/conductor layers it uses. If all required program arguments are not specified, the PADS to Allegro Translation Options dialog box appears.

Related Topics

- [pads in](#)
- [PADS IN Dialog Box](#)
- [Creating a PADS ASCII Database File](#)
- [Changing an Element Mapping](#)
- [Editing the Database after Importing PADS Data](#)

PADS to Allegro Translation Options Dialog Box

This dialog provides the number and name of PADS layer. Use this dialog box for layer mapping between PADS Layout layer stack and Allegro class/subclass pairs.

<i>PADS to Allegro layer mapping</i>	Specifies the layer number, layer name, Class and Subclass for all the PADS objects (<i>Line, Copper, Text, Decal, Pad and Via</i>). For the LINE, COPPER and TEXT objects, layer number 0 means all layers. The translator automatically maps this layer to the BOARD/SUBSTRATE GEOMETRY class an ALL subclass. For the PAD and VIA objects, layer 0 means all internal layers. The translator maps this layer to class ETCH/ VIA class. All the internal ETCH/CONDUCTOR subclasses are defined in <code>pads_in</code> as internal pad definitions. Similarly, for the LINE, COPPER and TEXT objects, layer number 1 means first route layer. The translator automatically maps this layer to ETCH/CONDUCTOR class. The first and last route layers maps to TOP and BOTTOM subclass respectively. The name of the other subclass for internal route layers is chosen from the PADS and is same as the PADS layer name. For the DECAL objects, the translator maps layers 0 and 1 to PACKAGE GEOMETRY/SILKSCREEN_TOP class/subclass. ⚠ The translator also maps additional information about layers which is present in the ASCII file. These layers usually contain soldermask, silkscreen, and pastemask objects.
<i>Class</i>	Choose to specify the Class for an unused layer in PADS or to change the Class of a pre-defined layer in PADS.
<i>Sub Class</i>	Choose to specify the Subclass for an unused layer in PADS or to change the Subclass of a pre-defined layer in PADS.
<i>Automatic solder layer creation</i>	
<i>Create solder layers</i>	Choose to create solder mask and solder paste padstack layers.
<i>Mils to oversize</i>	Specify the oversize radius in mils.
<i>Do not create Teardrops</i>	Choose to not create teardrops.
<i>Create Dynamic Shapes</i>	Choose to create dynamic shapes from the *POUR* section of PADS ASCII file. Dynamic shapes coming in through PADS and other VIF/IGES sources are sorted by relative area (or size) to preserve the smaller shapes in your design.
<i>Through pin thermal/ anti pad creation</i>	
<i>Create thermal/anti pad</i>	Choose to create thermal or anti pad.
<i>Antipad oversize</i>	Specify the antipad oversize value in outer diameter. Default is 5 mils.
<i>Thermal pad oversize</i>	Specify the thermal pad oversize value beyond the inner diameter. Default is 5 mils.
<i>Spoke width</i>	Specify thermal pad channel width. Default is set to 10 mils.
<i>OK</i>	Click <i>OK</i> to confirm PADS to Allegro layer mapping and start the translating process.
<i>Cancel</i>	Click <i>Cancel</i> to stop the translating process.

Related Topics

- [pads in](#)
- [PADS IN Dialog Box](#)
- [Creating a PADS ASCII Database File](#)
- [Importing a PADS Database in Interactive Mode](#)
- [Editing the Database after Importing PADS Data](#)

Changing an Element Mapping

To change an element in mapping:

1. Choose the mapping in the dialog box of the PADS to Allegro Translation Options dialog box.
The target class and subclass mappings of the element are chosen in the *Class* and *Sub Class* fields.
2. To change the target class, select a new class from the drop-down list.
3. To change the target subclass, select a new subclass from the drop-down list.
—or—
Type a new subclass name in the *Sub Class* field.

 You cannot define new subclass names if the class is PIN or VIA.

4. Choose *Create solder layers* to create solder mask and solder paste padstack layer entries.
5. Specify the oversize radius in mils.
All non-zero sized pad entries are copied, and the oversize radius added to the pad entry size. For example, if the oversize is 15, a 60-mil pad generates a 75-mil solder mask and paste layer entry. The solder mask pad layer entry is added to the SOLDERMASK_TOP subclass, and the solder paste entry is added to the PASTEMASK_TOP subclass.
6. Choose *Do not create Teardrops* if you do not want the translator to create teardrops.
7. Choose *Create Dynamic Shapes* if you want the translator to create dynamic shapes from POUROUT and HATOUT pieces from the *POUR* section.
For more information on dynamic shapes, see *Preparing for Layout in the user guide*.
8. Choose *Create thermal/anti pad* if you want to generate thermal or anti pad during translation.
9. Specify the oversize values for thermal and anti pads.
10. Specify the *Spoke width*.
11. Click *OK* to translate or *Cancel* to stop translating.

Related Topics

- [pads in](#)
- [PADS IN Dialog Box](#)
- [Creating a PADS ASCII Database File](#)
- [Importing a PADS Database in Interactive Mode](#)
- [Importing a PADS Database in Batch Mode](#)

Editing the Database after Importing PADS Data

After the PADS or PCAD translation finishes, load the board/substrate file into the editor. To create a design that can be maintained completely within the editor, follow these steps:

1. Run `viewlog` to display the translators' log file. Examine the file for any errors or warnings. Also examine `netin.log` file for any warnings or errors.

 The log files are `pads_in.log` and `pcad_in.log`.

2. Choose *Display – Color/Visibility* (`color192` command) and choose *All Invisible* from the *Global Visibility* box of the *Color /Visibility* dialog box.

3. Click *Yes* in the confirmation box, and click *Apply* on the *Color/Visibility* dialog box.

All the classes and subclasses in the design become invisible.

4. Enable visibility for the following classes/subclasses:

- *Drawing Format/Outline* in the *Manufacturing* group
- *Pin, Via, DRC, and Etch/Conductor* in the *Stack-Up* group

5. Set colors for the following classes/subclasses.

- *Drawing Format/Outline* in the *Manufacturing* group
- *Pin, Via, DRC, and Etch/Conductor* in the *Stack-Up* group
- *Ratsnest* in the *Display* group

Cadence recommends for ease of viewing that you set *DRC* and *ratsnests* in red.

6. Click *Apply*.

The reset classes/subclasses in the design become visible.

7. Click *OK*.

8. Choose *Setup – Design Parameters* (`prmed` command) to display the *Design Parameter Editor*.

9. Choose the *Display* tab and set *DRC Marker Size* to 125 (or a parameter of your own choosing) and clear the *Grid* check box.

The grid display in the interface disappears.

10. Choose the *Design* tab and set *Default Symbol Height* to 125 (or a parameter of your own choosing).

11. Click *OK* to implement the changes and close the dialog box.

12. Verify that thermal relief and antipad are created for through pin.

13. Verify the keepin/keepout areas. (The translator creates placement keepouts for all package/part symbols based on the boundaries of objects found within the PADS decal.)

14. Set the appropriate constraints for your design. Depending on the version of the tool you are using and the type of constraints you are setting, choose *Setup – Constraints Constraint Manager* (`cmgr` command).

15. Choose *Tools – Database Check* (`dbdoctor` command) to verify the integrity of a design drawing database.

The DBdoctor (Database Health Monitor) dialog box appears.

16. Set the parameters and click *Check* to verify the database.

17. View the log file, and make the necessary corrections.

The translator also adds an anti-pad and thermal-relief layer entries to padstacks. Thermal reliefs are a flash with the same name as the pad. You may modify these settings. The translator determines the size of anti-pads using the *COPPOURSPACE* parameter in the **PCB** section of the ASCII file. Any oval or rectangular finger pad entries convert to shapes in the design database because PADS allows ovals and fingers to be rotated within a padstack.

If your PADS database contained negative power planes with power ties, generate these ties as thermal-reliefs during film creation.

Related Topics

- [pads in](#)
- [PADS IN Dialog Box](#)
- [Creating a PADS ASCII Database File](#)
- [Importing a PADS Database in Interactive Mode](#)
- [Importing a PADS Database in Batch Mode](#)
- [PADS to Allegro Translation Options Dialog Box](#)

pads lib in

The `pads lib in` command imports information from PowerPCB and Pads Layout 2005 ASCII library files into Allegro X PCB Editor symbol drawing databases.

Familiarity with PowerPCB and Pads Layout 2005 is assumed. For additional information, refer to PowerPCB and Pads Layout 2005 documentation or contact the vendor.

Before running the `pads lib in` translator, you must export PADS library into ASCII files, which contain all decals and parts data.

During translation, the *Pads Layout Library to Allegro Translator* dialog box displays information about the translation progress. End the translation by clicking *Cancel* on this dialog box. All generated files write to your output directory, and you can use these files for reference. You need the symbol drawing file (`.dra`) file to edit the translated components.

When the translation finishes, the status dialog box closes. Use *File – Viewlog* or *File – File Viewer* to open the `pads_lib_in.log` file and review any errors.

For more information, see the the *Transferring Logic Design Data* user guide.

Syntax

To translate PADS Layout ASCII library files using general script batch files, you run the PADS Library translator in batch mode by specifying all required information on the command line.

```
pads_lib_in [-psm] [-dev] [-bsm] [-custom] {-input<input_directory>} [-output<output_directory>] [-opt<options_file>]
```

Required Arguments

PADS Library input directory	Specifies the full path and name of the PADS Layout ASCII library directory.
<i>Output directory</i>	Specifies the full path and name of the output directory.
<i>Options file</i>	Specifies the full path and name of an options file. If the option file does not exist, enter a new name in the <i>Enter Options Output File</i> dialog box and translator creates it with default settings.

Optional Arguments

-psm	Creates package symbol files.
-dev	Creates device files.
-bsm	Create mechanical symbol files.
-custom	Creates drawing files for the custom padstacks.

If you run `pads_lib_in` (note the underscore) at the operating system prompt without specifying arguments, a dialog box appears, prompting you for the data listed above. For details, see [Importing a PADS Layout Library in Batch Mode](#).

Related Topics

- [PADS Layout Library to Allegro Translator Options Dialog Box](#)
- [PADS to Allegro Layer Mapping](#)
- [Importing a PADS Layout Library in Interactive Mode](#)
- [Importing a PADS Layout Library in Batch Mode](#)
- [Changing an element mapping](#)

PADS Library Translator Dialog Box

Access Using

- *Menu Path: File – Import – CAD Translators – PADS Library*

Use this dialog box to convert PADS Layout ASCII library files information.

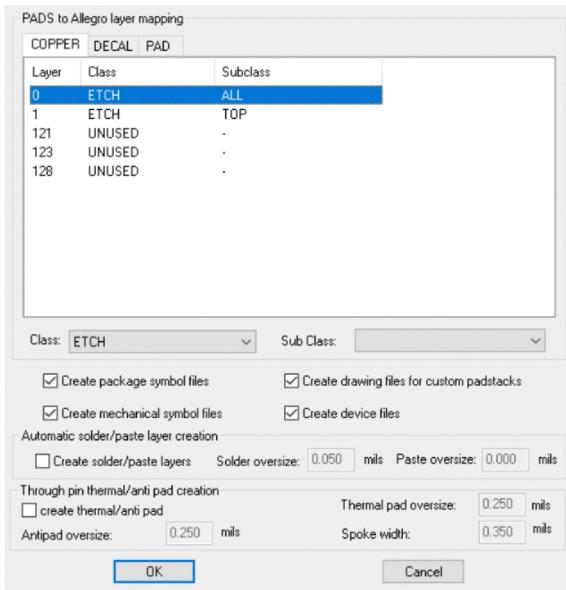
PADS Library Directory	Specifies the full path and name of the directory with PADS Layout ASCII library files.
Options File	Specifies the full path and name of an options file. If this file does not exist, enter a new name in the <i>Enter Options Output File</i> dialog box and translator creates it with default settings.
Output Directory	Specifies the full path and name of the output directory.
Show options dialog	Shows set options dialog box of translator. If this checkbox is not selected translator takes options from .ini file.

Related Topics

- [PADS to Allegro Layer Mapping](#)
- [Importing a PADS Layout Library in Interactive Mode](#)
- [Importing a PADS Layout Library in Batch Mode](#)
- [Changing an Element Mapping](#)

PADS Layout Library to Allegro Translator Options Dialog Box

Use this dialog box for layer mapping between PADS Layout layer stack and Allegro class/subclass pairs.



<i>PADS to Allegro layer mapping</i>	Specifies the layer name, Class and Subclass for all the PADS objects (<i>Copper, Decal, and Pad</i>).
<i>Class</i>	Choose to specify the Class for an unused layer in PADS or to change the Class of a pre-defined layer in PADS.
<i>Sub Class</i>	Choose to specify the Subclass for an unused layer in PADS or to change the Subclass of a pre-defined layer in PADS.
<i>Create package symbol files</i>	Choose to create package symbol files(<i>.psm</i>).
<i>Create mechanical symbol files</i>	Choose to create mechanical symbol files(<i>.bsm</i>). The translator automatically creates padstack files (<i>.pad</i>) files with <i>.psm</i> and <i>.bsm</i> files. The padstack files are required for correct loading of them into Allegro.
<i>Create drawing files for custom padstacks</i>	Choose to create drawing files for custom padstacks to create symbol drawing (<i>.dra</i>) files for the custom shapes.
<i>Create device files</i>	Choose to create device files(<i>.txt</i>).
Automatic solder/paste layers	
<i>Create solder/paste layers</i>	Choose to create solder mask and solder paste padstack layers.
<i>Solder oversize</i>	Specify the oversize radius in mils for solder mask layer.
<i>Paste oversize</i>	Specify the oversize radius in mils for solder paste layer.
Through pin thermal/ anti pad creation	
<i>Create thermal/anti pad</i>	Choose to create thermal or anti pad.
<i>Antipad oversize</i>	Specify the antipad oversize value in outer diameter.
<i>Thermal pad oversize</i>	Specify the thermal pad oversize value beyond the inner diameter.
<i>Spoke width</i>	Specify thermal pad channel width.
<i>OK</i>	Click <i>OK</i> to confirm PADS to Allegro layer mapping and start the translating process.

Cancel

Click *Cancel* to stop the translating process.

Related Topics

- [pads lib in](#)
- [Importing a PADS Layout Library in Interactive Mode](#)
- [Importing a PADS Layout Library in Batch Mode](#)
- [Changing an Element Mapping](#)

PADS to Allegro Layer Mapping

The translator shows only used PADS layers and provides layer mapping from PADS to Allegro for each object.

- COPPER: Use to create etch items (shapes, clines) on top or bottom layer in the layout editor.
- DECAL: Use to create package geometry (lines, and shapes).
- PAD: Use to create padstack layers and solder/paste mask layers.

Related Topics

- [pads lib in](#)
- [PADS Library Translator Dialog Box](#)
- [Importing a PADS Layout Library in Batch Mode](#)
- [Changing an Element Mapping](#)

Importing a PADS Layout Library in Interactive Mode

Follow these steps to import a PADS layout library to your design in interactive mode:

1. Run `pads lib in` at the command prompt.
The PADS Library Translator dialog box displays.
2. Enter the PADS layout ASCII library files directory path in the *PADS Library Directory* field. Use the *Browse* button to locate the directory if necessary.
3. Enter the full path and name of an options (`.ini`) file in the *Options File* box. The translator saves all options in this file, which can be used for later translations of the same or different ASCII library files.

 If you enter a file name that does not exist in the Options File box, the PADS Layout Library to Allegro Translation Options dialog box displays.

1. Enter the output directory name in the *Output Directory* box.
2. Select the checkbox *Show options dialog* to open the options dialog box.
3. Click *Run*.
The PADS Layout Library to Allegro Translator dialog box displays the translation status.
4. When the translation stops, click *Close* to dismiss the PADS LIBRARY TRANSLATOR dialog box.

 Click *Cancel* to stop the translation.

Related Topics

- [pads lib in](#)
- [PADS Library Translator Dialog Box](#)
- [PADS Layout Library to Allegro Translator Options Dialog Box](#)
- [Changing an Element Mapping](#)

Importing a PADS Layout Library in Batch Mode

To run `pads_lib_in` from the operating system prompt when you do not qualify the command with arguments:

1. At a command line prompt, type:

`pads_lib_in`

The PADS Layout Library to Allegro Translator dialog box appears.

2. Enter the full path of the PADS layout ASCII library files directory in the *PADS Library input directory* field. Use the *Browse* button to locate the directory if necessary.
3. Enter a directory to which to write the output file in the *Output directory* field. Use the *Browse* button to locate the directory if necessary.
4. Enter the full path to an options file in the *Options File* field. Use the *Browse* button to locate the file if necessary.
5. Click OK to continue.

Or

You can also run the `pads_lib_in` command from the operating system prompt with all the required arguments. The following example uses `pads_lib` as PADS Library path and `test_output` as an output directory path.

```
pads_lib_in -input ./pads_lib -opt ./test.ini -output ./test_output
```

`pads_lib_in` reads the input files and determines the number of etch/conductor layers it uses. If all required program arguments are not specified, the PADS Layout Library to Allegro Translation Options dialog box appears.

The PADS to Allegro Layer Mapping fields define the element-layer mappings. The PADS objects (Copper, Decal, and Pad) are displayed in different tabs with the name of the class and subclass to which you can map the objects.

Related Topics

- [pads lib in](#)
- [PADS Library Translator Dialog Box](#)
- [PADS Layout Library to Allegro Translator Options Dialog Box](#)
- [PADS to Allegro Layer Mapping](#)

padstack_editor

The `padstack_editor` batch command invokes the *Padstack Editor*, where you create and edit padstacks and save them to a library. The command also provides options to create and modify padstack definitions using an XML file.

For information concerning the user interface fields and procedures, see [Padstack Editor](#).

For overview and prerequisite information, see the *Defining and Developing Libraries* user guide in your documentation set.

Related commands are [paddeditdb](#) and [paddeditlib](#).

Access Using

For Windows only:

From the *Start* menu, choose *Cadence PCB Utilities 17.4-2019 – Padstack Editor 17.4*

Syntax

For invoking Padstack Editor UI:

```
padstack_editor
```

For exporting padstack data in an pXML format:

```
padstack_editor -xo <file name>.pad
```

For importing pXML data in non-graphical mode:

```
padstack_editor -x <file name>.pxml
```

For importing pXML data in graphical mode:

```
padstack_editor -xg <file name>.pxml
```

 The input .pxml file may contain single or multiple padstack definitions.

Examples

- Exporting Padstack Data

```
padstack_editor -xo via.pad
```

The command process the input `via.pad` file, save it as `via_pad2xml.pxml` file and exit. The *Padstack Editor* does not invoke and saves the errors or warnings in the `via_pad2xml.log` file.

```
padstack_editor -xo *.pad
```

The command process all the input `.pad` files, save them as a single `.pxml` file and exit. The pXML file is named after the first `.pad` file, for example `via_pad2xml.pxml`. The *Padstack Editor* does not invoke and saves all the errors or warnings for each padstack in a single log file that is also named after the first `.pad` file, which is processed by the command.

- Importing Padstack Data

```
padstack_editor -x via.pxml
```

The command parses the input pXML file to fill in the padstack data, runs the checks, save padstack (`.pad`) file and exit. Parsing pXML file with multiple padstack definitions creates separate `.pad` file for each padstack.

```
padstack_editor -xg smd.pxml
```

The command parses the input pXML file to fill in the padstack data and display it in the *Padstack Editor*. You can run the checks and save the padstack.

If the input pXML file has multiple padstacks data, then the only first padstack is parsed and displayed in the *Padstack Editor*.

For more information, see [Managing Padstack Data Using XML Files](#).

Padstack Editor

Padstack Editor lets you create and edit padstacks and save them to your design, to a library, or to both at once. You define the pad size and shape for all etch/conductor and non-etch/conductor mask layers in the Padstack Editor.

Default routing layers are BEGIN layer, DEFAULT INTERNAL, and END layer. The DEFAULT INTERNAL padstack definition is used by default when you add more layers in your design. When the padstack is placed in the footprint, the BEGIN layer is mapped to the TOP substrate layer, and the END layer is mapped to the BOTTOM substrate layer.

Non-etch/conductor mask layers include fixed and user-defined mask layers. The fixed mask layers are SOLDERMASK_TOP, SOLDERMASK_BOTTOM (for soldermask artwork) and PASTEMASK_TOP, PASTEMASK_BOTTOM (for solder paste artwork). An extra layer pair named FILMMASK_TOP and FILMMASK_BOTTOM is available for use in whatever means you wish. These two layers are optional and do not have to be used or defined. You can also define up to a maximum of 32 user-defined mask layers, which you add according to your design flow requirements. This may include, but not be limited to, applications associated with via plugging, hard gold, soft gold, silver, or other soldermask alternatives.

The Padstack Editor also has the capability to coordinate with external tools for creating and maintaining padstack libraries. Using import and export functionality, you can share the padstack data in an XML format to external tools for modification and then import it back to Padstack Editor.

For an overview concerning padstack creation, see the *Defining and Developing Libraries* user guide in your documentation set.

Related Topics

- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Padstack Editor Tabs and Panels

The Padstack Editor comprises the *Start*, *Drill*, *Secondary Drill*, *Drill Symbol*, *Drill Offset*, *Design Layers*, *Mask Layers*, *Options*, and *Summary* tabs.

2D Padstack Views

The editor has 2D padstack views, consisting of 2D Top Padstack View and 2D Padstack Side Views. You can dock, undock, and resize these view windows.

Status Bar: Padstack Type, Units, and Decimal Places



The Status bar displays the padstack type on the left corner.

In addition, there are two fields for units and accuracy or decimal places. All of the fields that use measurement refer to this value. You should choose the units and accuracy to suit your design environment, but remember that these units are converted to board units and accuracy, which may result in rounding when you apply the units to a design.

Units	Specifies the unit of measurement for the padstack. Choices are Mils, Inch, Millimeter, Centimeter, and Micron.
<i>Decimal places</i>	Specifies the precision of the units of measurement. Choose a number from 0 through 4, which sets the number of decimal places.

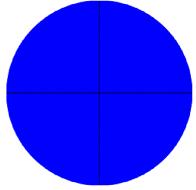
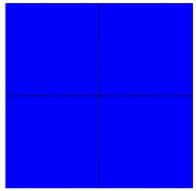
Start Tab

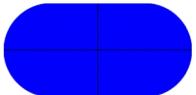
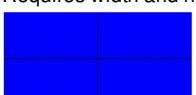
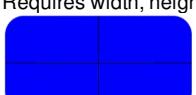
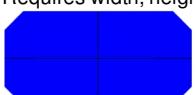
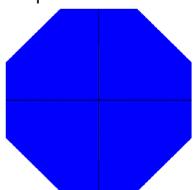
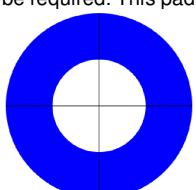
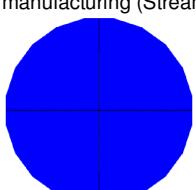
Select padstack usage	Select a padstack type based on the intended usage. Depending on the padstack type selected, other tabs in the editor may be grayed out to prevent an incorrect padstack definition.
Select pad geometries	Select a pad geometry to define the default pad geometry type for the padstack definition in the Design Layers and Mask Layers tabs.

Padstack Type

<i>Thru Pin</i>	Specifies that the padstack penetrates all layers. This is the default padstack type. You must specify the drill hole parameters for the through padstack type.
<i>SMP Pin</i>	Specifies that the padstack is for only one layer. You usually use this option for SMT pads.
<i>Via</i>	Specifies a plated through hole.
<i>BBVia</i>	Specifies that the padstack is blind (navigates the surface and internal layers) or buried (traverses internal layers). When creating a blind or buried via, you can also define the via diameter, height, and width. You can also define the via as a microvia.
<i>Microvia</i>	Specifies a blind or buried via as a microvia, defined by the IPC – Association Connecting Electronics Industries as less than or equal to 0.15 mm or 5.91 mils in diameter, with a pad diameter less than 0.35 mm or 13.8 mils formed by laser or mechanical drilling, wet/dry etching, photo imaging, or conductive ink formation followed by a plating operation. Enables the padstack to be used for HDI features, including advanced DRC. Use on HDI boards where spacing rule sets differ between HDI and conventional blind/buried or core vias. Only applicable to a blind/buried via type, which should be used to represent mechanical drilled vias, such as the core via on an HDI design. The two via types are supported in the Spacing and Same Net Spacing Constraint domains of Constraint Manager. Note: With a <i>Microvia</i> padstack, you can apply microvia specific constraints to the vias instead of the standard blind or buried constraints. If a design with a microvias is opened with a product license that does not support microvia constraints, a warning message appears indicating that the DRCs will no longer check microvia constraints. With the next DRC update, the DRCs of the design become obsolete; microvias in the design will be checked for design rule violations as blind or buried vias, and not as microvias.
<i>Slot</i>	Specifies a slotted hole.
<i>Microvia Slot</i>	Specifies a slotted microvia hole, which is similar to a microvia but with the ability to specify a plated slot instead of a circular or square hole. Note: <i>Microvia slots</i> will use the same constraints as standard <i>Microvias</i> .
<i>Hole</i>	Specifies a type of mechanical hole used during the manufacturing process and for component mounting and alignment. Depending upon the manufacturing purpose, you can select from <i>Mechanical Hole</i> , <i>Tooling Hole</i> , or <i>Mounting Hole</i> . These holes align with the current IPC-2581 schema and exported as manufacturing data for IPC-2581. For more information about IPC-2581 schema, visit ipc2581.com .
<i>Fiducial</i>	Specifies a fiducial pad on the beginning or end layer.
<i>Bond Finger</i>	Specifies a padstack on one layer, either beginning or end. Select this if you are working with wire bond dies.
<i>Die Pad</i>	Specifies a padstack for a die on one layer, either beginning or end.

Pad Geometries

<i>Circle</i>	Requires a diameter. 
<i>Square</i>	Requires a width. 

<i>Oblong</i>	Requires width and height. 
<i>Rectangle</i>	Requires width and height. 
<i>Rounded Rectangle</i>	Requires width, height, and a corner radius. You can select one or more corners. By default, all four corners are selected. 
<i>Chamfered Rectangle</i>	Requires width, height, and length of the chamfers. You can select one or more corners. By default, all four corners are selected. 
<i>Octagon</i>	Requires width. 
<i>Donut</i>	Requires a inner and outer diameters. You can use this geometry for fiducials and mounting holes where contact with a chassis ground may be required. This pad geometry uses a different connectivity model (touch vs. connect point) from all other pad geometries. 
<i>n-Sided Polygon</i>	Requires diameter and number of sides. This can be used in applications related to vectoring of pad figures when exporting artwork data for manufacturing (Stream data). The number sides must be even. 

Drill Tab

The fields in this tab depends on whether you selected a padstack type with a hole or a slot.

The following fields are available for padstacks with a hole.

<i>Drill hole</i>		
<i>Hole Type</i>		Specifies the type of drill to be created. The values are <i>circle</i> and <i>square</i> .
<i>Finished hole</i>		
	<i>Finished Diameter</i>	Specifies the intended finished hole size. This value is based on the current units and accuracy defined.

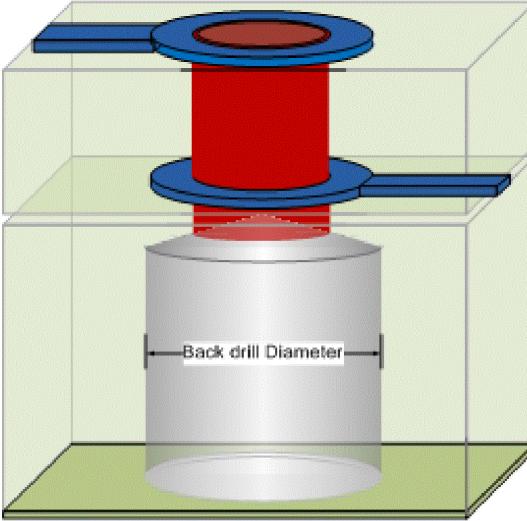
	<i>+Tolerance</i>	Specifies the + (positive) tolerance of the finished hole. The value must be greater than or equal to zero.
	<i>-Tolerance</i>	Specifies the - (negative) tolerance of the finished hole. The value must be greater than or equal to zero.
<i>Drilled hole</i>		Note: The <i>Drilled hole</i> fields are enabled only when the drilled hole is plated.
	<i>Hole Diameter</i>	Specifies the intended drilled hole size before plating. This value is based on the current units and accuracy defined for the padstack. The diameter of the drilled hole must be greater than the intended size of the finished hole. This value must be greater than or equal to zero.
	<i>+Tolerance</i>	Specifies the + (positive) tolerance of the drilled hole. The value must be greater than or equal to zero.
	<i>-Tolerance</i>	Specifies the - (negative) tolerance of the drilled hole. The value must be greater than or equal to zero.
<i>Drill tool size</i>		Specifies the actual drill size or tool size to be used to drill the hole before plating. For example, if specified is not aligned with database units.
<i>Non-standard drill</i>		Specifies a non-standard drill bit. The values are <i>Laser</i> , <i>Plasma</i> , <i>Punch</i> , <i>Wet/dry etching</i> , <i>Photo imaging</i> .
<i>Hole plating</i>		
<i>Hole/slot plating</i>		Specifies if the hole is <i>Plated</i> or <i>Non-plated</i> . The available values depend on the padstack type selected.
<i>Drill rows and columns</i>		Define custom multi-drill patterns for adding hole patterns.
<i>Pattern style</i>		Three style types are available: <i>Array</i> , <i>Polar</i> , and <i>Custom</i>
	<i>Array</i>	Choose to define the default multi-drill configuration
	<i>Number of drill rows</i>	Specifies the number of drills along the Y-axis.
	<i>Number of drill columns</i>	Specifies the number of drills along the X-axis.
	<i>Clearance between columns</i>	Specifies distance between the hole edge to hole edge of the drilled hole column.
	<i>Clearance between rows</i>	Specifies the distance between the hole edge to hole edge of the drilled hole row.
	<i>Pitch between columns</i>	Specifies the distance between the centers of columns.
	<i>Pitch between rows</i>	Specifies the distance between the centers of rows.
	<i>Drills are staggered</i>	Specifies that for multiple drills, the holes should be staggered. The stagger offsets each row and column.
	<i>Polar</i>	Choose to define circular multi-drill pattern
	<i>Pattern radius</i>	Specifies the radius of the circle formed by the drill holes
	<i>Drills</i>	Specifies number of drill holes
	<i>Start angle</i>	Specifies the angle of the first drill hole
	<i>Custom</i>	Choose to define custom multi-drill pattern in a grid format. Multiple holes locations can be added and deleted.
	<i>Drill</i>	Specifies number of drill hole
	<i>X, Y location</i>	Specifies location of drill hole

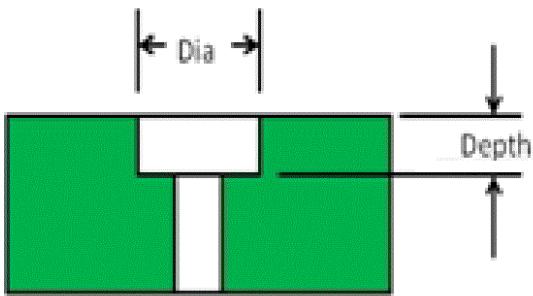
The following fields are available if you selected a Slot.

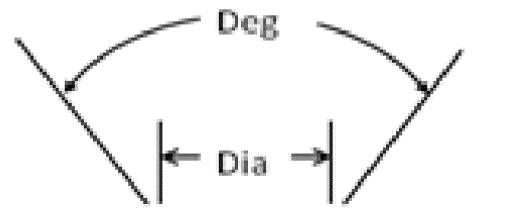
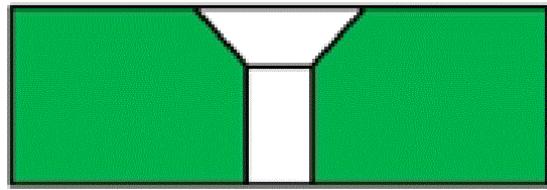
<i>Slot</i>	
<i>Slot type</i>	Specifies the type of slot. The values are <i>Rectangular slot</i> and <i>Oval slot</i> .
<i>X Size</i>	Specifies the size of the finished hole along the X-axis. The value is based on the current units and accuracy defined for the padstack. The value for the slot X axis size must be greater than zero (0.0).

		X Tolerance	Specifies the tolerance along the X-axis. The value is based on the current units and accuracy defined for the padstack. The value for the slot X-axis size must be greater than zero (0.0).
Y Tolerance	Specifies the tolerance along the Y-axis. The value is based on the current units and accuracy defined for the padstack. The value for the slot Y-axis size must be greater than zero (0.0).		
Hole plating			
Hole/slot plating	Specifies if the hole is <i>Plated</i> or <i>Non-plated</i> .		

Secondary Drill Tab

Backdrill	Enabling backdrill adds two additional layers BACKDRILL_START and BACKDRILL_CLEARANCE to Design Layers tab and a new layer BACKDRILL_SOLDERMASK to Mask Layers tabs. BACKDRILL START: Specifies regular pad geometry used on start layer of a backdrill. BACKDRILL CLEARANCE: Specifies anti pad geometry used on internal negative plane layers which are backdrilled. BACKDRILL CLEARANCE: Specifies route keepout used on internal layers which are backdrilled. BACKDRILL SOLDERMASK: Specifies soldermask opening used on backdrill start layer.
Diameter	Specifies finished hole size for backdrill. The value is based on the current units and accuracy defined for the padstack. The value for the diameter size must be greater than zero (0.0).
	
Backdrill drill symbol	
Type of drill figure	Specifies the drill figure type. The available drill figure types are <i>None</i> , <i>Circle</i> , <i>Square</i> , <i>Hexagon X</i> , <i>Hexagon Y</i> , <i>Octagon</i> , <i>Cross</i> , <i>Diamond</i> , <i>Triangle</i> , and , , and .
Characters	Specifies the character(s) to be used to distinguish one drill from another with similar drill figures. A maximum of three (3) characters is allowed. The supported characters are A-Z,a-z,0-9, @#\$%^&*()_+=<>,?/[]{};::` and space. The characters ! and " are not allowed.
Drill figure width	Specifies the size of the figure along the X-axis of the drill figure. This value is based on the current units and accuracy defined for the padstack and must be greater than 0.0. Available for all figures other than Square and Circle.

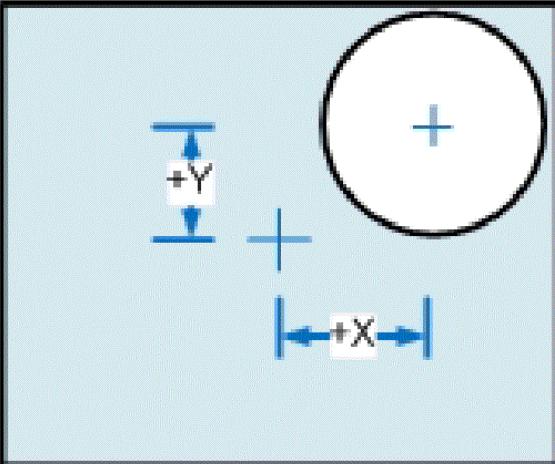
<i>Drill figure height</i>	Specifies the size of the figure along the Y-axis of the drill figure. This value is based on the current units and accuracy defined for the padstack and must be greater than 0.0. Available for all figures other than Square and Circle.
<i>Counterbore/Countersink</i>	Enables the secondary drill data definition set to provide documentation for counterbore and countersink operations.
<i>Counterbore</i>	Specifies whether a <i>Counterbore</i> or <i>Countersink</i> is to be drilled. No DRC action is performed based on these values or special drill files are created; used for documentation purposes only with a special Drill Legend generated.
<i>Primary side</i>	Specifies the side from which the counterbore/countersink is to be drilled. The options are <i>Primary side</i> (default) and <i>Secondary side</i> . <i>Primary side</i> indicates that the counterbore/countersink is to be drilled from the side of the board which the component is placed on. <i>Secondary side</i> indicates the opposite side of the board. For example, if the component is placed mirrored on the bottom layer of the board and <i>Primary side</i> is selected, the counterbore/countersink is drilled from the bottom side of the board. Conversely, the counterbore/countersink is drilled from the top side of the board if <i>Secondary side</i> is selected.
<i>Diameter</i>	Specifies finished hole size for the counterbore or countersink diameter. The value is based on the current units and accuracy defined for the padstack. The value for the diameter size must be greater than zero (0.0).
<i>+Tolerance</i>	Specifies the + tolerance. The value is based on the current units and accuracy defined for the padstack. The values for the drill tolerances must not be less than or zero (0.0).
<i>-Tolerance</i>	Specifies the - tolerance. The value is based on the current units and accuracy defined for the padstack. The values for the drill tolerances must not be less than or zero (0.0).
<i>Depth</i>	Specifies the depth of the counterbore hole from the top surface of the board. This value is based on the current units and accuracy defined for the padstack. The values for the counterbore depth must be greater than zero (0.0). Available if <i>Counterbore</i> is selected.  A technical diagram illustrating a counterbore hole. It shows a cross-section of a green rectangular board. A white rectangular component is mounted on the board. A vertical dimension line with arrows at both ends is labeled "Dia", indicating the diameter of the hole. Another vertical dimension line with arrows at both ends is labeled "Depth", indicating the distance from the top surface of the board to the bottom of the counterbore.

<i>Angle</i>	<p>Specifies the angle of the countersink. This value is defined in angles of degrees. Available if <i>Countersink</i> is selected.</p>  
--------------	---

Drill Symbol Tab

<i>Define a drill symbol</i>	
<i>Type of drill figure</i>	Specifies the drill figure type. The available drill figure types are <i>None</i> , <i>Circle</i> , <i>Square</i> , <i>Hexagon X</i> , <i>Hexagon Y</i> , <i>Octagon</i> , <i>Cross</i> , <i>Diamond</i> , <i>Triangle</i> , and , , and .
<i>Characters</i>	Specifies the character(s) to be used to distinguish one drill from another with similar drill figures. A maximum of three (3) characters is allowed. The supported characters are A-Z,a-z,0-9, @#\$%^&*()_+-=<>,?/[]{}";::` and space. The characters ! and " are not allowed.
<i>Drill figure size</i>	Specifies the width of a square drill figure or the diameter of a circular drill figure. Available only if you select Square or circle.
<i>Drill figure width</i>	Specifies the size of the figure along the X-axis of the drill figure. This value is based on the current units and accuracy defined for the padstack and must be greater than 0.0. Available for all figures other than Square and Circle.
<i>Drill figure height</i>	Specifies the size of the figure along the Y-axis of the drill figure. This value is based on the current units and accuracy defined for the padstack and must be greater than 0.0. Available for all figures other than Square and Circle.

Drill Offset Tab

	
<i>Offset X</i>	Specifies the offset of the finished holes from the padstack origin along the X-axis.
<i>Offset Y</i>	Specifies the offset of the finished holes from the padstack origin along the Y-axis.

Design Layers Tab

This tab defines both the number of layers in the padstack and the individual layer type. When you define a padstack for your library, you only see the layers that you define in the table, but when you open a padstack from the design, you see all of the layers that are related to that padstack.

The top section displays all padstack layers with the corresponding *Regular Pad*, *Thermal Relief*, *Anti Pad*, and *Keep Out* definitions. A tabular format represents padstack layers with each row representing a single pad layer. For library padstacks, you can change the padstack layer name in the Layer column of any highlighted layer. For layout padstacks, the layer name is not editable in the Padstack Designer. To edit a layer name in a layout padstack, you must use the Layout Cross-section. You can edit any of the white fields and Insert, Delete, Copy, and Paste layer definitions.

You do not need to fill in all of the options to save the padstack. To save a padstack file, you must define a pad for at least one layer.

You may define a default pad set to be used on internal layers for DRC (design rule checking) of line-to-pad spacing. The INTERNAL_LAYER pad layer allows you to define a single pad layer set that maps to all internal layers when the padstack is loaded into the board. Layers defined specifically by name override the INTERNAL_LAYER pad during board mapping. The pad definition triggers DRC.

If no pad definitions exist on internal layers, this particular check does not occur. This may cause etch/conductor to be placed in the path of a drill hole without DRC flagging it. You define the pad size smaller than the drill hole so the pad will be drilled out during manufacturing. As long as a pad is defined, DRC uses the larger of the drill hole or pad to check spacing. Do not specify Null pad definitions on internal layers of through-hole padstacks because those definitions cannot be connected.

For multiple drill padstacks (multi or plural vias), the overall array of the drill holes must fit within all of the individual pads.

! Do not use pad sizes equal to the hole size. Drilling is never perfectly centered, and off-center holes leave crescents of etch/conductor that can detach and move, causing shorts.

The bottom section displays the pad definition areas for *Regular Pad*, *Thermal Relief*, *Anti Pad*, and *Keep Out* for all conductor/etch layers and the Regular Pad definition for mask layers. A pad may be used on a routing layer or it may be used on a plane layer. For planes, based upon your design environment, the pad may be used on a negative plane or on a positive plane. Therefore, it is usually best to define all of the regular, thermal and anti-pad definitions for the Begin Layer, Default Internal and End Layer when creating the initial padstack. For each of these definitions, you must define the pad shape as circle, square rectangle, oblong, octagon, or shape. A shape (polygon) is used for any definition that is not a circle, a square, a rectangle, oblong or an octagon. A Shape symbol for the geometry of the pad must be created manually using the Symbol Editor.

<i>Select pad to change</i>	
-----------------------------	--

Pad Layer Table	<p>Sets the definition for pad geometries, sizes, flash symbols, and pad-shape symbols for the Regular, Thermal, and Anti-pads of the conductor layers of the padstack. The matrix table displays the basic definition of each field and also has cut and paste capability when creating or editing a padstack. Layer and Adjacent Layer Keepout figure and size definitions for the padstack are defined in the Padstack Editor tool for each pad layer entry. The keepout size can be defined in the library padstack definition or added/modified at the design level. The enabling of the keepouts is set within the Allegro X PCB Editor tool within the design database.</p> <ul style="list-style-type: none"> • Layer Name: Displays all the layers of your padstack, with the corresponding <i>Regular Pad</i>, <i>Thermal Relief</i>, <i>Anti Pad</i> and <i>Keep Out</i> definitions. You can edit any of the white fields. Begin and End layers are the outer layer pads: in a .brd file, the Top and Bottom; in an .mcml file, Surface and Base. Default internal is used for through-hole inner layer pad data. • Regular Pad: Specifies a positive pad with a regular shape flashed on positive layers only. Through hole pads require regular pad geometries be defined for every board layer. Surface mount pads require only Top and related Top mask information. Non-standard shaped pads are available as pad shapes and as pad flashes. • Thermal Pad: Specifies either a positive or a negative pad. A positive thermal relief connects pins to a positive copper area or to an embedded plane that prevents heat from concentrating near a pin or via during the unsoldering process. The thermal relief definition also may connect a pad to a copper area created on a routing layer, such as an external shield. A positive plane thermal relief comprises a "spoked wheel" pattern, which combines the regular positive pad, void areas, and tie bars, the latter two of which are defined in the Shape Parameters dialog box. A negative thermal relief is defined by a flash symbol used to connect pins to a negative copper area. The thermal relief may be plotted as a regular pad flash combined with the thermal values used as the shape-to-pad clearance.
<Pad Type> on layer <Layer>	<ul style="list-style-type: none"> • Anti Pad: Specifies a negative pad (clear, surrounded by black), where the size and type determine pad flashes for negative planes, or pad sizes for clearances when specified in positive shape parameters. Used to disconnect pins from a surrounding copper area. • Keep Out: Specifies an area around pad in which placing ETCH/CONDUCTOR, vias, or other symbols are restricted.
<Pad Type> on layer <Layer>	<p>Defines pad geometry type and dimension.</p> <ul style="list-style-type: none"> • Geometry: Specifies the standard shape of the pad. For anti pads and keep out pads, you can also choose <i>Flash</i> geometry. • Shape symbol: Specifies an irregularly shaped (custom) pad created with the Symbol Editor. You must specify the Shape symbol name (.ssm) file in the Geometry field. Click the browse button to open Library Shape Symbol Browser where you can either select a shape or click <i>Create New Shape Symbol</i> to create a new shape. • Flash symbol: Specifies the flash name referencing flash symbols displayed on WYSIWYG negative planes, or a user-defined name of an aperture for Gerber flashing of a unique pad shape. Define a flash symbol in the Symbol Editor if you are going to create a negative plane in your design. If you plan to use only positive planes, you do not need to create flash symbols. • Width: Specifies the width of the pad if it is a Square, Oblong, Octagon, Rectangle, or Flash. If the pad is a Circle, enter the diameter in the unit of measurement set for the padstack. If you specify a Square, Circle, or Octagon in the Geometry field, the value you enter here automatically sets the Height field to the same value. For Square and Octagon geometries, the specified size defines the horizontal distance between the two vertical sides/facets of the geometry. • Height: Specifies the height or diameter of the pad. If you specify a Square, Circle, or Octagon in the Geometry field, the value you enter here automatically sets the Width field to the same value. For Square and Octagon geometries, the specified size defines the vertical distance between the two horizontal sides/facets of the geometry. <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p>⚠ The width and height for circles, squares, octagons, n-sided polygon, are automatically the same. For example, if you enter 5 in this field, 5 appears in the width field.</p> </div>
<Pad Type> on layer <Layer>	<ul style="list-style-type: none"> • Diameter: Specifies the diameter for Circle, N-sided polygon pads. • Corner radius: Specifies the corner arc radius for Rounded rectangle pad geometries. • Corner chamfer: Specifies the corner arc radius for Chamfered rectangle pad geometries. • Offset: Specifies the x and y coordinates that indicate how far the pad origin shifts from the pad center (0,0) • Keep Out Exceptions: Creates keep out exceptions for <i>Clines</i>, <i>Pins</i>, <i>Shapes</i>, or <i>Vias</i> to keep them out from the keep out pads. • #sides: Specifies the number of sides for N-sided polygon geometry.

Positive Etch/Conductor Layer Pad Requirements

Padstacks requirements differ depending on whether artwork is positive or negative. Positive etch/conductor layers primarily use the regular pad. Pad shape symbols must be generated and used for positive regular pads when you require complex pad geometries. This solid shape displays, and DRC runs against the actual geometry. Pad shapes are limited to one solid shape, typically used for:

- SMD pads where corners must be chamfered,
- octagonal shaped pads when using versions prior to 15.5, or
- in a unique situation where pad boundary editing changes a pad geometry.

Flash names may also be used as regular pads. Pad flashes are used primarily as negative thermal reliefs and in limited applications when defined in the regular pad. (Prior to the support of pad shapes, pad flashes were used to generate irregular pad geometries in artwork.) Regular pads cannot contain voids and are limited to a single geometry. A flash is not limited to one outline and can contain multiple shapes, which may solve requirements for multi-shape pads on soldermasks or outer-layer footprints.

When using a flash, the display, DRC, and connectivity use the regular pad geometry and extents. The flash is a label. Ensure you specify a geometry and size that accommodates the size of the desired flash aperture. A multi-shape flash might not contain copper at the connect point (pin 0,0 location), and the pin may not physically connect if the pin center is not matched to copper in the flash. This application requires careful review of the connectivity in the board and in the artwork output.

In regards to pad flashes and artwork in positive data, if a regular pad is used, raster artwork searches PSMPATH for flash geometry (.fsm) file data. The flash name supersedes the geometry definition in the padstack if you chose a flash name in the padstack. The shape data in the flash symbol is embedded in the output artwork file, and displays when you run *File – Import – Artwork* ([load photoplot](#) command). Vector artwork, however, searches for a matching flash name in the aperture list. It does not use the library flash symbol. Running *File – Import – Artwork*, where vector artwork contains flashes in the aperture file, searches ARTPATH for the flash geometry (.fsm) file data. If found, the etch/conductor geometry displays; otherwise, a triangle with the flash name displays as an indicator of an aperture flash. The actual flash geometry must be provided with the aperture file to the vendor generating the artwork.

Negative Etch/Conductor Layer Pad Requirements

Negative layers use shapes for connectivity. Based on the pad information from the padstack's thermal relief and anti-pads definition, the shapes are drawn on the etch/conductor subclass, assigned to a net, and flash the thermal pad shape. Negative planes should contain shapes that reference, display, and DRC check to the thermal relief and anti-pads geometries defined in padstacks. With no expectation of autoavoiding, the intent is to quickly generate power and ground planes without processing all DRC rules, and instead create clearances-based padstack data.

To visually determine the sizes of thermal and antipad that artwork flashes or embeds as apertures (depending on artwork output type), enable *Thermal Pads* on the *Display* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* ([prm](#) command).

Older databases using the older pad flash process, in which the variable `old_style_flash_symbols` is enabled in the *Misc* category of the User Preferences Editor, available by choosing *Setup – User Preferences* ([enved](#) command), draw a thermal pad using the geometry and extents of the thermal pad listed in the padstack. The flash name is used for artwork, and its geometry is not displayed. A crosshair centered in that pad indicates connectivity. New databases using this old style methodology display this geometry and not the information in the flash symbol itself.

New databases (in which the `old_style_flash_symbols` variable is disabled) draw the actual flash symbol .fsm geometry when you enable *Thermal Pads*, using true display of flashes and older data that has been updated to exploit this feature. See the [flash_convert](#) utility.

When the negative planes setup is correct and contains flash names for thermal pads, artwork output will not differ between the old and new style pad flashes. Old and new style flashes require a flash symbol for raster artwork to embed in the output file. The flash is either in the .brd file for new style flashes, or in the PSMPATH library path for old style flash data. Artwork fails when no flash appears in the PSMPATH library path, and the following error appears in the photoplot.log:

```
ERROR: aborting film - undefined aperture symbol - cannot continue
```

```
Load photoplot will read and display the raster artwork flashes.
```

Vector artwork relies on the aperture file to match pad flashes. Therefore, old and new style flash data write identical artwork output files. When you run *File – Import – Artwork*, ([load photoplot](#) command) PSMPATH is searched for flash symbols. If found, the flash draws that geometry; if not, a triangle figure is placed as the flash indicator and text containing the flash name inserted.

Negative pads in artwork

If thermal flash names are not defined in padstacks, warnings and errors occur if flash names are missing from the thermal relief when processing negative layers. Artwork expects a pad flash in this case. Without a flash name, raster artwork uses the regular pad size and issues this warning in the photoplot.log:

```
WARNING: No thermal flash for padstack "PADNAME", regular pad used.
```

Without a flash name, vector artwork reports this as an error:

```
PADSTACKS MISSING THERMAL/ANTIPAD DEFINITIONS:
```

VIA

PAD60CIR36D

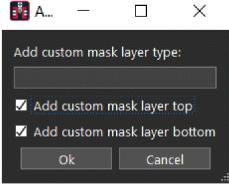
In both cases, Cadence advises adding a thermal flash name to the padstack.

All other pads on a negative layer flash the antipad. Positive shapes create voids based on the thermal and antipad sizes if DRC is not selected as the voiding rule. The true intent of these pads is for negative layer flashes. Therefore, Cadence does not recommend using thermal and anti pads for positive shape voiding but rather DRC rules. The thermal relief can use any standard geometry, or point to type Flash, where a library flash symbol representing the true thermal geometry can be assigned.

This can be used in old style flash symbol mode where raster artwork uses the geometry in the artwork file, but the display contains just cross hair indicators for thermal connectivity. In new style flash symbol mode, the display, negative plane DRC, and the artwork file integrate the true flash geometry in all areas. Flash symbols allow multiple shapes, but no voids. These will not flash on positive layers, nor display on positive layers.

Non-plated holes can be defined with small spotting pads that are drilled away during fabrication, or the pad can be defined as null, meaning no pad will be drawn. Pads add benefit when dimensioning or using tools to locate the pins for viewing or checking. Adding adequate route keepout areas around these small pads prevents etch/conductor clearance issues. Dynamic shapes void around these large areas. Using keepout areas for DRC clearances on non-plated holes is recommended in conjunction with voiding to DRC rules.

Mask Layers Tab

Select pad to change	Defines mask pad geometries, sizes, flash symbols and pad-shape symbols. Multi-shape flash pads may be associated with any mask layer. The Select Pad to Change table displays the basic definition of each field and also has cut and paste capability when creating or editing a padstack mask definitions.
Pad Layer Table	Sets the definition for pad geometries, sizes, flash symbols, and pad-shape symbols for the Regular, Thermal, and Anti-pads of the conductor layers of the padstack. The matrix table displays the basic definition of each field and also has cut and paste capability when creating or editing a padstack. Layer and Adjacent Layer Keepout figure and size definitions for the padstack are defined in the Padstack Editor tool for each pad layer entry. The keepout size can be defined in the library padstack definition or added/modified at the design level. The enabling of the keepouts is set within the Allegro X PCB Editor tool within the design database.
Add Layer	Adds custom mask layers. The form that appears enables the creation of additional user-defined mask layers for a maximum of 16 layer pairs (Maximum of 32 layers total). The layers can either be added. Enabling <i>Add custom mask layer top</i> and <i>Add custom mask layer bottom</i> options add custom mask layers for top and bottom layers, respectively. 
Delete Layer	Deletes user-defined layers. Available only if you select a user-defined layer.
<Pad Type> on layer <Layer>	Defines pad geometry types and dimensions.

Options Tab

Suppress unconnected internal pad; legacy artwork	Enable to suppress unused inner layer pads, thereby making them unavailable for DRC, routing, and display, with the <i>Suppress Unconnected Pads</i> option on the <i>Film Control</i> tab of the Artwork Control Form, available by choosing <i>Manufacturing – Artwork</i> (film param command) or with dynamic unused pad suppression by using the xsection command. Otherwise, you cannot suppress any pads during photoplotting.
Lock layer span	Prevents layer expansion for BBVias when a new signal/plane layer is inserted. For example, a BBVia exists in a design that starts at layer_4 and ends on layer_5. A new layer, say Layer_4A, is inserted between layer_4 and layer_5. The via will start on layer_4, but end on Layer_4A.

Summary Tab

The summary tab displays type of padstack, design unit, drill data, design and mask layer pad definitions along with usage options and date of creation. This information can be saved to a file in HTML format, or printed.

Related Topics

- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Padstack Editor: Menu Commands

The commands on the Padstack Editor menu bar differ depending on whether you started the Padstack Designer as a standalone tool (using the `pad_designer` command) or from within a design (using the `pateditdb` or `pateditlib` command).

File – New	Displays the New Padstack dialog box where you can define a new padstack or edit an existing one using the following fields: <i>Directory</i> : Displays the current working directory. <i>Padstack Name</i> : Enter a name for the new padstack. A padstack filename can consist of the characters A to Z, 0 through 9, dash (-), and underscore (_). <i>Browse</i> : Click to choose an existing padstack from the file browser that appears. Doing so overwrites the existing padstack. <i>Padstack Type</i> : Select a padstack type. <i>OK</i> : Click to create a new padstack. <i>Cancel</i> : Closes the New Padstack dialog box without creating a padstack. If you are currently editing a padstack, you are asked if you want to save it. When you have saved the current padstack, the new file browser appears with the filter set to *.pad.
File – Padstack Library Browser	Displays <i>Library Padstack Browser</i> to view the parameters of padstacks available in the pads library. The dialog box displays a list of all padstacks present in the library under <i>Select a padstack from the list</i> . You can filter the name of padstack using <i>Name filter</i> and <i>Type filter</i> . When you choose a padstack in the list on the left of the dialog box, the graphic preview of the padstack is visible in the right of the dialog box.
File – Import XML	Creates a padstack from the data specified in an XML (.pxml) file. The padstack data is loaded into Padstack Editor and errors/warnings are displayed. The default template of padstack XML type definition file <code>cdn_padstack.dtd</code> is available at <code><installation_hierarchy>\share\pcb\xml-formats</code> . You can find other examples of padstack XML files at <code><installation_hierarchy>\share\pcb\examples\padstack_xml</code> . For more information, see Managing Padstack Data Using XML Files .
File – Export XML	Exports the current padstack definition to the specified XML (.pxml) file. If padstack has errors and warnings, then you cannot export the padstack data.
File – Update to Design	Available only when Padstack Designer is invoked within a design using <code>pateditdb</code> or <code>pateditlib</code> . This command also has a toolbar button for quick access. Adds the current padstack definition back to the design, if there are no errors. If the padstack is new, it is added to the design. If the padstack already exists in the design, the design padstack is updated with the new information from the Padstack Designer. If there are errors in the padstack, the update fails and the errors are displayed in the Editor's status area. Note: This command does not save the new definition. Choose <i>File – Save to File</i> or <i>File – Save As</i> to save the data.
File – Recent Padstacks – Recent File List	Available only when Padstack Designer is invoked as a stand-alone tool. Opens a list (up to 20) of your most recently used (MRU) padstacks with their specified path names. The default number of padstacks is 10. When you choose a padstack from the list, the editor opens the file and changes the current directory to the directory where the specified padstack resides.
File – Save	This command appears in the Padstack Designer when you open the application in standalone mode. Saves the current padstack to your current working directory. If there are errors in your padstack, you are warned about them. If a padstack of the same name already exists in your working directory, you are warned that the padstack file will be overwritten. If you do not want to overwrite the padstack, choose <i>File – Save As</i> to save the padstack with a different name.
File – Save As	Saves the current padstack definition under a different name. When you choose this option, the current padstack is checked for errors and the <i>Save As</i> browser displays. Enter the new name in the filename field. If a padstack already exists with the same name, you are warned that it will be overwritten. Enter a unique name to avoid overwriting another file.
File – Check	Checks the current padstack definition for any potential problems. Any problems are displayed in the Pad Stack Warnings window. ⚠ This command does not save the padstack.
File – Properties	Lets you set an optional password-protected lock from the File Properties dialog box so the file is marked as read-only in the database. For details, see File Properties Dialog Box in the <code>file_property</code> command.
File – Script	Available only when Padstack Designer is invoked as a stand-alone tool. Starts the scripting process to record or replay padstack scripts. Recording a script lets you automate the padstack definition process. Replaying a script cuts the time that is involved when you define new padstacks that share similar definitions.
File – Exit	Exits the Padstack Designer. You are prompted to save all data and update the design.

Related Topics

- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Prerequisites to Defining New Library Padstacks

- Check your manufacturer specification sheets and verify your design requirements.
- Gather the dimensions, physical data, logical data, manufacturing requirements, and documentation requirements for the padstacks that you want to define.
- Check that your padstacks are not already in the library.
- Determine your photoplot requirements, such as flash names, NC Drill data, and offset requirements.
- Create any custom (unique) pad shapes that you need.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor Tasks](#)

Padstack Definition Guidelines

- Avoid specifying Null pad definitions on internal layers of through-hole padstacks.
- Define a pad size for every internal layer so that design rule checking (DRC) checks the line-to-pad spacing. DRCs occur on line-to-pad spacing only if pad definitions exist on internal layers. Therefore, an etch/conductor layer can be placed in the path of a drill hole without being flagged by DRC.
- Define the pad size to be smaller than the drill hole, so that the pad is drilled out during manufacturing, and the DRC can verify the spacing. As long as a pad is defined, the DRC uses the larger of the drill hole or pad to check your spacing.
- Define all layers through which the padstack passes for editing when you define blind/buried padstacks. Because undefined layers do not reveal information about the objects on them, it may appear that data is lost because the layer is undefined.

⚠ When creating a blind or buried padstack, layers to be excluded must have their pads removed. Pad information must exist on a given layer to be considered drilled to that layer. The criterion for connectivity is the existence of a pad definition on the subclass. The connectivity model for any padstack reads all/any type pad information defined in the padstack, and both the model and the add connect command interpret the existence of a pad of any type (thermal, antipad or regular) as a legal layer for a connection. When an outer subclass is missing a regular pad, it is still considered a thru drill when it contains some pad information. If all pad data is missing, then the layer is not considered drilled, and it is a candidate for blind/buried via status. No connection is available on layers with no pad definitions.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)

Padstack Editor Tasks

The following padstack procedures are addressed:

- [Starting Padstack Editor](#)
- [Defining Padstacks](#)
- [Displaying Derived Padstack Names](#)
- [Modifying Padstacks in a Library](#)
- [Modifying Padstacks in the Design](#)
- [Purging Unused Padstacks](#)
- [Recording and Replaying Padstack Scripts](#)
- [Viewing Padstack Instances](#)
- [Creating or Modifying Padstacks from XML](#)
- [Exporting Padstack Definition to XML file](#)

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)

Starting Padstack Editor

Padstack Editor can be used independently of other Cadence tools, or can be launched from them.

As a stand-alone tool:

On Windows

From the Start menu, *Cadence PCB Utilities 17.4-2019 – Padstack Editor 17.4*

or

1. Click *Start – Run*
2. Change directory to <installation_directory>/tools/bin and enter the following:

```
padstack_editor.exe
```

The Padstack Editor dialog box appears.

On a Unix Workstation

1. From your system prompt, type the full path name to the directory in which your Cadence tools are installed, and invoke the Padstack Editor:

```
padstack_editor
```

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Defining Padstacks

Whether you are defining a padstack for your design, your library, or both, you must set both the padstack parameters and the pad definitions for your padstack layers. Follow these steps to define a padstack:

1. Select *File – New* from the Padstack Editor main menu.
The *New Padstack* dialog box displays.
2. Enter a name for the new padstack. A padstack filename can consist of the characters A to Z, 0 through 9, dash (-), and underscore (_).
 - or– Click *Browse* to choose an existing padstack from the file browser that appears and click *OK*.
 - or– Click *Template* to create a new padstack based on a template containing default .pad definition information and click *OK*.
3. Specify the definitions in the tabs as explained in [Padstack Editor Tabs and Panels](#).

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Displaying Derived Padstack Names

You can use the following methods to display the derived padstack names in your design:

Method 1

1. Run `pateditdb` in your tool user interface.
The *Options* panel reconfigures the parameters.
2. Click *Derived*.
The derived padstacks in your design appear in the Available Padstacks listing.

Method 2

1. Use the Constraint Manager.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Modifying Padstacks in a Library

To modify a padstack, perform the following steps:

1. Run [pateditlib](#).
 2. Choose the library padstack that you want to modify from the file browser and click OK.
The Padstack Designer dialog box appears with the padstack definition loaded. The banner of the Padstack Designer lists the name of the padstack that you are modifying.
 3. Edit the padstack parameters and layers as appropriate.
- To save the padstack to the library, choose *File – Save to File* to overwrite the current padstack definition with the newly modified one, or choose *File – Save As* to specify a new padstack name.
 - To load the padstack into your design, choose *Update To Design* from the toolbar or the *File* menu. This option is available only if you invoke the Padstack Editor from your tool's user interface instead of using the Padstack Editor as a standalone program. For more information, see [Modifying Padstacks in the Design](#), below.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Modifying Padstacks in the Design

You can edit both the padstack definitions and the padstack instances in the design that you are currently working on. To modify padstacks:

1. Run `pateditdb`.
2. Click on the *Options* panel of the Control Panel.
3. Click *Definition*.
4. Choose the padstack definition that you want to edit from the list in the *Options* panel.
The padstack information appears in *Name* field.
5. Click *Edit*. You can also choose *Edit* by right-clicking in the design window and selecting it from a pop-up menu.
The padstack designer opens with the padstack definition loaded.
6. Modify the padstack parameters and pad layers as necessary.
7. Choose *File – Update to Design* from the Padstack Designer to update all of the padstacks in the design that have the current name. DRC executes on all of the changes to your padstacks in the layout. All of the padstacks in the design refresh unless errors exist in the padstack definition, in which case, a warning message appears and you can correct the errors in the Padstack Designer. If the padstack exists in the design (that is, you have not renamed the padstack), you are prompted to overwrite the existing padstack.

You can also choose *File – Save* from the Padstack Designer to save your padstack data to a library.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Purging Unused Padstacks

You can remove unused padstacks from the list of available padstacks for your design. Purging unused padstacks increases the performance of the editor because the program must load all padstacks, used or unused, load in a design. To purge unused padstacks from the design, perform these steps:

1. Run `pateditdb`.
2. Choose the *Options* panel on the ministatus.
3. Click *Purge* and then choose one of the following options:
 - a. Click All.
If your design contains any unused padstacks, the Unused Padstacks window lists the unused padstacks. A window appears, asking if you want to purge the unused padstacks.
 - b. Choose one of the following:
Yes to purge the unused padstacks from the list of available padstacks and close the window.
No to leave the unused padstacks in the list of available padstacks and close the window.

To delete all derived padstacks

- a. Click Derived.
If your design contains any unused derived padstacks, the Unused Padstacks window lists the unused derived padstacks. An Error window appears, asking if you want to purge the unused derived padstacks.
- b. Choose one of the following:
Yes to purge the unused derived padstacks from the list of available padstacks and close the window.
No to leave the unused derived padstacks in the list of available padstacks and close the window.
If your design does not contain any unused padstacks, the Padstack Selection dialog box displays a message with this information.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Recording and Replaying Padstack Scripts

You can automate the process of entering padstack data by creating a script that lets you record the entries that you make on the Padstack Designer dialog box. When you want to define new padstacks that share similar padstack specifications, you can replay the script file and edit the new padstacks as necessary. See [script](#) for more information.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Viewing Padstack Instances

When you edit layout padstacks, and you start the Padstack Designer from within your tool user interface using [pateditdb](#), you can view a report that shows all of the pins and vias that currently use the padstack you are editing. To view padstack instances:

1. Run [pateditdb](#) to choose a design padstack.
2. Click *Edit* in the Options panel of the Control Panel.
The Padstack Designer appears.
3. Choose *Reports – Show Instances*.
A window displays all the pins and vias in your design that currently use the padstack that you are editing.
4. Choose *File – Save* to save the listing with the default name of *pin_instances.txt*.
5. Click *Close* to close the window.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Creating or Modifying Padstacks from XML

To transfer a padstack data defined in a format other than .pad file, follow these steps:

1. Select *File – Import XML* from the *Padstack Editor* main menu.
The *Import pXML* dialog box displays.
2. Browse padstack definition file (.pxml) and select *Open*.
The padstack information is imported in *Padstack Editor* and checks are run. The errors are displayed in the *Padstack Errors* dialog box.
3. Open *Summary* tab to verify the padstack.
4. Choose *File – Save* to save the padstack data.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

Exporting Padstack Definition to XML file

To modify the padstack data outside *Padstack Editor*, you can export the padstack definition in an XML file and save it to an external directory. To do this task from *Padstack Editor* UI, following steps are required:

1. Select *File – Open* or *File – Padstack Library Browser* to select and open the padstack in *Padstack Editor*.
2. Choose *File – Export pXML* from the main menu.
The *Export Padstack to pXML* dialog box appears.
3. Browse the location to save the padstack definition in pXML file (.pxml). Ensure that *Change Directory* option is enabled.
4. Click *Save* to export the padstack data.

Related Topics

- [Padstack Editor Tabs and Panels](#)
- [Padstack Editor: Menu Commands](#)
- [Prerequisites to Defining New Library Padstacks](#)
- [Padstack Definition Guidelines](#)
- [Padstack Editor Tasks](#)

pad to shape

The `pad to shape` command copies selected pins, vias, or fingers to shapes on a different layer.

Related Topics

- [Copying Selected Pads to a Different Layer](#)

Pad to Shape Command: Options Panel

Access Using

- *Menu Path: Tools – Convert – Pad to Shape*

<i>Pad layer</i>	Specifies the layer of the selected regular pad.
<i>Shape layer</i>	Specifies the layer for shape creation.
<i>Delete existing shapes</i>	Deletes all existing shapes in the specified Shape layer.
<i>Expansion/Contraction</i>	Specifies the expansion or contraction. You can edit this value.
<i>Create dynamic shapes</i>	Specifies that the shapes are to be dynamic. Selected by default.
<i>Assign pad net to shape</i>	Specifies that pad nets are to be assigned to the generated shapes. Selected by default.
<i>Generate Shapes</i>	Generates shapes for the selected pads based on the configurations specified in the Options pane.

Copying Selected Pads to a Different Layer

Follow these steps to copy selected pads on a different layer in your design:

1. Choose *Tools – Convert – Pad to Shape*.
2. Configure the Options pane.
3. Select pads (pins, vias, or fingers) to convert.
4. Click *Generate Shapes*

Related Topics

- [pad to shape](#)

panel links

This command is not supported in current versions of Cadence design tools.

panel setup

This command is not supported in current versions of Cadence design tools.

parallel

The `parallel` batch command analyzes interconnection parallelism between pairs of xnets on a design. Controlling interconnection parallelism provides crude control over crosstalk. The `parallel` command examines the spacing between etch segments and checks for potential crosstalk problems caused by segment edges that run parallel for too great a distance.

Since crosstalk decreases as the distance between parallel lines of etch increases, a separation window is applied to all segment pairs. Segment pairs whose separation is greater than this distance window are assumed to have negligible crosstalk and, in order to reduce processing time, are not examined.

Embedded etch planes are considered to have a shielding effect. Thus, segments that are separated by an intervening embedded plane (defined as a shield layer on the design *Cross Section* form) are not examined.

For segment pairs that fall within the separation window, and that are not separated by an embedded plane, the total length of parallel segment pairs between two xnets can not exceed a specified limit. This limit is user-defined, based on user expertise on crosstalk occurring for this design technology and style of layout.

Parallelism differs from DRC spacing rules in that, while DRC rules apply to individual segments, parallelism rules apply to the total parallel length accumulated between xnets. The Parallelism program checks for both same layer parallelism and layer to layer parallelism.

Xnets crossing each other perpendicularly on adjacent layers or running parallel for extremely short distances do not cause significant crosstalk. Thus, the parallel program tries to expose long accumulations of geometric parallelism.

Parallel performs calculations for entire xnets. Thus, it does not take into account the specific positions of driver pins, receiver pins, and terminators.

A parallelism report should be created after the design has been routed, though a complete routing is not required. Between automatic routing runs, you can reschedule xnets, manually route, and/or change route parameters. This report provides an effective measure of the impact of glossing routines that spread out closely packed etch to fill available space.

Parameters used to control parallel processing include:

- Length rules

Generally, some degree of parallelism is acceptable. Designers often establish rules that govern the maximum length of parallelism. The parallel length rules, when adhered to, are usually sufficient to prevent the majority of crosstalk problems. Two length rules are observed:

- Minimum parallel length between any two xnets (*length rule*).
- Minimum parallel length between any xnet and all other xnets (*all_other rule*).

- Separation window

Separation windows are used for determining whether or not two connect lines are parallel for a predefined distance. This can be visualized as a rectangular window of a user unit-defined width, within which two line segments are determined to be parallel.

- Layer spacing

The amount of space between layers of a design affects crosstalk. Proximity of layers also affects the amount of crosstalk resulting between segments on the same layer.

Crosstalk algorithms use the height above the nearest embedded plane as a parameter for calculating crosstalk between same-layer segments. Such considerations are beyond the scope of a simple parallelism report, so ground plane separation are not a factor.

- Layer Ranges

Separation on surface layers has a different effect on crosstalk than it does on internal layers. You can have different rules for different layers. For this reason, a single layer, or a range of layers can be examined. Segments on other layers are overlooked on a segment-by-segment basis.

 The editor starts the top layer numbering at zero.

- Netlists

A specific set of xnets can be examined by using the Define – List command to create a file of the chosen xnet names. The report generator attempts to use the first word (up to 32 characters in length) of each file line as a xnet name. An error message is inserted into the report for any listed xnet names not found in the design.

Syntax

```
parallel [-w|-l|-a|-f|-r|-t|-s] [-version] [-versionlong] [drawingFile] [reportFile]
```

-w	Specifies the separation window. The default, in user units, is 100 mils. Separation windows are used for determining whether or not two connect lines are parallel for a predefined distance. This can be visualized as a rectangular window of a user unit-defined width, within which two line segments are determined to be parallel.
-l	Specifies the minimum parallel length between any two xnets. (<i>length rule</i>). The default, in user units, is 5000 mils.

-a	Specifies the minimum parallel length between any xnet and all other xnets (<i>all_otherrule</i>). The default, in user units, is 20000 mils.
-f	Specifies the name of a "from" netlist file. The default is none . By default, all xnets in a design are examined. You can specify the name of a file containing a list of xnet names to be checked. Analysis is restricted to this list. Each xnet is checked for parallelism against every other xnet in the drawing, including those not in the list. In other words, parallelism is measured from each listed xnet to every other xnet in the design.
-r <start>-<end>	Specifies the start and end layers of the board to be analyzed. These are the numeric layer numbers.
-t	Specifies the title of the report. The default name is Parallelism report for <design>.
-s	Specifies the progress message when the report is generated.
-version	Prints the version and exits.
-versionlong	Prints the long version and exits.

Related Topics

- [Analyzing Interconnection Parallelism](#)

Interpreting Results

For each xnet, at least a summary line is printed. This gives the total parallel length of segments from all other xnets that fall within the separation window. This provides a worst-case figure of merit, since it is doubtful that all other xnets might encounter an identical signal transition at the same moment in time. Xnets that are members of multi-signal busses, however, can be subject to simultaneous transitions of a number of other xnets on the bus. For these, the all-other summary lines become a more relevant statistic.

In general, forward crosstalk is largely a function of parallel length, and backward crosstalk is a function of minimum separation. The separate reporting of these parameters allows you to, for example, ignore parallel length for analyses of internal layers, where forward crosstalk is minimal, due to relatively constant wave propagation velocity.

The minimum separation parameters are distinguished from the DRC spacing checks in that segments on different layers are considered. DRC typically makes same-layer checks only; however, segments on adjacent layers that are exactly parallel in the z-axis will produce DRCs unless they are separated by a shield layer. Due to the manner in which z-axis auditing of parallelism occurs, the results (DRC detection) may not appear to be consistent; therefore, before conducting a z-axis audit, you should take into account the following:

- Are only layers that are directly adjacent to each other being audited? This condition is defined as two etch layers separated by a single dielectric layer.
- What is the thickness of the dielectric layer?
- Does a shield layer separate the conductive layers being audited?

Your determination of these conditions will influence the outcome of the z-axis audit. This becomes clear when you consider how a parallelism gap for two segments that are not on the same layer is computed:

```
gap = sqrt (xygap*xygap + zgap*zgap)
```

where xygap is the gap in the x or y dimension and zgap is the separation between layers

The following examples illustrate different scenarios:

Example 1

Two connect lines (clines) are positioned in exactly the same xy axis on adjacent layers TOP and INT1 (therefore, the parallelism gap will simply be the zgap value). The xnets to which the clines are attached are assigned to the same ECset that has a parallelism rule of 1000 mil length at a gap of 30 mils. In this example, a DRC is generated between the two xnets because the gap between the clines is the thickness of the dielectric (30 mils) between layers TOP and INT1. If you change the parallelism constraint to a 29 mil gap, DRC for the zgap value is eliminated.

Example 2

The cline on layer INT1 in the first example is moved to INT2, and the parallelism constraint is set to a gap of 55 mils. The zgap value between the two xnets is now:

TOP - INT1 dielectric thickness	=	30 mils
INT1 thickness	=	1.2 mils
INT1 - INT2 dielectric thickness	=	23.0 mils

Total thickness	=	54.2 mils

The difference between the thickness and the gap constraint (0.8 mils) results in a DRC. If you change the parallelism constraint to a 54 mil gap, DRC for the zgap value is eliminated.

Example 3

The cline on layer INT2 in the second example is moved to BOTTOM/BASE, and the parallelism constraint is set to a gap of 200 mils. This increases the zgap between the xnets to 79.6 mils (the sum of the thickness of the dielectric and conductive layers between TOP/SURFACE and BOTTOM/BASE. Despite the disparity between the actual thickness and the gap constraint (123.4 mils), no DRC is generated because there is a shield layer separating the top and bottom layers, therefore parallelism between these layers is not considered.

Command Line Options

Switch settings can be manipulated to achieve various results:

- To print an all-other summary, set the length rule to a very large number and the all_other rule to zero.
- To report parallelism only for same-layer segments, set the layer spacing larger than the separation window.
- To exclude power and ground xnets from the parallelism report, prepare a list file of all xnets.
- Delete the power and ground xnets from the file using an editor. Use the `-f` option to report parallelism for the xnets in this file.
- To find segments on different layers that are not orthogonal, set the separation window to zero and all the cross section thicknesses to zero. It can be necessary to use the layer range option to limit the analysis to two layers.

Setting very short length rules or very wide separation windows can result in very long execution times and very long report files. Setting the length rule to 0 results in approximately N-squared violations. for example, a design with 1,000 xnets could produce a report file with 1,000,000 lines.

Analyzing Interconnection Parallelism

Follow these steps to analyze interconnection parallelism:

1. Type `parallel` at your operating system prompt.
You are prompted to enter a design file name.
2. Enter the name of the file.
You are prompted to enter parameters for each of the program's processes.
3. Enter a value for each process or press the Enter/Return key to accept the program defaults
4. When you have completed step 3, the program runs.

Related Topics

- [parallel](#)

param in

The `param in` command imports a database parameter file (`.prm`) containing customized parameters from a central location in a corporate library, or from your local working directory into another design for reuse. The environment variable `PARAMPATH` determines the path of library-based files. By locating parameter files in a centralized directory, you promote the sharing of design settings across similar designs.

Customized parameters are those that have a single instance in the database and include global values such as dynamic fill; grid settings; artwork format; and `Xhatch style`, `line width`, `spacing`, and `angle`. You create the `.prm` text file with the *File – Export – Parameters* (`param out` command).

If a parameter record of the same name already exists in the design database, the `.prm` file overwrites the existing record when you import. When no parameter name exists, a new record is created.

The `techfile` batch command can also be used to import a database parameter file.

Related Topics

- [Select Parameter File browser](#)
- [Importing a Database Parameter File](#)

Import Parameter Dialog Box

Access Using

- *Menu Path: File – Import – Parameters*

<i>Input Parameter File</i>	Enter the name of a file containing parameter values, or click ... to locate existing database parameter files in the current working directory.
<i>Library</i>	Click to display the Select Parameters File dialog box, which contains all database parameter files in the directories defined in the PARAMPATH environment variable in the User Preferences Editor, available by choosing <i>Setup – User Preferences</i> (enved command).
<i>Import</i>	Click to import parameters into the current design.
<i>Close</i>	Closes the dialog box and exits the command.
<i>Viewlog</i>	Click to view error messages and other process information in the current or last generated <code>param_read.log</code> report.

Related Topics

- [Importing a Database Parameter File](#)

Select Parameter File browser

To choose a file, type the name in the search field, or highlight it in the list box, and click the OK button.

To narrow the list, enter a search string in the search field and click the OK button. The asterisk (*) displays the complete list.

Database and Library are permanently greyed out.

Selecting a database parameter file and choosing OK, or double-clicking the file brings up the Import Parameter dialog box.

Related Topics

- [param in](#)

Importing a Database Parameter File

To import a database parameter file, follow these steps:

1. Open the design into which you want to import a database parameter file.
2. Run the `param in` command. (Ensure you have created the `.prm` file by executing *File – Export – Parameters* (`param out` command.)
3. Enter the name of a file containing parameter values in the *Input Parameter File* field, or click ... to locate existing database parameter files in your local directory. Click *Library* to display the Select Parameters File dialog box, and access database parameter files in centralized directories defined in the `PARAMPATH` environment variable.
4. Click *Import* to read the chosen `.prm` file into your design.
5. Click *Viewlog* to view error messages and other process information in the current or last generated `param_read.log` report.

Related Topics

- [param in](#)
- [Import Parameter Dialog Box](#)

param out

The `param out` command creates a database parameter (`.prm`) file containing customized parameter records from a design. Customized parameters are those that have a single instance in the database and include global values such as dynamic fill; grid settings; artwork format; and Xhatch style, line width, spacing, and angle, for instance.

You then use *File – Import – Parameters* (`param in` command) to import the values from `.prm` files centrally located in a corporate library, or in your local working directory into another design for reuse.

The `techfile` batch command can also be used to export a database parameter file or individual parameters you specify.

Related Topics

- [Exporting Database Parameters](#)

Export Parameters Dialog Box

Access Using

- *Menu Path: File – Export – Parameters*

<i>Output file name</i>	Enter the name of a file to which to save database parameter records, or click ... to locate an existing database parameter file in the current working directory.	
<i>Available Parameters</i>	Displays a list of the possible parameter records available for export. If no parameter record exists in the database, it does not display. Click to include the parameter group in the <code>.prm</code> file.	
	<i>Design Setting</i>	Global values and grid settings
	<i>Artwork</i>	Artwork film definitions
	<i>Color Layer</i>	Priority in which layers are drawn
	<i>Color Palette</i>	Color parameters and color table
	<i>Color Component</i>	Custom color for symbol instance of the component
	<i>Color Profile</i>	Custom color for wire profile group
	<i>Color Net</i>	Net custom color and states. When a file containing net color data is imported into any design, only the nets that exist in that design are read; the rest are ignored. Net color assignments are not overwritten, but rather incremented. To completely replace net color assignments, click <i>Clear All Nets</i> in the <i>Nets</i> section of the Color dialog box before importing a file containing net color data.
	<i>Text Size</i>	Text size settings
	<i>Application or Command Parameters</i>	All other supported parameters, including those for auto rename, auto assignment, auto silkscreen, global dynamic fill, autovoid, export logic, drafting, gloss line flattening, gloss dielectric generation, <i>Options</i> panel settings, test prep, automatic placement, auto swap, thieving, backdrill, interactive flow planner (Allegro only), and Signoise analysis.
<i>Select All</i>	Choose to include all available parameters in the <code>.prm</code> file.	
<i>Clear All</i>	Choose to remove all available parameters from the <code>.prm</code> file	
<i>Export</i>	Click to create a <code>.prm</code> file based on the chosen parameters.	
<i>Close</i>	Closes the dialog box and exits the command.	
<i>Viewlog</i>	Click to view error messages and other process information in the current or last generated <code>param_write.log</code> report.	

Exporting Database Parameters

To generate a database parameter file, follow these steps:

1. Run the `param out` command. The Export Parameters dialog box appears.
2. Enter the name of a file to which to save database parameter records, or click ... to locate an existing database parameter file.
3. Click *Viewlog* to view error messages and other process information in the current or last generated `param_write.log` report.
4. Click *Close*.

Related Topics

- [param out](#)

parasitics

The `parasitics` batch command calculates the capacitance between any two etch/conductor elements, including connect lines, filled rectangles, or shapes. Capacitance to shapes is based on a rectangular approximation to the shape area. For example, voids and cutout detail are not precisely evaluated. The shape capacitance to the enclosing rectangular area is always pessimistic (that is, the voids which are ignored slightly reduce the capacitance).

To provide a fast estimate, the reported capacitance values are based upon a parallel plate formula. More accurate capacitance values can be obtained by using the field solver.

You can also run `show parasitic` from the so command window prompt.

Access Using

- *Menu Path: Display – Parasitic*

Syntax

```
parasitics [-s|-c|-f|-v|-g] input_design.brd [output.brd]
```

-s	The separation window, in user units. The default is 0.002 meters. Capacitance between items separated by more than this window is assumed to be zero and is ignored.
-c	Capacitance threshold in picofarads. Ignores net pairs whose total capacitance is less than this value. The default is 1pf.
-f <filename>	The name of a netlist file (defaults to analyze all nets). Nets not in the list appear in the report whenever they have sufficient capacitance to a second net which is in this "from" list.
-v	Specifies that vias have a defined capacitance (measured in picofarads) - otherwise the editor estimates the capacitance using the cylindrical size of the via.
-g	The attempt to map output net names into their associated GED schematic net names. Note that the schematic <code>pstchip.dat</code> , <code>pstxprt.dat</code> , and <code>pstxnet.dat</code> files may be unassociated if they are in the current working directory. If the net name mapping fails, net names are still output. The <code>-g</code> option also changes the output capacitance format from Farads to picofarads, for example, <code>1.23e-12</code> to <code>1.23pF</code> . For additional information, refer to the comments incorporated into the <code>prs2ged</code> script regarding adding parasitic capacitors into the schematic. All nets are analyzed unless you identify a file as a list of net names. The name of the default output report file is the name of the design with the <code>.prs</code> extension.

Calculating Capacitance

Perform this step to calculate capacitance:

1. Type `parasitics` with the appropriate options at your operating system command prompt, and press Enter/Return.
The program calculates capacitance between nets and lists the results in the output file.

partition

The `partition` command divides physical areas of the design database into separate sections, or partitions, to allow several designers to collaborate and expedite the design schedule. The master designer partitions a design and assigns each designer a partition, and then exports the design to multiple designers. Board partition files have an extension of `.dpf` (Design Partition File); APD partition files have the extension of `.dpm`. Two pre-defined partitions—`SILKSCREEN_TOP_BOTTOM` and `DIMENSION_DRAFTING`—are created automatically that cannot be changed or deleted.

Each designer has a copy of the database and works on a partition independently. The designer on a specific partition is restricted to that partition and works with a limited command set. However, the master designer can designate certain nets as `soft` meaning the partition designer can pick and route these nets even if they extend beyond the boundary of the active assigned partition. Soft nets are highlighted in the owner's partition database but are dimmed and read-only in all other partitions. The master designer manages soft net assignments using *Place – Design Partition – Soft Net Assignment* (`soft net` command).

The master designer can create partitions by layer to restrict partition designers to specific layers instead of all layers in the Z axis, allowing them to work directly over and under each other. The master designer specifies the start and stop layers of each layer-based partition.

During design partitioning, a master designer attaches the `LOCKED` property to symbols and modules to prevent partition designers from making modifications; the partition designers cannot remove the `LOCKED` property from objects in exported partitions.

Changes in the partitions can be imported into the master database or refreshed back into the master database or into other partition databases. The master designer controls when updates occur. No automatic updates to the master database occur.

A master designer can create a partition anywhere within the drawing extents of a design as long as it does not overlap other partitions. Overlapping partitions are red and cannot be exported. The master designer must eliminate overlaps prior to exporting partitions.

The tool highlights active partition boundaries. To delete partition boundaries, use the [Workflow Manager](#).

Related Topics

- [Partition Notes Dialog Box](#)
- [Creating Partitions](#)
- [Refreshing Partitions](#)
- [Previewing Partitions Prior to Export](#)
- [Appending a Note](#)

Partition Command: Options Panel

Access Using

- *Menu Path: Place – Design Partition – Create Partitions*
- *Menu Path: Place – Design Partition – Partitioning (from within .dpf, .dpm, and .dps files)*

<i>Partition Data</i>	
<i>Name</i>	Defaults to MASTER_DESIGN. On the initial creation of a partition, its name defaults to PARTITION_2, PARTITION_3, and so on. The master is always PARTITION_1. Once the partition exists, but prior to exporting it, you can change its name and the attached text that appears in a default location on the initial creation of a partition.
<i>Location</i>	For the master design, the system automatically generates the path, which defaults to the current working directory. For a partition, the system creates a directory and a partition file beneath it named PARTITION_2, PARTITION_3, and so on. An extension of .dpf, .dpm, or .dps is appended to partition files. This field is read-only until you export partitions. Once exported, change the partition path by left clicking to display a browser from which a master partition may search for an exported partition whose default path may have changed; a partition may search for the master partition or another partition in the design. A red background in this cell indicates the current path is invalid.
<i>Designer</i>	Defaults to the login that the master designer assigns to the partition. The login must be valid to support the e-mail program. Each time you invoke Workflow Manager, the tool partially opens each partition to populate this field.
<i>Status</i>	Specifies the state of the database as <i>Master</i> , <i>Inactive</i> , <i>Active</i> , or <i>Exported</i> . An <i>Inactive</i> status indicates an unexported database, or one exported and then re-imported without modification.
<i>Progress</i>	Specifies the state of a partition as <i>New</i> , <i>In Progress</i> , or <i>Complete</i> . Each time you invoke Workflow Manager, the tool partially opens each partition to populate this field.
<i>Start Layer</i>	For partitioning by layer, specifies the start layer of a horizontal partition (as opposed to all layers in the Z axis). You can assign one layer or multiple consecutive layers. For example, you can specify layer 2 as both the <i>Start</i> layer and <i>Stop</i> layer. Or you can assign multiple consecutive layers and assign layer 2 as the <i>Start</i> layer and layer 4 as the <i>Stop</i> layer. This means that the partition designer is responsible for layers 2, 3, and 4. The partition command is disabled once you export a partition. You can also assign layers using the <code>workflow</code> command.
<i>Stop Layer</i>	For partitioning by layer, specifies the end layer of a horizontal partition (as opposed to all layers in the Z axis). You can assign one layer or multiple consecutive layers. For example, you can specify layer 2 as both the <i>Start</i> layer and <i>Stop</i> layer. Or you can assign multiple consecutive layers and assign layer 2 as the <i>Start</i> layer and layer 4 as the <i>Stop</i> layer. This means that the partition designer is responsible for layers 2, 3, and 4. The partition command is disabled once you export a partition. You can also assign layers using the <code>workflow</code> command.
<i>Overlap</i>	Specifies that partitions are overlapped. If the partitions overlap, the cell turns red and the text reads YES. The master designer must eliminate overlaps prior to exporting partitions.
<i>Notes</i>	Click to display the Partition Notes dialog box and add timestamped guidelines, comments, and design information unique to each partition. Subsequent notes append to previous comments and remain as a permanent record during the life of the partition.
<i>View Next</i>	Click to scroll through each partition. The master partition is always PARTITION_1, with additional partitions numbered sequentially: PARTITION_2, PARTITION_3, and so on.
<i>Apply</i>	Click to create the partitions using the cut lines you defined, or to accept parameter modifications to the <i>Options</i> panel.
<i>Reset</i>	Click to revert <i>Options</i> panel settings that you modified to their original settings.
<i>Partition Commands</i>	
<i>Preview</i>	Click to produce a report for an unexported partition that lists refdes, component, and package information.
<i>Refresh</i>	Updates the work done by the partition designer within an assigned partition. The partition designer can continue working.
<i>Workflow Manager</i>	Click to display the Workflow Manager , which allows the master designer to manage all partitions in a design.
<i>Line Lock</i>	Specifies the type of segments to use and their direction (<i>90 degree</i> , <i>45 degree</i> , or <i>Off</i>) when adding lines or shapes. Arcs are not allowed.

Related Topics

- [Creating Partitions](#)
- [Refreshing Partitions](#)
- [Previewing Partitions Prior to Export](#)
- [Appending a Note](#)

Partition Notes Dialog Box

This dialog box appears when you click *Notes* in the *Options* panel of the Control Panel when running design partition.

<i>Notes</i>	This field is read-only and displays a chronology of instructions, guidelines, comments, and design information. Each time you invoke Workflow Manager, the design tool partially opens each partition to populate this field.
<i>Append Note</i>	Enter additional instructions, guidelines, comments, and design information for one or more unexported partitions. Subsequent notes append to previous comments and remain as a permanent record during a partition's existence. Each time you invoke Workflow Manager, the design tool partially opens each partition to populate the <i>Notes</i> field.
<i>Save</i>	Click to accept the information you entered and exit the dialog box.
<i>Cancel</i>	Click to discard the modified information you entered and exit the dialog box.

Related Topics

- [partition](#)
- [Refreshing Partitions](#)
- [Previewing Partitions Prior to Export](#)
- [Appending a Note](#)

Creating Partitions

All partitions are restricted by a boundary. When you create a partition, it defaults to include all layers within the boundary or you can assign one or multiple consecutive layers to the partition. To create partitions in your design, follow these steps:

1. Choose *Place – Design Partition – Create Partitions* ([partition](#) command). The *Options* panel changes to reflect partition-related fields. The command window prompt displays the following message:

Draw partition cut lines or use right mouse button options

2. Create partitions using one of the following methods:

- a. Use the mouse to draw several cut lines that divide the board into as many partition sections as required. The lines appear in the color specified for the *Through All Boundary* category in the Color dialog box. The command window displays the following message:

Enter partition cut line(s). Select 'Apply' when Complete

Right-click and choose *Apply* from the pop-up menu that appears or on the *Options* panel. The partition name appears within the partition sections.

- b. Right-click and choose *Add Shape* from the pop-up menu that appears. The command window prompt displays the following message:

Enter shape outline

Use the cursor to position the shape of the required partition.

Right-click and choose *Close Shape* from the pop-up menu that appears. The partition name appears within the shape extents.

- c. Right-click and choose *Add Rectangle* from the pop-up menu that appears. The command window prompt displays the following message:

Enter shape outline

Drag the cursor to position a rectangle the size of the required partition, and left click to instantiate it. The partition name appears within the rectangle extents. You can draw a rectangle that extends beyond the board extents, to accommodate placement.

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

- d. Copy and create from existing shapes such as constrained areas, rooms, or board outlines.

3. If you are creating partitions by layer:

- a. In the *Options* panel, click on the drop-down list in the *Start Layer* field and choose the first layer of your horizontal partition.

- b. In the *Stop Layer* field, click on the drop-down list and choose the ending layer of your horizontal plane.

The layers must be consecutive. For example, if you specify layer 2 as the *Start* layer and specify layer 4 as the *Stop* layer, this means that the partition designer is responsible for layers 2, 3, and 4. This command is disabled once you export a partition.

4. Add more partitions as required.

5. Right-click and choose *Done* from the pop-up menu that appears.

6. Review the *Options* panel fields now populated with partition data. The *Name* field defaults to **MASTER_DESIGN** with a *Status* of **Master** and *Progress* of **New**.

7. Assign each partition to a designer, and change the default names of partitions and locations as required; then click *Apply*.

 The *Name* and *Designer* fields cannot be changed for exported partitions.

8. Click *Notes* and enter any information as required in the Partition Notes dialog box.

9. Click *View Next* to display each partition you created and assign partitions to specific designers, change the default names of partitions and locations as required.

10. Run [guideport](#) (optional) to create visual checkpoints that suggest where potential connections can occur for unrouted nets that cross partition boundaries.

11. Choose *Place – Design Partition – Workflow Manager* ([workflow](#) command) to display the Workflow Manager, from which you can manage all partitions in a design, including importing, exporting, refreshing, deleting, retracting, or previewing. You can also generate a partition report or append notes as necessary.

Related Topics

- [partition](#)
- [Partition Command: Options Panel](#)
- [Previewing Partitions Prior to Export](#)
- [Appending a Note](#)

Refreshing Partitions

To refresh partitions in your design, follow these steps:

1. Choose *Place – Design Partition – Create Partitions* ([partition](#) command).
2. Click *View Next* to display the partition to refresh.
3. Ensure the partition to be refreshed has a *Status* field of *Exported*. Use *Place – Design Partition – Workflow Manager* ([workflow](#) command) to display the Workflow Manager and export partitions.
4. Click *Refresh* to update the work done by the partition designer within an assigned partition.

Related Topics

- [partition](#)
- [Partition Command: Options Panel](#)
- [Partition Notes Dialog Box](#)
- [Appending a Note](#)

Previewing Partitions Prior to Export

Use the Partition Preview Report to ensure the partition contains the correct contents before the master designer exports it, by following these steps:

1. Choose *Place – Design Partition – Create Partitions* ([partition](#) command).
2. Click *View Next* to display the partition to preview.
3. Verify the partition's *Status* field is not *Exported*.
4. Click *Preview*. The Partition Preview Report appears.

Related Topics

- [partition](#)
- [Partition Command: Options Panel](#)
- [Partition Notes Dialog Box](#)
- [Creating Partitions](#)

Appending a Note

Follow these steps to append a note in the Partition Notes dialog box:

1. Choose *Place – Design Partition – Create Partitions* ([partition](#) command).
2. Click *View Next* to display an unexported partition to which to add a note. You cannot add notes to an exported partition, although you can view all notes previously added.
3. Click *Notes*. The Partition Notes dialog box appears.
4. Enter the information in the *Append Notes* section. The Notes section displays previously entered information.
5. Click *Save*. The *Notes* section includes the latest note you entered the next time you open the Partition Notes dialog box.

Related Topics

- [partition](#)
- [Partition Command: Options Panel](#)
- [Partition Notes Dialog Box](#)
- [Creating Partitions](#)
- [Refreshing Partitions](#)

partlogic

The `partlogic` command allows you to view and edit the parts list.

Related Topics

- [Component Browser](#)
- [Adding Parts to the Parts List](#)
- [Adding Models from SI Libraries](#)
- [Adding New Instances of Existing Parts](#)
- [Adding Packages from Package Libraries](#)
- [Adding Parts from Component Libraries](#)
- [Creating Temporary Devices](#)
- [Deleting Parts from the Parts List](#)
- [Modifying Parts](#)

Parts List Dialog Box

Use this dialog box to view and edit the parts list.

Access Using

Menu Path:

- From Allegro X PCB Editor: *Logic – Part Logic*
- From APD: *Logic – Edit Parts List*

Part Selection Area	
<i>RefDes Filter</i>	Searches on part data by reference designator.
<i>Device Filter</i>	Searches on part data by device name.
<i>Sort By</i>	Sorts on part data by <i>Refdes</i> or <i>Device</i> and displays the corresponding information in the Part Selection list box.
<i>Part Selection list box</i>	Displays part data sorted by <i>Device</i> or by <i>Refdes</i> . Data in this area can be chosen for changes in the <i>Part Modification Area</i> .
<i>Browsers</i>	Four browsers are available to let you search libraries for component or package data. Library data that is chosen appears in the <i>Part Modification Area</i> . <i>Schematic Components</i> : Choose to display the Component Browser. <i>Physical Devices</i> : Choose to display the Library Browser. <i>Physical Packages</i> : Choose to display the Package Symbol Browser. <i>SI Components</i> : Choose to display the Model Browser.
<i>Part Modification Area</i>	Specifies a workspace that lets you add, modify or delete parts. The number of data type-in fields depends on the type of component or package chosen.

Related Topics

- [Adding Parts to the Parts List](#)
- [Adding Models from SI Libraries](#)
- [Adding New Instances of Existing Parts](#)
- [Adding Packages from Package Libraries](#)
- [Adding Parts from Component Libraries](#)
- [Creating Temporary Devices](#)
- [Deleting Parts from the Parts List](#)
- [Modifying Parts](#)

Component Browser

The Universal Component Browser is used with commands in Allegro X PCB Editor, PCB SI, and PCB PI option: `edit parts`, `partlogic`, and `power integrity`. Use this window to search for components of your libraries in the `<projectname>.cpm` file of your design project.

(i) The `.cpm` file must exist in the appropriate directory before you invoke the Component Browser.

You can also add new components and replace existing components in your schematic. The Component Browser window has a standard tab, Part, and can have multiple tabs specific to a part. A part-specific tab appears when you view part details.

For more information on the Component Browser, see the Component Browser Interface in the *Allegro Design Entry HDL User Guide*.

Left Pane (Tree View)	By default (Part tab selected), this pane contains three standard nodes, <i>Browse Libraries</i> , <i>Classifications</i> , and <i>Libraries</i> in a tree-like hierarchy. This hierarchy signifies the categorization of cells for performing a part search. The standard nodes contain sub-nodes that can either be a library name or a classification type. The last node in this hierarchy is always a cell. The nodes and sub-nodes of the tree are expandable and collapsible. You can select either a node, multiple sub-nodes, or multiple cells of the same or different sub-nodes for a part search. However, all sub-nodes and the cells you select must be either of library or classification. With a part table row selected, this pane displays the information available for it. This information appears in the form of nodes such as Classification.
Browse Libraries	This node displays the design libraries and the corresponding cells of your design project in the right pane.
Classifications	This node categorizes components into categories, and are listed hierarchically. For example, a category, VCC, may contain all the power part such as VCC_ARROW and PVCC. <p>⚠ To view the categories of the components correctly, make sure you have the <code>.cpm</code> file.</p>
Libraries:	This node lists all the libraries included in the <code>.cpm</code> file as sub-nodes. The cells contained in a library (sub-node) appear as leaf nodes.
Right pane (Search/Details pane)	With the Part tab selected, this pane contains the fields that help you define search parameters for the hierarchy level selected in the Tree View pane. With a part table row selected, this pane contains the detailed information such as attribute, classification, features, symbol and footprint models, and manufacturer for the part. For example, if you click the Part tab, then various search parameters appear in the Search/Details pane. On selecting a part table row, the Search/Details pane will contain corresponding attributes and symbol details for the cell. If you chose <i>Classification</i> or <i>Libraries</i> , the following appear: <i>First List box</i> : Lets you enter or choose an attribute for search. <i>Second List box</i> : Lets you enter or choose a logical operator for the attribute <i>Third List box</i> : Lets you enter or choose a value for attribute. <i>More</i> : Click to add a new attribute for a multi-attribute search criteria. You can add a maximum of six attributes. <i>Fewer</i> : Click to delete the last attribute-search criterion. At least one attribute row always remains; you cannot delete it. <i>Match Any</i> : Click to specify an OR condition between multiple attributes for search. <i>Match All</i> : Click to specify an AND condition between multiple attributes for search. <i>Search</i> : Click to run a search.
Bottom Pane	This pane displays the part table rows that meet the search criteria pane.
Search Path	Appears at the top of the Search/Details pane that signifies the location, where the search will be performed.

Physical Devices

This button invokes the Libraries Browser dialog box that can be used to load a device or package file from the components library when modifying a part in Allegro SI.

File Filter	Uses wildcard to limit the library search
List Box	Displays all parts (<code>.txt</code>) or packages (<code>.psm</code>) allowed by the filter

Physical Packages

This button invokes the Package Symbol Browser dialog box to load a device or package file from the components library when modifying a part in Allegro SI.

File Filter	Uses wildcard to limit the library search
List Box	Displays all parts (<code>.txt</code>) or packages (<code>.psm</code>) allowed by the filter

SI Components

This button invokes the SI Model Browser dialog box that can be used to create, add, delete, or edit models in your designated working device or interconnect library. You establish a working device model library and a working interconnect model library from the Library Browser. You can leave the Signal Analysis Library Browser open at the same time as the Model Browser.

You can also use SigXsect to examine the electrical fields surrounding a chosen interconnect model in cross-section.

The Model Filters area displays drop-down menus and text fields you can use to specify

- Which library to choose models from
- Which device model type to display
- Which model name pattern you want the search to match.

You can select libraries in the Signal Analysis Library Browser, which can be open at the same time as the Model Browser.

Library Filter	Displays a drop-down menu you can use to choose the library to show models from. <i>Selected Device Library</i> : Displays models from the device library chosen from the device library search list in the Signal Analysis Library Browser. <i>Working Device Library</i> : Displays models from the working device model library as designated in the Signal Analysis Library Browser. <i>Selected Interconnect Library</i> : Displays models from the interconnect library chosen from the interconnect library search list in the Signal Analysis Library Browser. <i>Working Interconnect Library</i> : Displays models from the working interconnect model library as designated in the Signal Analysis Library Browser. <i>All Libraries</i> : Displays models from all libraries listed in both search lists in the Signal Analysis Library Browser.
Model Type Filter	Displays a drop-down menu you can use to control the display of device and interconnect models. Device and interconnect model types are listed individually, along with the following: <i>Any</i> : Displays all models. <i>AnyDevModel</i> : Displays all models that are not interconnect models. <i>AnylcnModel</i> : Displays all interconnect models. <i>AnyDevice</i> : Displays all models for devices <i>AnylOCell</i> : Displays IBIS IOCell models.
Model Name Pattern	Lists models whose names match a designated character pattern. To filter by model name pattern, enter a wildcard (*) with part of the model name character string. An asterisk (*) alone shows all models matching the specified library and model type filters.
Library Buttons	Use these buttons to add, modify or delete models from a chosen library.
Add Model	Displays the Add Model pop-up menu of the types of device and interconnect models you can add to the working device or interconnect library. Options include: <i>CloneSelection</i> -- Copies or clones the model that you choose in the Model Browser list box, prompts you to name the copy, and adds the renamed copy to the working library. <i>EspiceDevice</i> – Displays the Create ESpice Device Model dialog box. <i>IBISDevice</i> – Displays the Create IBIS Device Model dialog box. <i>PackageModel through Via</i> – Creates an empty text file in the working library that you must edit to create the model. The simulator prompts you to name the model. Depending on your selection, the Create Model dialog box for the chosen model appears, or a dialog box for specifying a new model name appears.
Delete	Deletes the chosen model.
Edit	Displays a model editor or a text editor, depending on the type of model you choose in the Model Browser search list.
TextEdit	Displays a text editor for the chosen model.
View	Displays the SigXsect window for a geometric cross section of the interconnect model you chosen in the Model Browser search list.
Solve	Runs the field solver to regenerate an interconnect model.

Related Topics

- [partlogic](#)
- [Adding Models from SI Libraries](#)
- [Adding New Instances of Existing Parts](#)
- [Adding Packages from Package Libraries](#)
- [Adding Parts from Component Libraries](#)
- [Creating Temporary Devices](#)
- [Deleting Parts from the Parts List](#)
- [Modifying Parts](#)
- [edit parts](#)
- [partlogic](#)
- [power integrity](#)

Adding Parts to the Parts List

You can add from Allegro Design Entry HDL L or Allegro System Architect GXL Component Libraries to the parts list. To add parts, follow these steps:

1. Choose *Logic – Part Logic*.
Alternatively, you can also type `part logic` in the Command window. The Edit Parts List dialog box appears.
2. In the *Browsers* area, click *Schematic Components*.
The Component Browser appears.
3. To update the data in the *Part Modification Area* of the Edit Parts List dialog box, search for a part in the Component Browser.
4. Click a part to choose in the Search Results pane.
The *<Part Name>* tab appears with the part information.
5. Add a unique reference designator for each new instance to be created in the *Refdes* field.
6. Click *Add* in the Search/Details pane. Alternatively, right-click the part table row for the part, and choose *Add to Design* from the pop-up menu. You can also double-click the part table row to add the selected part to the design.

Related Topics

- [partlogic](#)
- [Parts List Dialog Box](#)
- [Adding New Instances of Existing Parts](#)
- [Adding Packages from Package Libraries](#)
- [Adding Parts from Component Libraries](#)
- [Creating Temporary Devices](#)
- [Deleting Parts from the Parts List](#)
- [Modifying Parts](#)
- [edit parts](#)
- [power integrity](#)

Adding Models from SI Libraries

To add models from the SI libraries to the parts list, follow these steps:

1. Choose *Logic – Part Logic*.

Alternatively, you can also type `part logic` in the Command window. The Edit Parts List dialog box appears.

2. In the *Browsers* area, click *SI Components*.

3. Choose a device in the Model Browser.

Data in the *Part Modification Area (Device field)* of the Edit Parts List dialog box is updated accordingly.

 You must add the package information.

4. Add a unique reference designator for each new instance to be created.

5. Click *Add*.

The new item is added and highlighted in the parts list.

Related Topics

- [partlogic](#)
- [Parts List Dialog Box](#)
- [Component Browser](#)
- [Adding Packages from Package Libraries](#)
- [Adding Parts from Component Libraries](#)
- [Creating Temporary Devices](#)
- [Deleting Parts from the Parts List](#)
- [Modifying Parts](#)
- [edit parts](#)
- [power integrity](#)

Adding New Instances of Existing Parts

To add a new instance of an existing part to the parts list, follow these steps:

1. Choose *Logic – Part Logic*.
Alternatively, you can also type `part logic` in the Command window. The Edit Parts List dialog box appears.
2. Choose an instance of the part in the parts list.
The data components of the chosen part are loaded into the *Part Modification Area*.
3. In the *Refdes* field, add one or more unique reference designators for the new instance or instances to be created.
4. Click *Add*.
The new items are added and highlighted in the parts list.

Related Topics

- [partlogic](#)
- [Parts List Dialog Box](#)
- [Component Browser](#)
- [Adding Parts to the Parts List](#)
- [Adding Parts from Component Libraries](#)
- [Creating Temporary Devices](#)
- [Deleting Parts from the Parts List](#)
- [Modifying Parts](#)
- [edit parts](#)
- [power integrity](#)

Adding Packages from Package Libraries

To add packages from package libraries, follow these steps:

1. Choose *Logic – Part Logic*.
Alternatively, you can also type `part logic` in the Command window. The Edit Parts List dialog box appears.
2. In the *Browsers* area, click *Physical Packages*.
3. Choose a package name in the Package Library browser to update the data in the *Package* field of the Edit Parts List dialog box.
4. Add a unique reference designator and device name for each new instance to be created.
5. Click *Add*.

The new item is added and highlighted in the parts list.

Related Topics

- [partlogic](#)
- [Parts List Dialog Box](#)
- [Component Browser](#)
- [Adding Parts to the Parts List](#)
- [Adding Models from SI Libraries](#)
- [Creating Temporary Devices](#)
- [Deleting Parts from the Parts List](#)
- [Modifying Parts](#)
- [edit parts](#)
- [power integrity](#)

Adding Parts from Component Libraries

To add parts from component libraries to the parts list, follow these steps:

1. Choose *Logic – Part Logic*.
Alternatively, you can also type `part logic` in the Command window. The Edit Parts List dialog box appears.
2. In the *Browsers* area, click *Physical Devices*.
3. Choose a device in the Component Library Browser to update the data in the *Part Modification Area* of the Edit Parts List dialog box.
4. Add a unique reference designator for each new instance to be created.
5. Click *Add*.

The new item is added and highlighted in the parts list.

Related Topics

- [partlogic](#)
- [Parts List Dialog Box](#)
- [Component Browser](#)
- [Adding Parts to the Parts List](#)
- [Adding Models from SI Libraries](#)
- [Adding New Instances of Existing Parts](#)
- [Deleting Parts from the Parts List](#)
- [Modifying Parts](#)
- [edit parts](#)
- [power integrity](#)

Creating Temporary Devices

Follow these steps to create temporary devices:

1. Choose *Logic – Part Logic*.
Alternatively, you can also type `part logic` in the Command window. The Edit Parts List dialog box appears.
2. Type the name in the *Device* field in the *Part Modification Area* section.
3. Click *OK*.
4. Type a package name for the temporary component in the *Package* field or choose an existing package from the Package Library Browser.
5. Type a unique reference designator for the temporary component in the *Refdes* field.
6. Click *Add*.
If the package name that you entered does not exist, a pop-up box appears and prompts you for a pin count for the temporary device.
7. Enter the number of pins you want on the temporary device, then choose *Done* in the pop-up menu.
The new item is added and highlighted in the parts list.

Related Topics

- [partlogic](#)
- [Parts List Dialog Box](#)
- [Component Browser](#)
- [Adding Parts to the Parts List](#)
- [Adding Models from SI Libraries](#)
- [Adding New Instances of Existing Parts](#)
- [Adding Packages from Package Libraries](#)
- [Modifying Parts](#)
- [edit parts](#)
- [power integrity](#)

Deleting Parts from the Parts List

To delete parts from the Edit Parts List dialog box, follow these steps:

1. Choose *Logic – Part Logic*.
Alternatively, you can also type `part logic` in the Command window. The Edit Parts List dialog box appears.
2. In the parts list, choose the part to be deleted.
The chosen instances are highlighted in the parts list and the component data is displayed in the *Part Modification Area*.
3. Click *Delete*.
4. Click *Apply* or *OK*.

 If you delete instances by mistake, click *Add* to add the instances back into your design.

Related Topics

- [partlogic](#)
- [Parts List Dialog Box](#)
- [Component Browser](#)
- [Adding Parts to the Parts List](#)
- [Adding Models from SI Libraries](#)
- [Adding New Instances of Existing Parts](#)
- [Adding Packages from Package Libraries](#)
- [Adding Parts from Component Libraries](#)
- [edit parts](#)
- [power integrity](#)

Modifying Parts

To modify a part in the Edit Parts list dialog box, follow these steps:

1. Choose *Logic – Part Logic*.

Alternatively, you can also type `part logic` in the Command window. The Edit Parts List dialog box appears.

2. Click the specified part from the parts list located in the *Part Selection Area* list box.

All currently available part parameters (*Reference Designator, Device, Value, Tolerance, and Package*) appears in the *Part Modification Area* fields in the bottom right portion of the dialog box.

3. Edit the specified parameters.

4. Click *Modify*.

5. Click *Apply* or *OK*.

Related Topics

- [partlogic](#)
- [Parts List Dialog Box](#)
- [Component Browser](#)
- [Adding Parts to the Parts List](#)
- [Adding Models from SI Libraries](#)
- [Adding New Instances of Existing Parts](#)
- [Adding Packages from Package Libraries](#)
- [Adding Parts from Component Libraries](#)
- [Creating Temporary Devices](#)
- [edit parts](#)
- [power integrity](#)

paste

The `paste` command lets you repeatedly paste the same objects to different destinations without re-selecting the copied objects. This command works in conjunction with the `copy` command to paste objects stored in the paste buffer to multiple destinations at any time in the design.

You can paste multiple copies of elements by pasting them in horizontal/vertical (rectangular) patterns or in radial (polar) patterns around a user-defined point. In the *Options* panel, set the quantity or number of copies to paste more than one time.

When invoked in the pre-selection use model, the command is restricted to following object types:

- Vias
- Pins
- Fingers
- Dangling Cline ends

 In pre-select mode, the `paste` command is only available under General Edit and Etch Edit application modes.

When pasting objects the command automatically snaps to the center of the destination objects. Destination objects can be in a different layer than the copied object.

For selecting destinations you can use any of the following methods:

- Find filter
- Window Selection
- Select by Polygon
- Select by Lasso
- Select on Path
- Temp Group
- Find by Query

Syntax

`paste`

The `paste` command is also available in the pop-up if the selected objects are pins, vias, fingers, or clines and the paste buffer has content.

Related Topics

- [Pasting Elements to Multiple Destinations in Rectangular Patterns using Post-select Mode](#)
- [Pasting Elements to Multiple Destinations in Polar Patterns using Post-select Mode](#)
- [Pasting Elements using Pre-select Mode](#)

Paste Command: Options Panel

Access Using

- Menu Path: Edit – Paste

Use the fields on the *Options* panel to control how elements are pasted. The choices vary depending on whether you set *Paste mode* to *Rectangular* or *Polar*.

<i>Paste mode</i>	Specifies the type of pattern you are pasting. These are the choices:	
	<i>Rectangular</i>	Pastes copies in a rectangular grid array.
	<i>Polar</i>	Pastes copies around a user-defined point in angular increments.
<i>Rectangular Options</i>		
<i>Qty X</i>	Defines the number of columns to be created.	
<i>Qty Y</i>	Defines the number of rows to be created.	
<i>Spacing</i>	Indicates row and column (X and Y) spacing. <i>Spacing X</i> indicates the amount of distance in user units between items in a row. <i>Spacing Y</i> indicates the amount of distance in user units between items in a column.	
<i>Order</i>	Indicates the direction in which the tool should paste the copies in each row and column. <i>Order X</i> specifies the direction that rows are pasted. <i>Right</i> is the default. <i>Order Y</i> specifies the direction that columns are pasted. <i>Down</i> is the default.	
<i>Rotation angle</i>	Specifies the angle at which each element is placed if you choose <i>Rotate</i> from the pop-up menu. Enter a number between 0 and 360 or choose a number from the pop-up. The tool allows accuracy for up to three decimal places. ⚠ <i>Rotate</i> works with all elements that you can copy except figures. Figures appear to rotate to any angle, but when you choose a location point, the tool snaps the figure to the nearest 90-degree increment.	
<i>Polar Options</i>		
<i>Direction</i>	Specifies a direction for pasting the elements: <i>Cww</i> (counter-clockwise) or <i>Cw</i> .(clockwise).	
<i>Copies</i>	Specifies how many copies of the element the tool pastes. The default value is 1.	
<i>Rotation angle</i>	Specifies the incremental angle to be used when pasting the elements. Enter a number between 0 and 360 or choose a number from the pop-up. The tool allows accuracy for up to three decimal places.	
<i>Retain net of vias</i>	Allows vias to either retain their source nets or inherit net of the destination object. When disabled, via inherits the net. This option is disabled by default. When via touches a shape, the via inherits the net of the destination shape. If a via touches multiple nets from different shapes, a random net will be assigned to the via.	
<i>Retain net of shapes</i>	Allows shapes to either retain their source nets or inherit net of the destination object. When enabled, shapes retains source nets. This option is enabled by default. When disabled, the shape inherits the net of the destination object based on the objects hierarchy. For a shape to inherit the net of a <ul style="list-style-type: none"> • Pin: The shape must touch center or pin • Via: The shape must touch center of via • Cline: The shape must touch segment vertex • Shape: The shape must touch any part of shape ⚠ If a shape touches multiple nets of the same hierarchical objects, a random net will be assigned to shape. ⚠ If there is no net to assign to a shape, a dummy net will be assigned to the shape.	

Related Topics

- [Pasting Elements to Multiple Destinations in Polar Patterns using Post-select Mode](#)
- [Pasting Elements using Pre-select Mode](#)

Pasting Elements to Multiple Destinations in Rectangular Patterns using Post-select Mode

Follow these steps to paste elements in a rectangular pattern using post-select mode:

1. Choose *Edit – Paste* command.

The copied element attached to the cursor and the following message appears in the command window:

To select destination, LMB click anywhere on canvas or Window/group select enabled Find Filter object(s).

2. Right-click and choose *Options* from the pop-up menu, or in the *Options* panel, choose *Rectangular* and complete the other entries.

3. Select objects in the *Find* filter.

4. If you entered a value in the *Rotation angle* field:

- a. Choose *Rotate* from the pop-up menu.

- b. Rotate the elements to the angle you need and choose to lock them in that position.

5. Select destination objects using standard methods of selection.

The following message appears in the command window.

Pasted to # out of # selected destination objects.

6. Right-click and choose *Done* to exit the command.

Related Topics

- [paste](#)
- [Pasting Elements using Pre-select Mode](#)

Pasting Elements to Multiple Destinations in Polar Patterns using Post-select Mode

Follow these steps to paste elements in a polar pattern using post-select mode:

1. Choose *Edit – Paste* command.

The copied element attached to the cursor and the following message appears in the command window:

To select destination, LMB click anywhere on canvas or Window/group select enabled Find Filter object(s).

2. Right-choose and choose *Options* or in the *Options* panel, choose *Polar* and complete the other entries.

3. Select objects in the *Find* filter.

4. Select destination objects using standard methods of selection.

The tool pastes a pattern of copies around the point of selection and displays following message in the command window.

Pasted to # out of # selected destination objects.

5. Right-click and choose *Done* to exit the command.

Related Topics

- [paste](#)
- [Paste Command: Options Panel](#)

Pasting Elements using Pre-select Mode

Follow these steps to paste elements using pre-select mode:

1. Select valid destination objects using standard methods of selection.

2. Hover your cursor over one of the selected elements.

The tool highlights the element.

3. Right-click and choose *Paste* from the pop-up menu or type `paste` in the command window.

The copied object is pasted at the selected location and the following message appears in the command window:

Pasted to # out of # selected destination objects.

4. Repeat the steps from 1 to 3 to paste copied object to other destinations.

Related Topics

- [paste](#)
- [Paste Command: Options Panel](#)
- [Pasting Elements to Multiple Destinations in Rectangular Patterns using Post-select Mode](#)

pause

The `pause` command suspends processing when playing a script. When the pause command is accompanied by an integer, the script suspends processing for the specified number of seconds before processing is continued. A pause box appears in the display when a script encounters a pause. (See also [stop](#), [stopwatch](#))

Syntax

```
pause [<time in seconds>]
```

Suspending Processing During Script Playing

Perform this step to suspend processing when scripting is playing:

1. Type pause and a value representing the number of seconds to suspend scripting.

To restart scripting at any time while pause is running, press any key or click the mouse.

Example

```
pause
```

```
pause 30
```

pbar check

The `pbar check` command lets you verify and report plating bar connectivity errors and plating trace spacing violations, and delete DRC errors generated in previous runs of the command.

The tool checks the following objects for plating connections:

- BGA balls
- Bond fingers
- Flip-chip pads
- Discrete components pads

For additional information, see *Working with a Plating Bar* in the *Routing User Guide*.

Related Topics

- [Plating Bar Check Dialog Box](#)
- [Reporting Plating Bar Connectivity Errors](#)

Plating-Bar Selection Dialog Box

Because your design should not contain more than one plating bar, the Plating Bar Selection dialog box appears if the command recognizes another component instance that it cannot distinguish from a plating bar symbol. This occurs when there is more than one component in the design that meets the criteria the command uses to recognize the plating bar; for instance, if components other than the plating bar have the PLATING_BAR property or if another component of the class DISCRETE is located outside the package outline.

If this dialog box appears while you are creating a plating bar, see [pbar create](#) for further instructions; if you are deleting a plating bar, go to [pbar delete](#).

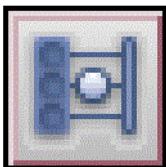
Related Topics

- [Reporting Plating Bar Connectivity Errors](#)

Plating Bar Check Dialog Box

Access Using

- Menu Path: Manufacture – Plating Bar Check
- Toolbar Icons:



<i>Missing Connections</i>		
<i>Check for unplated nets</i>		Controls the fields below, and governs the plating checks relating to actual (logical or physical) connections to the plating bar. The default setting is <i>On</i> .
<i>For multi-pin nets, check all:</i>		The fields below apply to nets that have more than one item to be plated, for example, a net from a wirebond die to a bond finger and out to a BGA ball, has two items to be plated (the ball and finger). If you disable a specific item type from this group, as soon as any item (of any type) from the net is plated, all objects of that type are plated. To continue the example, if you turn off BGA balls as an option, and the bond finger is properly plated, then the BGA ball, if unplated, will not report an error.
	<i>BGA balls</i>	<p>If you check this box, the tool checks BGA balls (IO class components) in the design for plating connections. The default setting is <i>On</i>. The design tool generates:</p> <ul style="list-style-type: none"> ◦ Errors for balls on real nets that are not plated. ◦ Warnings for balls on dummy nets that are not plated. <p style="margin-left: 40px;">These are errors if you check <i>Report dummy net violations as errors</i>.</p> <ul style="list-style-type: none"> ◦ Warnings for balls that have multiple plating bar connections. <p>If you do not check this box, the tool considers the entire net plated as long as any item on the net is plated.</p>
	<i>Bond fingers</i>	<p>When you check this box, the tool checks bond fingers in the design for plating bar connections. Wire bond die pads are not checked directly. Only bond fingers are checked because there is no way to connect a wire bond die pad down to the substrate surface other than by using a bond wire. The default setting is <i>On</i>. The design tool checks:</p> <ul style="list-style-type: none"> ◦ Errors for bond fingers on real nets that are not plated. ◦ Warnings for bond fingers on dummy nets that are not plated. <p style="margin-left: 40px;">These are errors if you check <i>Report dummy net violations as errors</i>.</p> <ul style="list-style-type: none"> ◦ Warnings for bond fingers that have multiple plating bar connections. <p>If you do not check this box, the tool considers the entire net plated as long as any item on the net is plated.</p>
	<i>Flip-Chip pads</i>	When you check this box, the tool checks all the pins of flip-chip die components for plating connections. The default setting is <i>On</i> . If you do not check this box, the tool considers the entire net plated as long as any item on the net is plated.
	<i>Discrete Component pins</i>	When you check this box, the tool checks the discrete component pads for plating connections. The default setting is <i>On</i> . If you do not check this box, the tool considers the entire net plated as long as any item on the net is plated.

P Commands
P Commands--pbar check

<i>Allow indirect connection through other components</i>	Indirect connections are connections to the plating bar that go through pins or other components than the BGA to get to the plating bar, or for the plating bar, BGA ball connections that first go through other BGA balls. These are examples of some types of connections that the tool considers as indirect: <ul style="list-style-type: none"> ◦ The connection goes through another IC class component pin (die-to-die to plating bar). ◦ The connection goes through another pin of the same component (bga-to-bga to plating bar). ◦ Bond finger connections that go through another bond finger (bond finger-to bond finger to plating bar) If you are unsure about the results reported when the <i>Allow indirect connection through other components</i> box is turned off and believe that some of these pins are plated, then turn this option on and re-run the report. The default setting is <i>Off</i> .
<i>Perform etch-back plating checks</i>	When you check this box, the tool checks any nets not directly connected to the plating bar for connections through an etch-back trace. If there is no etch-back in your design, leave this option unchecked as it can affect performance. The default setting is <i>Off</i> .
<i>Report dummy net violations as errors</i>	By default, the plating bar check reports dummy net pins only as warnings. Since these pins are not currently in use, their plating is not required. Some design flows require that the tool plate dummy nets (for example, in the case where the pin is used in a later iteration of the substrate). In these situations, enable this option. The default setting is <i>Off</i> . When checked, the tool displays unplated dummy nets as DRC errors.
<i>Highlight unplated nets (dehighlights plated nets)</i>	When checked, the tool highlights only the unplated nets in the color of your choice, and dehighlights the plated nets in the design. This makes it easier to identify the unplated nets when making changes to the design. The default setting is <i>Off</i> , to preserve the design coloring you specified.
<i>Highlight on any required connection(s) missing</i>	When checked, highlights a net if the net has ANY missing connections (even just one pin/finger not plated).
<i>Color</i>	When you Highlight unplated nets, you can choose a color from the pull-down menu. The default setting is the first color in the drop-down list.
Spacing Violations	
<i>Check for plating trace spacing violations</i>	When you check this box, the fields are enabled. The tool searches your design for spacing violations by measuring the clearances between neighboring plating bar traces to see if they are likely to cause manufacturing problems when the plating bar is removed during the late stages of manufacturing. The default setting is <i>On</i> .
	<i>Min separation</i> This field is active when you <i>Check for plating trace spacing violations</i> . Specify this value as the minimum clearance (as measured between the clines where they meet the plating bar itself) between two plating traces on the same routing layer. All clines connected directly to plating bar pins are checked. This value also specifies the distance from the edge of the plating bar inward (toward the BGA). Minimum separation ensures that when the plating bar is removed, extra metal from one plating trace does not bend down and short the other. The default value is the current line-to-line minimum spacing on the bottom conductor layer.
	<i>Edge distance</i> This field is active when you <i>Check for plating trace spacing violations</i> . You can modify the <i>Min separation</i> parameter. If you enter a non-zero value, the tool uses the value to check the minimum separation from the plating bar boundary. The default setting is 0, which implies that separation is measured at the plating bar edge only.
	<i>Min offset</i> This field is active when you <i>Check for plating trace spacing violations</i> . Specify this value as the minimum separation distance (as measured between the clines where they meet the plating bar itself) between two plating traces on adjacent layers. Setting this value ensures that when the plating bar is removed, extra metal from one plating trace does not bend down and short the other. The default value is one-half the line-line spacing constraint value on the bottom conductor layer.
	<i>Min length</i> Specifies the minimum length of the final, straight-line segment going to the plating bar pin. This ensures that the cline does not get sheared off at an angle when the plating bar is removed. This field defaults to 0.00, which means that any angle clines may be routed directly to the plating bar pin. Increase the distance to increase the length requirement for orthogonal entry to the plating bar pad.
<i>Clear DRCs</i>	Clears all plating bar-related DRCs from the design.
<i>OK</i>	Exits from the command, after running a plating bar check on the design based on the current configuration in the Plating Bar Check dialog box.
<i>Close</i>	Exits from the command without running a plating bar check.
<i>Help</i>	Displays help for this command.

 Plating Bar Check settings are saved with the design and will be used any time this check is run in the future.

Related Topics

- [pbar check](#)

Reporting Plating Bar Connectivity Errors

Before performing this task, be sure that you route the design (at least partially) and create a plating bar (`pbar create` command). Then you can run the plating bar check at any time to obtain a report on nets that are not currently plated in the design.

To report a plating bar connectivity error, perform these steps:

1. Run the `pbar check` command.
The Plating Bar Check dialog box appears.
2. **To generate and display a plating bar check report, *Check for unplated nets*.**
You do not need to check the other *Check...* control boxes to generate a report, but only the conditions you activate are reported.
3. To check your design for minimum line-to-line spacing between connections on the same layer (minimum separation) and line-to-line spacing between connections on different layers (minimum offset), check the *Check for plating trace spacing violations* box.
4. Click *OK* to generate and display the plating trace spacing violations report. A report is not generated if the design contains no DRCs.
5. **To clear existing DRCs, click *Clear DRCs*.**

Related Topics

- [pbar check](#)
- [Plating-Bar Selection Dialog Box](#)

pbar create

The `pbar create` command creates a plating bar component with a rectangular symbol that parallels the coordinate axes of your layout design. A wider boundary is added to the plating bar (drawn using the default line width on the Conductor layer). This helps you to see or select the plating bar. Now the tool does not extend plating traces beyond the boundary of the plating bar.

If a plating bar already exists in your design, this command replaces it with a new one. The symbol and component definitions and reference designation of the old plating bar are attached to the replaced symbol. Note the two following conditions when replacing an existing plating bar with a new one:

- DRCs can occur if minimum pin-to-pin spacing exceeds minimum line-to-line spacing of the new plating bar.
- If the existing plating bar has the FIXED property attached, it cannot be replaced. In this case, the following error message is generated in the command window:

The plating bar component has the FIXED property, so it cannot be deleted.

The `pbar create` command treats connect lines (clines) in your design in the following manner:

- Clines at the package boundary are extended orthogonally to the plating bar boundary
- Clines that extend past the plating bar symbol are trimmed to the plating bar boundary
- Clines whose endpoints fall between the package boundary and plating bar are extended orthogonally from the endpoints to the plating bar

These behaviors occur without regard for the FIXED or NO_ROUTE properties attached to nets or clines.

Before you can begin the plating bar process, you must

- Determine the number of nets, including no-connects, in the design so that you know how many pins are needed.
If you do not know the exact number of pins, you should overestimate. Multiple connections on the same net may be made to the plating bar (for example, as with power and ground connections).
- Attach the DISCRETE device type property to the plating bar symbol (for auto-net assignment).

 The `pbar create` command automatically attaches the property PLATING_BAR to the component instance.

- Include the layer from which package pins are routed and the layer on which the plating bar resides (the copper shape that defines the plating bar) in the plating bar padstack definitions.
- Attach the PLATING property to the plating bar's copper shape.

 The layout tools support the generation of a plating bar in designs that have more than one BGA. The plating bar will be based on the combined extents of all the BGAs in the design.

For additional information on plating bars, see the *Placing the Elements* user guide in your documentation set.

Related Topics

- [Create Plating Bar Dialog Box](#)
- [Creating a Plating Bar Component](#)

Create Plating Bar Dialog Box

Access Using

- *Menu Path: Manufacture – Create Plating Bar*

The Create Plating Bar dialog box lets you create a plating bar using the editor's plating bar generator. You have the option of creating a plating bar connection to unused package pins in your design. This option assigns a net name to each unused package pin and routes it to the plating bar.

Related Topics

- [pbar create](#)

Creating a Plating Bar Component

To create a plating bar component:

1. Run `pbar create`.
One of two dialog boxes display. If there is more than one component in the design that meets the criteria the command uses to recognize the plating bar, the Plating-Bar Selection dialog box appears. If so:
 2. Choose the component in the list that you want to replace (normally the plating bar), and click *OK*.
If the command recognizes only one component that meets its criteria, or if there is no plating bar in the design, the Create Plating Bar dialog box appears.
 3. If you want to assign a net name to each unused package pin and route it to the plating bar, click the check box.
 4. Enter the distance you want the plating bar to lie outside the package outline. The dialog box displays a default distance that is twice the pin pitch.
 5. Choose from the drop-down lists to assign a class and subclass where the tool draws the plating bar outline. The default layer is ASSEMBLY_TOP. This affects the thick line only; the assembly rectangle still goes on the ASSEMBLY_TOP layer.
 6. Click *Create*.
The new plating bar is created at the specified distance, with all clines that were previously routed to the package outline connected. The new plating bar replaces an existing one.

Related Topics

- [pbar create](#)
- [Plating-Bar Selection Dialog Box](#)

pbar delete

The `pbar delete` command deletes the plating bar in your design. This action removes the plating bar's component and symbol instance and definition, unless the plating bar has the **FIXED** property attached. Error messages are generated if the command does not detect the presence of a plating bar, or if the plating bar component cannot be distinguished from other components of the class **DISCRETE**. In circumstances where the command detects ambiguous components, a list of such elements displays from which you can choose the plating bar component.

Related Topics

- [Delete Plating Bar Dialog Box](#)
- [Deleting the Plating Bar in Your Design](#)

Delete Plating Bar Dialog Box

Access Using

- *Menu Path: Manufacture – Delete Plating Bar*

This dialog box prompts you to delete the highlighted component and gives you the options to:

- Trim connect lines back to the package outline
- Trim connect lines back to the package pins
- Retain existing connect lines without change

Related Topics

- [pbar delete](#)

Deleting the Plating Bar in Your Design

To delete a plating bar from your design:

1. Enter the command `pbar delete` at the command prompt.
The Delete Plating Bar dialog box appears.
2. Click the appropriate check box governing the configuration of the connect lines.
3. Click *OK*.

If the command detects an existing plating bar, it is deleted. Warning or error messages may be displayed if `pbar delete` cannot be performed.

Related Topics

- [pbar delete](#)
- [Plating-Bar Selection Dialog Box](#)

pcad in

The `pcad in` command imports information from PCAD, PDIF, and PCB database files into board/substrate databases. It is assumed that the Altium PCAD databases being translated are completed (placed and routed). The command can be run in batch mode or as an interactive command within the user interface.

Syntax

The PCAD translator can run in batch mode by specifying all required information on the command line. This removes the need for intervention with the user interface and allows for batch translation of several databases using DOS batch files.

1. Export a PCAD job file to a PCAD PDIF database that can be read by the translator, as explained in [Creating a PCAD PDIF Database File](#).
2. At an operating system prompt, type `pcad_in` (note the underscore) and arguments on a single line.

The command syntax is

```
pcad_in -i <input_filename> -o <output_directoryname> -f <Options_file> -d <drill_filename> -s <aperture_filename>
```

 You can enter the arguments in any order, but you must enter the full path name to each file if they are not in the current working directory.

If all required command line arguments are specified, the translator works in batch mode. Otherwise the *File Names dialog box* appears.

3. Enter all required file names in their full path configuration.

4. Click *OK* to continue.

`pcad_in` reads the input file. If all required program arguments are not specified, the *P-CAD to Allegro Translation Options* dialog box displays.

The *P-CAD To Allegro Layer Mapping* group of controls is used to define the layer mappings. The list box has a list of all PCAD layer names found in the PDIF file and the name of the class and subclass the layers are mapped to.

To change an element mapping

1. Choose the mapping in the list box of the *P-CAD to Allegro Translation Options* dialog box.
The objects target class and subclass mappings are chosen in the Class and Sub Class fields.
2. To change the target class, click the arrow to the right of the Class field and choose one of the allowed class names.
3. To change the target subclass, do one of the following:
 - Click the arrow to the right of the Class field and choose one of the allowed class names

—or—

- Type a new subclass name in the Sub Class field

 You can not define new subclass names if the class is set to PIN or VIA.

4. In the *Location of P-CAD data* area, click the arrow to the right of the *Board/Substrate Outline* field and choose the PCAD layer that contains the board/substrate outline.
—or—
Choose {none} if no board/substrate outline exists.
5. Click the arrow to the right of the *Route Keepin* field to choose the PCAD layer that contains route keepins.
—or—
Choose {none} if no route keepin data exists.
6. Click *OK* to perform the translation or *Cancel* to cancel the translation.

Related Topics

- [Creating a PCAD PDIF Database File](#)
- [Importing Information into Board/substrate Databases](#)
- [Editing the Database](#)

PCAD IN Dialog Box

Access Using

- *Menu Path: File – Import – CAD Translators – PCAD*

<i>PDIF Input File</i>	Specifies the full path and name of the PCAD database file.
<i>Aperture File</i>	Specifies the full path and name of the aperture file. The aperture file contains all aperture information for flashes.
<i>Options File</i>	Specifies the full path and name of the options file. If this does not exist it is created.
<i>[Drill Table]</i>	Specifies the full path and name of the drill table file. This is a PCAD drill table file that contains drill sizes for each pad code
<i>Output Design</i>	Specifies the full path and name of the output directory.
<i>Show options dialog</i>	Shows set options dialog box of translator. If this checkbox is not selected translator takes options from .ini file.

Related Topics

- [Importing Information into Board/substrate Databases](#)
- [Editing the Database](#)

Creating a PCAD PDIF Database File

The PCAD job file must be converted to a PCAD PDIF database that can be read by the translator. This file contains all component, component instances and net information found in the PCB database. A PDIF file is self-contained does not require any information from PCAD library files.

1. Run PCAD PC-CARDS and load the PCB database file that you would like translated.
2. For versions 4 and 5: Run the SCMD/GSSF command to attach padstacks to the database.
For versions 6 and 7: Run the *Environment — Attach Padstacks* command to attach padstacks to the database.
3. For versions 4 and 5: Run the SCMD/VMRG command to merge the free-standing voids with any overlapping polygons.
For version 6 and 7: Run the Environment—Merge Voids command to merge the free-standing voids with any overlapping polygons.
4. Save the database and exit PC-CARDS.
5. Convert the PCB database using the `PDIFOUT.EXE` program using the default configuration options.

See the PCAD documentation for more information on this subject.

Related Topics

- [pcad in](#)
- [Editing the Database](#)

Importing Information into Board/substrate Databases

After you have created the PDIF database file, you are ready to run the `pcad in` program. Perform the following steps:

1. Run `pcad in`.
The PCAD IN dialog box appears.
2. In the input file field, enter the full path to the PDIF file to be translated.
3. In the aperture field, enter the name of the aperture file. This is a file that contains all aperture information for flashes. This file is only required if you have flashes in a version 4 or 5 database. Version 6 and 7 PDIF files contain the aperture information.
4. In the *Options* field, enter the full path to an options file. The options file must exist in order for the translator to run. The translator saves all translator options (described later) to this file. Later translations of the same or different ASCII file can be performed using this option file.
5. In the Drill Table field, enter the name of the PCAD drill table file that contains drill sizes for each pad code.
6. In the Output field, enter the full path to where all output files are to be written.
The translator requires that the current working directory must be set to the output directory. For example, if you are running PCAD IN from the Program Manager, make sure the program item working directory is set to the output path. If you are running PCAD IN from a command console, make sure the current directory is the output path.
7. Select the checkbox *Show options dialog* to open the options dialog box.
8. Click Run .
The translator starts.

9. To view error messages or other process information, click *File – Viewlog*.

Related Topics

- [pcad in](#)
- [PCAD IN Dialog Box](#)

Editing the Database

After the translation is complete, load the board/substrate file. We recommend that you perform the following steps to create a design that can be maintained completely within the editor:

1. Run `viewlog` in the editor to display the `pcad.in.log` file. Examine the file for any errors or warnings. Also examine `netin.log` file for any warnings or errors.
2. Set the colors, visibility, and color priorities in the design.
 - a. Run `color192`.
The Color dialog box appears.
 - b. To make all classes invisible, in Global Visibility area, click Off, and then click Yes in the confirmation box
 - c. Click *Apply*.
All the classes and subclasses in the design become invisible.
 - d. Turn on visibility for the following classes/subclasses:
Drawing Format/Outline in the Manufacturing group
Pin, Via, DRC, and Etch/Conductor in the Stack-Up group
 - e. Set colors for the above classes/subclasses.
Drawing Format/Outline in the Manufacturing group
Pin, Via, DRC, and Etch/Conductor in the Stack-Up group
Ratsnest in the Display group
Note: We recommend that, for ease of viewing, you set DRC and ratsnests in red.
 - f. Click *Apply*.
The reset classes/subclasses in the design become visible.
 - g. Click *OK* to close the *Color – Visibility* dialog box.
3. Run `prmed` to display the *Design Parameter Editor*, choose the *Display* tab, and clear *Grids on*.
The grid display in the interface disappears.
 - a. Set *DRC Marker Size* to 125 (or a parameter of your own choosing).
 - b. Click *OK* to implement the changes and close the dialog box.
4. Choose the *Display* tab and set *DRC Marker Size* to 125 (or a parameter of your own choosing).
5. Verify the keepin/keepout areas. (The translator creates placement keepouts for all package/part symbols based on the boundaries of objects found within the PADS decal.)
6. Set the appropriate constraints for your design.
7. Perform a database check/fix.
 - a. Run `dbdoctor`.
The Dbdoctor dialog box appears.
 - b. Run the database check program.
 - c. When completed, check the results of the run by viewing the log file.
 - d. Make the necessary corrections.

The translator also adds an anti-pad and thermal-relief layer entries to pad stacks. Thermal reliefs are a flash with the same name as the pad. You may modify these settings. The translator determines the size of anti-pads using the COPPOURSPACE parameter in the *PCB* section of the ASCII file. Any oval or rectangular finger pad entries are converted to shapes in the design database. This is done because PADS allows ovals and fingers to be rotated within a padstack.

If your PADS database contained negative power planes with power ties, generate these ties as thermal-reliefs during film creation.

Related Topics

- [pcad in](#)
- [PCAD IN Dialog Box](#)
- [Creating a PCAD PDIF Database File](#)

pdf out

The `pdf_out` command is used to export the physical design as a PDF file. You can invoke this command from the GUI or in the batch mode.

 You need to define Art films for the design, before you can export design to PDF. See [artwork](#) for more details.

Syntax

```
pdf_out <design_name> [-slBCrhpPtUnmiveS] [-f <art_film_name1> -f <art_film_name2>...]  
[-c <config_file_name>] [-o output_name] [-u user_pass] [-w perm_pass]
```

Enter `pdf_out -help` on the operating system prompt to see details on using this command in the batch mode.

Related Topics

- [Exporting Physical Design Information](#)

Allegro PDF Publisher Dialog Box

Access Using

- *Menu Path: File — Export — PDF*

PDF Export Tab	
<i>Output file name</i>	Enter the name of the output PDF file.
<i>Available films</i>	Select the art films to export to PDF.
<i>Select All</i>	Click to select all art films.
<i>Clear All</i>	Click to clear the selection.
<i>Export Options</i>	
<i>Export board outline, symbol outline, refdes if pin exported</i>	Check to board outlines, symbol outlines, and Refdes if you add PIN/ SUB CLASS entry in the Artwork.
<i>Filled Pads</i>	Check to fill pins and vias on export.
<i>Filter Holes</i>	Check to exclude hole symbols during export.
<i>Filter Traces</i>	Check to exclude any traces or connection lines.
<i>Filled shapes</i>	Check to blank out any filled spaces.
<i>Filter drawing origin</i>	Check to exclude drawing origin.
<i>Filter header/footer</i>	Check to exclude header/footer.
<i>Property Options</i>	Check to specify property export options.
<i>Comp Tree</i>	Creates component data model tree and display properties.
<i>Net Tree</i>	Creates net data model tree and display properties.
<i>TestPoint Tree</i>	Creates test point tree and outline.
<i>Pin</i>	Creates link data annotation for pin.
<i>Via</i>	Creates link data annotation for via.
<i>Cline</i>	Creates link data annotation for cline.
<i>Shape</i>	Creates link data annotation for shape.
<i>Security</i>	Check to create a password protected PDF.
<i>Document open password</i>	Specify the password required to open the PDF file.
<i>Permissions password</i>	Specify the password required to open and modify the PDF file.
<i>Output PDF in black and white mode</i>	Check to create a monochrome PDF file, for example, to minimize ink usage. If not checked, the PDF file uses the colors as specified in Allegro.
<i>Create separate PDF file for each art film</i>	Creates a separate PDF file for each art film. By default one PDF file is created for all exported art films. The output file name for each film will be <output_file_name>_<film_name>.pdf or <prefix>_<output_file_name>_<film_name>_<suffix>.pdf. For large designs, a single PDF file that contains all films pages may become very big in size. To output these designs, you may need to separate each film to its own PDF file.

<i>Create PDF optimized for print (no metadata but smaller files)</i>	Creates a size optimized PDF file for printing. No design data is exported.
<i>Display Component by RefDes without the package name</i>	By default, the component RefDes are listed under its parent node - package name.
<i>Launch PDF Viewer</i>	Open the output PDF file.
Property Parameters tab	
	Lists the component and net properties to be exported to PDF.
<i>Right-click — Add</i>	Use to add a property.
<i>Right-click — Remove</i>	Use to remove a property.
Page Setup tab	
	Use this tab to specify the page options for the PDF file. These settings are optional. Allegro uses the actual sizes to generate PDF pages if these settings are not specified.
<i>Unit</i>	Specify the measurement unit for the page. You can specify millimeter or inches. All the values displayed in this tab are shown in the units specified.
<i>Paper size</i>	Choose a value from the standard paper sizes. The <i>Width</i> and <i>Height</i> fields display the values of the selected paper size. Alternatively, choose <i>Custom Size</i> and specify the dimensions in the <i>Width</i> and <i>Height</i> fields.
<i>Width</i>	Displays the width of the selected paper size.
<i>Height</i>	Displays the height of the selected paper size.
<i>Orientation</i>	Specify the orientation of the paper.
<i>Portrait</i>	Sets the paper orientation to partite.
<i>Landscape</i>	Sets the paper orientation to landscape.
<i>Margins</i>	Use these fields to specify the margins.
<i>Left</i>	Sets the distance between the left edge of the page and the left edge drawing area.
<i>Right</i>	Sets the distance between the right edge of the page and the right edge drawing area.
<i>Top</i>	Sets the distance between the top edge of the page and the top edge drawing area.
<i>Bottom</i>	Sets the distance between the bottom edge of the page and the bottom edge drawing area.
<i>Scaling</i>	
<i>Scale Factor</i>	Specify the scale factor to resize the design.
<i>Fit to page</i>	Adjust the document to fit in the selected paper size.
<i>Text Size for Header/Footer</i>	Use these fields to set the text size for header/footer
<i>Width</i>	Displays the width of text in header/footer
<i>Height</i>	Displays the text height in header/footer
<i>Export</i>	Click to export the PDF.
<i>Close</i>	Click to close the Allegro PDF Publisher dialog box.
<i>Viewlog</i>	Click to view the log file.

Exporting Physical Design Information

Follow these steps to export physical design information:

1. Choose File – Export –PDF. The *Allegro PDF Publisher* dialog box displays.
2. In the *Output file name* field specify the PDF file name.
3. From the *Available Films* list, select the Art films to export.
4. Specify the *Export Options*.
5. Optionally, to specify property parameters:
 - a. Click the *Property Parameters* tab.
 - b. Right-click and choose *Add Property*. The *Property Selection* dialog box displays.
 - c. Select the properties to add and click *OK*.
6. Optionally, to specify Page settings:
 - a. Click the *Page Setup* tab.
 - b. Specify the units, paper size, orientation, margins, and scaling.
7. Click *Export* to export the PDF. The design is exported as PDF.

For more information and for help on viewing the exported PDF file see the [Allegro Design Publisher User Guide](#).

Related Topics

- [pdf out](#)

phase_tune

The `phase_tune` command lets you add phase bumps at the proper locations to either member of the differential pair, so that any existing phase-tune drcs are eliminated post-pick.

Related Topics

- [Adding Phase Bumps to Differential Pairs](#)

Phase Tune Command: Options Panel

Access Using

- *Menu Path: Route – Phase Tune*

<i>Bump Style</i>	Defines bump as line or arc.
<i>Bump Length</i>	Defines the length of each bump. This length determines the minimum segment length between the new bump and the vertices of the original selected segment. The default value is two-times the minimum-line-width from the design cset.
<i>Bump Height</i>	Defines the width of each bump. The default value is two-times the minimum-line-width from the design cset.
<i>Length added per bump</i>	Shows the length each bump will be added to the cline. This value varies depending upon the value of bump height.

Adding Phase Bumps to Differential Pairs

To add phase bumps to differential pairs, follow these steps:

1. Choose command `phase_tune`.
The *Find* panel displays clines as the active design object. The *Options* panel displays the bump height, bump length and bump style.
2. Choose a single straight line cline segment which is a part of differential pair. The command creates the bump in the direction away from the mate.
3. Right-click to display the pop-up menu and choose Done to terminate the command.

⚠ If you choose bump style as line, on adding two or more line bumps next to each other, the bumps will be spaced apart by the bump length distance. While in case of arc style, the bumps will connect with no gap, so the arc of first bump connects directly with the arc of next bump.

Related Topics

- [phase_tune](#)

pick

The `pick` command, run at the command window prompt, lets you enter screen coordinates from the keyboard to find and highlight objects. If you do not provide any coordinates, the Pick dialog box appears. The picks are absolute values and are not snapped to grid.

Syntax

The format is as follows (all coordinates are in database units):

```
pick x y
```

Related Topics

- [Highlighting Objects](#)

Pick Dialog Box

The *Pick dialog box* appears when you use these commands: `pick`, `ipick`, `pick_to_grid`, and `ipick_to_grid`. You can also display the Pick dialog box when you are in an active command and you click the *P* button in the Status bar. See the *Getting Started with Physical Design* user guide in your documentation set.

Type	Select whether you want to specify cartesian coordinates (<i>XY Coordinate</i>) or polar coordinates (<i>Distance+Angle</i>).
Value	Type the x and y coordinates for the task you are performing if you choose <i>XY Coordinate</i> or type the distance and the angle if you choose <i>Distance+Angle</i> .
Snap to current grid	Check this box to have the editor pick the location on grid closest to the location specified by the coordinates. If you are using the <code>pick_to_grid</code> or the <code>ipick_to_grid</code> commands, this box is already checked.
Relative from last pick	Check this box to have the editor determine the location from the last pick, using the values or incremental coordinates you specified. If you are using the <code>ipick</code> or the <code>ipick_to_grid</code> commands, this box is already checked.
OK	Click to apply the parameters and close the dialog box.
Cancel	Click to close the dialog box without saving changes.

Highlighting Objects

Perform the following 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 `pick`, or click the *P* button in the status window.
The Pick dialog box appears. When you click the *P* button to display the Pick dialog box, the Pick dialog box remains displayed until you dismiss it.
3. Type the coordinates in the *Value* field. Be sure to leave a space between the numbers.
4. Click *OK* to dismiss the dialog box and establish the point.

 You can also type the coordinates on the command line after typing the command name. For example:

```
pick 200 -350
```

 To specify polar coordinates, use the `apick` command.

Related Topics

- [pick](#)

pick_origin

The `pick_origin` command is used in macro script files and is automatically recorded in a macro file at the first mouse click. When encountered in a script replay, it prompts you for an origin point that can be entered through either a mouse click or the keyboard. All subsequent `ipick_to_grid` or `pick` commands are originated to the user-chosen origin. If coordinates (in database units) are provided as arguments, they are used as the origin, and no prompt is issued.

Example

```
pick_to_grid [+ -] x [+ -]
```

```
ipick_to_grid [+ -] x [+ -] y
```

pick_to_grid

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

 Use the `pick_to_grid` command to specify polar coordinates.

Example

```
pick_to_grid x y
```

pin_delay in

The `pin_delay in` command is used to import pin-delay values, defined by the PIN_DELAY property, from one design and assign them to component instance pins in another design. The file format is:

```
[PIN_DELAY]

[REFDES  <refdes>]

[DEVICE  <package name>]

[UNITS   <mks units>]

<Pin number>  <delay value>  <...>

{repeat pin/delay value for each pin with a delay where delay can be a unit or unitless value in time or length}
```

File Format Example

The following is an example of the format of a pin-delay file, with the *Include Header* field enabled during export with the `pin_delay out` command.

```
PIN_DELAY

REFDES U1

DEVICE DIP14

1      1 MIL

12     12 MIL

2      2 MIL

8      8 MIL
```

Related Topics

- [Importing Pin Delays into a Design](#)
- [Updating Pin Delays in a Design using Constraint Manager](#)
- For more information, see [Creating Design Rules](#) in the user guide.

Pin Delay Import Dialog Box

Access Using

- *Menu Path: File – Import – Pin Delay*

<i>Pin Delay File</i>	Enter the name of a file containing pin-delay values, or click ... to locate an existing pin-delay file.
<i>Refdes Name</i>	In conjunction with the <i>Package Name</i> , used to initially match the <i>Refdes Name</i> in the pin-delay file with that of a component in the design. The initially chosen component highlights in the design, but you can override it and choose another to which to apply the pin delays in the design.
<i>Package Name</i>	In conjunction with the <i>Refdes Name</i> , used to initially match the <i>Package Name</i> in the pin-delay file with that of a component in the design, to which to apply the imported pin delays. The initially chosen component highlights in the design, but you can override the chosen component and choose another.
<i>Delay Units</i>	Displays the measurement units only if you disabled the <i>Include Units on Each Pin</i> field on the <i>Pin Delay Export</i> dialog box when you created the pin-delay file using the <code>pin_delay out</code> command.
<i>Import</i>	Click to assign the pin-delay values contained in the pin-delay file you entered in the <i>Pin Delay File</i> field to the chosen component instance pins in the current design.
<i>Close</i>	Closes the dialog box and exits the command.

Related Topics

- [Updating Pin Delays in a Design using Constraint Manager](#)

Importing Pin Delays into a Design

Perform the following steps to import pin delay values into your design:

1. Choose *File – Export – Pin Delay*.

Alternately, run `pin_delay out`. The following message appears in the command window prompt:

Select a component

2. Choose the component whose pin-delay values you want to write to a pin delay file.

The *Pin Delay Export* dialog box appears.

⚠ If you choose a component without pin delays (that is, no PIN_DELAY property is attached to any of its pins), the following message appears in the command window prompt:

No pin delays on component

Add the PIN_DELAY property using the [property edit](#) command or choose another component with pin-delay values, as defined by the PIN_DELAY property.

3. Enter the name of an existing pin-delay file or click... to locate an existing pin-delay file.
4. Choose a format in which to write the contents of the pin-delay file.
5. Choose whether to include the header in the pin-delay file.
6. Choose whether to include the units on each pin.
7. Click *Export* to create the pin-delay file.
8. Choose *File – Import – Pin Delay*.

Alternately, run `pin_delay in`. The *Pin Delay Import* dialog box appears.

9. Enter the name of an existing pin-delay file or click ... to locate an existing pin-delay file. Once you choose or enter the name of a pin-delay file, the *Refdes Name* and *Package Name* fields are populated.
10. Choose a component to which you want to assign the values in the chosen pin-delay file by pick or using the Find Filter tab, or click *Import* to read in the pin delays. The message box displays:

Finished `[name of component]`, select another component or close to exit the dialog box.

⚠ If pins referenced in the file are not on the component, then the following message appears. For example,

Pin `[number]`, not found for component `[name]`.

Pin `[number]`, not found for component `[name]`.

After applying all pin delays, you can choose another component instance to which you want to apply pin delays, or click *Close*.

✓ The layout editor does not remove pin delay values for the pins that are not specified in the import file. If you want to remove pin delays from certain pins, set their pin delay values as '0' in the import file.

Related Topics

- [pin_delay in](#)

Updating Pin Delays in a Design using Constraint Manager

To update pin delay values for all pins in your design, perform the following steps:

1. Open Constraint Manager and navigate to the *Properties – Component – Pin Properties* sheet.
2. Choose *File – Export – Worksheet File*.
The *Export Worksheet File* dialog box appears.
3. Enter the name of an existing pin-delay file or locate it in the file browser.
Alternately, if you're creating a new file, enter a name for the file you want to create.
4. Choose a file type for the export file; *CSV Comma delimited (*.csv)*, *Test Tab delimited (.txt)*, or *Formatted Test Space delimited (.prn)*.
5. Click *Save* to create the pin-delay file.
6. In the exported file, edit the pin delay values as required and remove the delays that are no longer needed. Save the changes to the file.
7. In Constraint Manager, choose *File – Import – Worksheet File*.
Navigate to the exported file that you've updated the pin delay values in, and click Open to import the file.
All the pin delay values are updated in your design.

Related Topics

- [pin_delay in](#)
- [Pin Delay Import Dialog Box](#)

pin_delay out

The `pin_delay out` command is used to create a file containing pin-delay values, as defined by associating the PIN_DELAY property with particular component pins that exist on component-instance pins in one design. You can then use the `pin_delay in` command to import the values in this file and assign them to component-instance pins in another design.

You can export the file in Comma Separated Value (`.csv`) or in Tab Separated (`.xls`) format. If the pin delay cells in the file do not have units of measure associated with them, and the file has no UNITS header, values default to the current design units in length.

The file format is:

```
[PIN_DELAY]  
  
[REFDES  <refdes>]  
  
[DEVICE  <package name>]  
  
[UNITS   <mks units>]  
  
<Pin number>  <delay value>  <...>  
  
(repeat pin/delay value for each pin with a delay where delay can be a unit or unitless value in time or length)
```

A pin without a pin-delay value assigned to it (that is, no PIN_DELAY property attached to it), is excluded from the file. Additional columns may appear in the file after the delay value but are ignored.

Related Topics

- [Exporting Pin Delays](#)

Pin Delay Export Dialog Box

Access Using

- *Menu Path: File – Export – Pin Delay*

<i>Pin Delay File</i>	Enter the name of a file to which to save pin-delay values or click ... to locate an existing pin-delay file.
<i>Export Format</i>	
<i>Tab Separated</i>	Choose to create a pin-delay file in an Excel (.xls) format, which uses a tab to separate each field within the record.
<i>CSV (comma)</i>	Choose to create a pin-delay file in a Comma Separated Value (.csv) format. Each line is a separate data record, and a comma separates each field within the record. All records have the same number of fields. The file's first line is the header row, which specifies the names of each field. CSV uses a double quote if the value contains a comma. CSV is the default file extension.
<i>Include Header</i>	Choose to display the <i>Refdes name</i> , <i>Package Name</i> , and the unit of measure of the component's first pin in the file header.
<i>Include Units on Each Pin</i>	Choose to display the unit of measure with each pin in the pin- delay file. If you choose to exclude the unit of measure per pin, then the file header displays the unit of measure of the component's first pin. If units of measure on component pins contain delay values in both length and time, then you must enable this field to include the unit of measure with each pin in the pin-delay file.
<i>Refdes Name</i>	Identifies the component instance whose pin-delay values are used to create the pin-delay file for export to another design.
<i>Package Name</i>	Identifies the package name of the component instance whose pin-delay values are used to create the pin-delay file for export to another design.
<i>Export</i>	Click to create a file containing pin-delay values of the chosen component.
<i>Cancel</i>	Ignores input and closes the dialog box.

Exporting Pin Delays

Perform the following steps to export pin delays:

1. Run `pin_delay out`. The following message appears in the command window prompt:

Select a component

2. Choose the component whose pin-delay values you want to write to a pin delay file.
The *Pin Delay Export* dialog box appears.

 If you choose a component without pin delays (that is, no PIN_DELAY property is attached to any of its pins), the following message appears in the command window prompt:

No pin delays on component

Add the PIN_DELAY property using the [property edit](#) command or choose another component with pin-delay values as defined by the PIN_DELAY property.

3. Enter the name of an existing pin-delay file or click ... to locate an existing pin-delay file.
4. Choose a format in which to write the contents of the pin-delay file.
5. Choose whether to include the header in the pin-delay file.
6. Choose whether to include the units on each pin.
7. Click *Export* to create the pin-delay file.
8. Run `pin_delay in`. The *Pin Delay Import* dialog box appears.
9. Enter the name of an existing pin-delay file or click ... to locate an existing pin-delay file. Once you choose or enter the name of a pin-delay file, the *Refdes Name* and *Package Name* fields are populated.
10. Choose a component to which you want to assign the values in the chosen pin-delay file by pick or using the *Find Filter* tab, or click *Import* to read in the pin delays. The message box displays:

Finished [name of component], select another component or close to exit the dialog box.

 If pins referenced in the file are not on the component, then the following message appears:

Pin [number], not found for component [name].

Pin [number], not found for component [name].

11. After applying all pin delays, you can choose another component instance to which you want to apply pin delays, or click *Close*.

Related Topics

- [pin_delay out](#)

pipe

The `pipe` command lets you run programs from the command console of your Cadence user interface that you would normally run from your operating system command line.

Example

`pipe a2dxf`

pkg ic overlay export

The `pkg ic overlay export` command generates an XML file that contains package overlay information. You can select layers of a package to be exported and specify the objects in the selected layers to be exported. You can also configure the area to be exported.

An IC designer can then use the `readPackage` command in EDI in the Floorplan view to load the exported XML to view it as a read-only package overlay.

The command is only available for co-design dies and is available in tools that support co-design dies.

Related Topics

- [Using Package Overlay Export](#)
- [Viewing Package Overlay in EDI](#)

The Write Package Overlay for IC Dialog Box

Access Using

- *Menu Path: File – Export – Package Overlay File for IC*

Die instance for package overlay generation	Select the co-design die instance relative to which you want the overlay. This field lists all the co-design dies available in the design. Default is the first die in the list. The IC design name is displayed in parentheses after the reference designator. To export overlays in context of more than one die, generate XML files for each die.
File name	Specify the name of the XML file to be generated. Default is <IC design name>_pkg_abstract.xml.
Display area	Configure the area which should be exported to the XML file. You can select one of the following options: <ul style="list-style-type: none"> • <i>Package Outline</i>: Define the area in terms of the package outline and an offset value specified in <i>Offset from edge</i>. The offset is a negative value for Package Outline. For example, if the package outline is (-5000 -5000) (5000 5000), specifying -100 UM offset will include only those objects inside the window with extents (-4900 -4900) (4900 4900). • <i>Die Outline</i>: Define the area in terms of the die outline and an offset value specified in <i>Offset from edge</i>. For example, if the die outline is (-1000 -1000) (1000 1000), specifying 100 UM offset will include only those objects inside the window with extents (-1100 -1100) (1100 1100). • <i>Window</i>: Define a rectangular area by drawing a window on the canvas. By default <i>Package Outline</i> is selected.
Offset from edge	Specify the offset value to be used to determine the area if you select either <i>Package Outline</i> or <i>Die Outline</i> for Display Area. The default value is 0 UM. Set a negative value for Package Outline and a positive value for Die Outline.
Routing Layers	Select the layers to be exported to the XML file. By default, the layer the die mounts to is the only one selected. You can select a different layer or select multiple layers from the listed layers. Check <All Layers> to select all the layers.
Package balls	Select to export the package ball information. This is selected by default.
Flip-chip package pads	Select to export the package-side pads that the flip-chip solder balls mount. These may be of a different size or shape than the pads on the die for which the overlay is being generated. This also includes pads for other components on the specified routing layers. This is selected by default.
Bond fingers	Select to include bond fingers in the XML file. This is selected by default.
Bond wires	Select to export all bond wires connected to pins of the reference die. This is selected by default.
Routing clines	Select to exports all clines that are on layers indicated in the routing layers list. This includes any fillets, if applicable. This is selected by default.
Via pads	Select to export all via pads that are on layers indicated in the routing layers list. This is selected by default.
Ratsnests (flight lines)	Select to export the flight lines, for a net that is not completely routed, between the end of the routing and the BGA ball or other item in the package netlist to which this connects. This is selected by default.
Symbol outlines	Select to export the outline of additional components mounted to the same layer(s) as the reference die. This is selected by default.
Shapes	Select to export shapes (such as power and ground planes, rings and flags for wire-bonded components) on the specified routing layers. This is selected by default.
Include voids	Select to export the void areas inside the outline of the shapes, rather than showing the shape as solid filled. This is available only if Shapes is selected. This is selected by default.
Write	Click to write the XML file based on the current form configuration and exit the dialog box.
Close	Click to close the dialog box without writing an XML file.

Related Topics

- [Viewing Package Overlay in EDI](#)

Using Package Overlay Export

Follow these steps to export package overlay information:

1. Choose *File – Export – Package Overlay File for IC*
The Write Package Overlay for IC dialog box appears.

2. Select the Die Instance.
3. Specify the XML file.
4. Select the display area and the offset, if necessary.

5. Specify the routing layers.
6. Select the object types to be exported.

By default all object types, such as vias, pads, and shapes, are selected.

7. Click Write.

The XML file is generated and the dialog box is closed.

If optical shrink is specified for the selected die, the package is expanded in EDI.

Objects written to the XML file are oriented relative to a chip-up, unrotated version of the die, as it will be viewed related to the IC design (definition of the die).

Related Topics

- [pkg ic overlay export](#)

Viewing Package Overlay in EDI

Use the `readPackage` command in the Floorplan view of EDI to load the XML file. The syntax of the command is:

```
readPackage <XML File Name>
```

After loading the file, select bumps to view the flight lines.

To control the layer display in Encounter:

1. Click All Colors to open the Color Preferences window
2. Activate the Custom tab.
It lists all the objects imported from the XML file.

You must exit and start a new session to load another XML file as the package data cannot be reset by the `freeDesign` command.

Note that:

- The package balls can be correctly displayed in Encounter even when the design is a flip-chip design.
- The dummy balls are displayed to provide an overall view of the package.
- The flight lines are displayed from the balls to the bumps instead of the IO pads.

Related Topics

- [pkg ic overlay export](#)
- [The Write Package Overlay for IC Dialog Box](#)

place area design

The `place area design` command sets the automatic placement mode to PLACE WITHIN BOARD and sets the place area to the package/part keepin. It then runs automatic placement in interactive mode.

For more details and the prerequisites for this command, see the *Placing the Elements* user guide in your documentation set.

Access Using

- *Menu Path: Place – Autoplace – Design*

Running the Place Area Design Command

1. Run `place area design`.

When placement is complete, you can display the placement log to review any warning or error messages.

place area list

The `place area list` command displays the LIST AREA window showing the current active area of the design for automatic placement.

For more details, see the *Placing the Elements* user guide in your documentation set.

Access Using

- *Menu Path: Place – Autoplace – List*

place area room

The `place area room` command lets you define a room into which automatic placement places components. The room must be located in a package/part keepin.

- ✓ If the room is not located in package/part keepin, the autoplace will not work. For example, if the room is present in a mechanical symbol, no autoplace of components takes place.

For more details, see the *Placing the Elements* user guide in your documentation set.

Access Using

- *Menu Path: Place – Autoplace – Room*

Placing Components in a Room

Follow these steps to place components in a room:

1. Run `place area room`.

The Room browser displays, which lists rooms defined in the design (using the `add rect` command described in the *Allegro PCB and Packaging Physical Layout Command Reference*).

2. Select a room name from the list and click *OK*.

The editor sets the automatic placement mode to PLACE BY ROOM mode.

3. To place components automatically in that room, run one of these commands:

- `place param` (automatic mode)
- `place execute` (automatic mode)
- `place interactive` (interactive mode)

place area window

The `place area window` command lets you define a window into which automatic placement places components. The window must be located in a package/part keepin.

For more details, see the *Placing the Elements* user guide in your documentation set.

Access Using

- *Menu Path: Place – Autoplace – Window*

Defining a Window and Placing Components

Perform these steps to define a window and place components within that window:

1. Run `place area window`.
2. Click to define one corner of a rectangular window.
3. Slide the cursor to expand the window and click again to define the diagonally opposite corner.
4. Click the right mouse button to display the pop-up menu and choose *Done*.
5. To place components automatically in that window, run one of these commands:
 - [place param](#) (automatic mode)
 - [place execute](#) (automatic mode)
 - [place interactive](#) (interactive mode)

place auto bottomgrid

The `place auto bottomgrid` command generates a grid on the bottom subclass of your design, based on the size of the most common component in your design. To run, the program must detect unplaced components in the database, components with a PLACE_TAG property assigned, and a package keepin. Missing items result in an error message.

 You can set your bottom placement grid interactively with [place set bottomgrid](#).

When you run `place auto bottomgrid`, two dialog boxes appear, prompting you for the horizontal and vertical extents of the component you plan to next place.

Generating a Grid on the Bottom Subclass of Your Design

To generate a grid on the bottom subclass of a design, follow these steps:

1. Run `place auto bottomgrid`.

You are prompted for the horizontal extents of the component you plan to next place in the placement area.

2. Enter the width of the most common component.

You are prompted for the vertical extents of the component you plan to next place in the placement area.

3. Enter the length of the most common component.

The defined placement grid is drawn, added to the design on the subclass you chose.

place auto topgrid

The `place auto topgrid` command generates a grid on the top subclass of your design, based on the size of the most common component in your design. To run, the program must detect unplaced components in the database, components with a PLACE_TAG property assigned, and a package keepin. Missing items result in an error message.

 You can set your top placement grid interactively with [place set topgrid](#).

When you run `place auto topgrid`, two dialog boxes display, prompting you for the horizontal and vertical extents of the component you plan to place.

Generating a Grid on the Top Subclass of Your Design

To generate a grid on the topsubclass of a design, follow these steps:

1. Run `place auto topgrid`.

You are prompted for the horizontal extents of the component you plan to next place in the placement area.

2. Enter the width of the most common component.

You are prompted for the vertical extents of the component you plan to next place in the placement area.

3. Enter the length of the most common component.

The defined placement grid is drawn, added to the design on the subclass you chose.

place execute

The `place execute` command performs automatic placement in automatic mode.

It uses the settings on the Automatic Placement dialog box ([place param](#) command), properties assigned to components and nets, and the boundary defined by the place area commands ([place area design](#), [place area list](#), [place area room](#), [place area window](#)).

For more details and the prerequisites for this command, see the *Placing the Elements* user guide in your documentation set.

Performing Automatic Placement in Automatic Mode

1. On the command line of the tool's command window, run `place execute`.
When placement is complete, you can display the placement log to review any warning or error messages.

place interactive

The `place interactive` command performs automatic placement in interactive mode.

It uses the settings on the Automatic Placement dialog box ([place param](#) command), properties assigned to components and nets, and the boundary defined by the place area commands ([place area design](#), [place area list](#), [place area room](#), [place area window](#)).

For more details and the prerequisites for this command, see the *Placing the Elements* user guide in your documentation set.

Access Using

- *Menu Path:* Place – Interactive

Performing Automatic Placement in Interactive Mode

The `place interactive` command performs the following steps for each component with a PLACE_TAG property attached:

1. Selects the next component for placement.
2. Calculates the best available location.
3. Temporarily places the component.
4. Waits for you to do one of the following:
 - Accept its component choice and location by clicking *Next* on the pop-up menu.
 - Accept the component choice but edit the position by moving, rotating, or mirroring with pop-up menu commands or by moving the cursor.
 - Reject the component choice and continue by clicking *Skip* on the pop-up menu.

place manual

The `place manual` command lets you place components, symbols, and module instances or definitions in your design. Various control settings specify which element sets are displayed and what actions you are allowed to perform during placement.

For more details, see the *Placing the Elements* user guide in your documentation set.

Syntax

```
place manual [<type> <name>]
```

<type>	Specifies the kind of design element you are placing. The choices are:	
	<ul style="list-style-type: none">• refdes• package• board	<ul style="list-style-type: none">• mech• moddef• modinst
	For definitions, see the description of <i>Select elements for placement</i> in Placement List Tab . ⚠ For a reference designator or a module instance, the item must be unplaced.	
<name>	Specifies the name of the design element you are placing.	

Related Topics

- [Placement Dialog Box](#)
- [Placing Components](#)
- [Placing Symbols](#)
- [Placing Alternate Symbols](#)
- [Placing Modules](#)
- [Rotating During Placement](#)
- [Mirroring During Placement](#)

Place Manual Command: Options Panel

Access Using

- Menu Path: Place – Manually



- Toolbar Icon:

Active Class and Subclass	Specifies the class and subclass on which you are placing components.
Unplaced symbols	Indicates the number of unplaced components.
Mirror	Relocates symbols you add to a drawing to the opposite side of the board/substrate (layer change). By default, this option is unchecked (symbols are not mirrored). Using the symbol origin as the location point, the tool anchors the symbol at the same grid point. When this option is checked, any test, etch/conductor, or vias built with the symbol are mirrored, and any attached connections and ratsnest lines become dynamic rubberband lines. This option is also available on the <i>Design</i> tab of the <i>Design Parameter Editor</i> , available by choosing <i>Setup – Design Parameters</i> (prmed command).
Mirror geometry	Creates an element (or a group of elements) on the current subclass layer that is a mirror image of the original, around the Y-coordinate of the copy origin (no layer change). Valid elements are: <ul style="list-style-type: none"> ▪ Vias (their padstacks are not mirrored) ▪ Connect line and line segments ▪ Rectangles ▪ Text ▪ Shapes
Rotation	Determines the mode of rotation.
Type (Incremental)	Provides a dynamic handle for controlling the element. It uses the number in the <i>Angle</i> field as the amount by which to increment the element as you rotate it.
Angle	Determines the angle of rotation, specifying how many degrees compose each increment as you rotate the element. You can enter a number between 0 and 360 or choose 45-degree increments from the pop-up menu. The program is accurate up to three decimal places.
Point (Sym Origin)	Indicates the 0,0 point of the element.

Related Topics

- [Placing Components](#)
- [Placing Symbols](#)
- [Placing Alternate Symbols](#)
- [Placing Modules](#)
- [Rotating During Placement](#)
- [Mirroring During Placement](#)

Placement Dialog Box

Use the Placement dialog box to place components, symbols, and modules into your design. Parameters that you set in the *Advanced Settings* tab determine which objects appear in the *Placement List* tab and how to place the objects.

-  If you are working in placement edit application mode, the *Options* panel displays a dockable version of the Placement List tab. Choose *Setup – Application Mode – Placement Edit* to access the placement edit application mode.

Placement List Tab

	Shows, in a collapsing tree view, all elements that you can place in your design. Each element type displays zero or more elements. Where some number of elements are listed for a type, the display area shows a folder icon. You can choose all elements of a type by clicking the check box next to the icon or individual elements by clicking the check box next to the element name. The elements you choose are set on the cursor one at a time, as you place them. Elements in this tree must match the requirements of both <i>Selection filters</i> on the right of the dialog box before you can choose them.
<i>Components by refdes</i>	Symbols that are associated with logic. Only unplaced components are displayed. These icons may contain a superimposed "M," indicating a component that is part of a module instance.
<i>Components by net group</i>	Displays all the components that are associated with a net group. If a component is part of more than one net group, it is listed under each net group.  For components that are part of multiple net groups, selecting a component under one net group will select all instances of the component. Until all the components are placed, a net group continues to appear in the list. In such partially placed net groups, placed components are indicated with  (green P).
<i>Module instances</i>	Modules that are already in the design (that is, brought in from the schematic) but unplaced.
<i>Module definitions</i>	Modules that have been created during a design session (that is, not brought in from the schematic). Can be displayed from the database, a library, or both. Modules must be in the module library path (modulepath) in order to be placed into the design.
<i>Package Symbols</i>	Represents the component package/part.
<i>Mechanical Symbols</i>	Represents a mechanical element; for example, the design outline
<i>Format Symbols</i>	Represents the drawing format. Symbols must be in the symbol library path (PSMPATH) in order to be placed in the design.
<i>New Component</i>	Display list of all library device files that are used in the design and the associated package symbols in the design. When placed, the new components are assigned reference designator that has the same prefix as the selected component type and uses the next available unused number. These changes to the netlist cannot be fed back to the schematic and require manual updates in the logic design. Using this option, you can place multiple copies of the selected component type with incremental reference designator values and add them to the design database. By default, this option is not available. Set an environment variable <i>logic_edit_enabled</i> to view this option.  Device files are loaded only when this option is selected for the manual placement of components.
<i>Selection filters</i>	Lets you determine placement options by limiting the elements available for selection.
<i>Match</i>	Lets you choose only the element name you enter in the field.
<i>Property/Value</i>	Places components by a property and its value. The pull-down list displays all properties, including user-defined, assigned to unplaced component instances and definitions in the current design.

<i>Room</i>	The pull-down list displays all values for the chosen property name, including a blank value, that places all unplaced component or definitions with the associated property whatever its value. Places components in one or all rooms simultaneously. If the room is too small for all components, the layout editor allows overlapping. Choosing <i>Place by Room</i> disables options in the <i>Placement Position</i> section.
<i>Part #</i>	Places all unplaced components with the part number you specify. The layout editor supports wildcard characters (* and ?).
<i>Net</i>	Places all unplaced parts associated by the chosen net name. Click the <i>browse</i> button to display the <i>Select Net Name</i> data browser from which you can choose a net name. You can use wildcard characters to specify multiple nets.
<i>Net group</i>	Places all unplaced parts associated by the chosen net group name. Click the <i>browse</i> button to display the <i>Select net group name</i> browser from which you can choose a net group which are associated with the unplaced parts. You can use wildcard characters to specify multiple net groups.
<i>Schematic page number</i>	Places all unplaced parts simultaneously on multiple schematic pages while maintaining an update of those with unplaced components. Click the <i>browse</i> button to display the <i>Schematic Page Number</i> dialog box from which you can choose schematic hierarchical blocks and pages. This option is only available in Allegro Design Entry HDL L as the front end.
<i>Place by refdes</i>	Filters the unplaced parts as you choose.
<i>Quickview</i>	When you choose an element in the list on the left of the dialog box, displays one of the following: <ul style="list-style-type: none"> • Graphic preview depicting a high-level view of the database. Symbol graphics consist of an outline and pins. • Descriptive text related to the properties of the element. If Quickview cannot display the preview or the properties of the element, a "Not Available" message appears in the quickview window. For additional information on Quickview, see <i>Getting Started with Physical Design</i> in the user guide.

Advanced Settings Tab

Advanced settings let you set general options and control list displays.

List construction		
<i>Display definitions from</i>	Lets you choose from where symbol and module definitions are displayed: the database of the current design, the libraries, or both.	
Symbols and Module Definitions		
<i>AutoNext</i>	<i>Enabled</i>	(Default) Makes the next symbol in your placement list available for placing without the need to choose <i>Next</i> from the pop-up menu. When you are in this mode, choosing <i>Next</i> puts the next element on the cursor without placing the current one.
	<i>Disable</i>	Requires you to choose <i>Next</i> from the pop-up menu to make the next element on the list available for placement. In this mode, the component on the cursor is moved with every new pick and an instance of a non-component symbol on the cursor is placed at each new pick until <i>Next</i> is chosen.
	The behavior of <i>AutoNext</i> differs somewhat when placing non-component symbols or module definitions. For these elements, only one copy of the element is placed when <i>AutoNext</i> is <i>Enabled</i> . If <i>AutoNext</i> is disabled, every pick places a copy of the non-component symbol or module until you choose <i>Next</i> from the pop-up. When, however, a component is on the cursor, it remains on the cursor after you place it; subsequent picks unplace the component from its last place location and move it to the next pick location.	
<i>AutoHide</i>	Lets you automatically hide the Placement dialog box once you begin placing elements and re-display it once all elements have been placed. To manually re-display the Placement dialog box, right-click and choose <i>Show</i> .	
<i>Modules net exception list</i>	Lets you create or use an existing exception list for renaming nets when you are about to place a module instance previously created in your design. When a module is added to a design, all components and nets in the module are given unique names so that they do not conflict with any other design objects already present.	
<i>File</i>	In most designs, you want to make an exception to this general rule in the case of power and ground nets, which you would normally want to merge with the design. The nets you want to merge with the existing board can be added to a module net exception list, an ASCII file containing net names that is not changed when a module is added to the design. You specify this file in this field. You can either enter the list file name or use the <i>Browse</i> button to locate the file.	
<i>Create</i>	Creates list files and brings up the Select Nets dialog box from which you can create an exception list. You can also create list files typing them in with an external text editor or by using the <code>define list</code> command.	
<i>OK</i>	Saves any changes made during the session and closes the dialog box.	

Hide	Causes the Placement dialog box to disappear. To re-display the Placement dialog box, choose <i>Show</i> from the pop-up menu.
------	--

Miscellaneous Behavior

The *Placement* dialog box closes while you are placing elements if you chose *Autohide*. You can modify any of your selections as long as the command remains active.

During the placement session, if no elements have been chosen in the placement list, you can move elements directly from the design area to other locations. If elements in the list are chosen, the *Move* item in the pop-up menu lets you choose and move an existing element from the design area.

Active items in the pop-up menu can be chosen to manipulate the behavior of the element on the cursor.

Related Topics

- [place manual](#)
- [Placing Symbols](#)
- [Placing Alternate Symbols](#)
- [Placing Modules](#)
- [Rotating During Placement](#)
- [Mirroring During Placement](#)

Placing Components

To place components in your design, follow these steps:

1. Run `place manual`.
 - If you run this command from the menu path, the *Placement* dialog box appears. Continue with step 2.
 - If you run this command from the console command line, note the syntax. The design element is attached to the cursor by its symbol origin. Continue with step 4.
2. Make sure the *Type filters* field is set to *Any* and the *Advanced Settings* controls are set accordingly so that all components are displayed.
3. Choose the component name(s) you want to place. Only unplaced components are listed.
The first component that you chosen is attached to the cursor by its symbol origin.
The symbol reflects the rotated and mirrored position specified on the *Options* panel or on the *Design* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* (`prmed` command). Rubberbanding ratsnest lines indicate the connections of the component with any components already placed.
4. Change the orientation of the component, if necessary.
Rotate and *Mirror* options are available in the pop-up menu. See [Rotating During Placement](#) and [Mirroring During Placement](#) for details.
5. Choose a location for the component by doing one of the following:
 - Place the component in the desired location and click left.
 - Type the X, Y coordinates of the component location in the command line.
Specify the coordinates as follows:

`x <x coordinate> <y coordinate>`

An example of specifying coordinates is

`x 450 3600`

As soon as you place the component, a "P," indicating "placed," is superimposed on the icon for the component in the *Select elements for placement* list. You can manipulate the component before completing placement by choosing *Done*, at which point the component name disappears from the list.

1. If *AutoNext* is enabled in the *Advanced Settings* tab and you have checked more than one component (or the entire group), continue placing components until all your selections have been placed.
2. When you are finished, choose *Done* from the pop-up menu.

Related Topics

- [place manual](#)
- [Place Manual Command: Options Panel](#)
- [Placing Alternate Symbols](#)
- [Placing Modules](#)
- [Rotating During Placement](#)
- [Mirroring During Placement](#)

Placing Symbols

When you place a symbol, you can use one of the following:

- The symbol, which is initially attached to the cursor. This procedure describes how to do this.
- The alternate symbol, described in [Placing Alternate Symbols](#).

To place a symbol:

1. Run `place manual`.
 - If you run this command from the menu path, the Placement dialog box appears. Continue with step 2.
 - If you run this command from the console command line, note the syntax. The design element is attached to the cursor by its symbol origin. Continue with step 4.
2. If the symbol you want to place is not already defined in the design, display definitions and the library path from the associated symbol library. Set the *Type filters* field to *Any* and the *Advanced Settings* controls to display all components.
3. Choose the symbol type or individual symbol name(s) you want to place.
The symbol is attached to the cursor by its symbol origin.
The symbol reflects the rotated and mirrored position specified on the *Options* panel or on the *Design* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* (`prmed` command). Rubberbanding ratsnest lines indicate the connections of the component with any components already placed.
4. Change the orientation of the component, if necessary.
Rotate and *Mirror* options are available in the pop-up menu. See [Rotating During Placement](#) and [Mirroring During Placement](#) for details.
5. When you are satisfied with the placement, click to place it.
The symbol is placed.
6. If *AutoNext* is enabled in the *Advanced Settings* tab and you have checked more than one symbol (or the entire group), continue placing symbols until all your selections have been placed, or until you choose *Done* from the pop-up menu.

 You can place symbols with keepins/keepouts on the board even if the keepins/keepouts are on layers that do not exist in the current design.

Related Topics

- [place manual](#)
- [Place Manual Command: Options Panel](#)
- [Placement Dialog Box](#)
- [Placing Modules](#)
- [Rotating During Placement](#)
- [Mirroring During Placement](#)

Placing Alternate Symbols

For details about setting up alternate symbols, see the *Placing the Elements* user guide in your documentation set.

 You can also choose an alternate symbol for one or all symbols in a design with the `use altsym` command, available only in Placement Edit application mode. This command functions in a pre-selection use model, in which you choose a symbol or a group of symbols first, then right-click to display a list of valid alternate symbols.

1. Run `place manual` to display the Placement dialog box and choose a symbol as detailed in steps 1 – 3 of [Placing Symbols](#).

2. Choose *Alt Symbol* from the pop-up menu.

A second pop-up menu appears containing all alternate symbols specified for the symbol.

1. Choose the alternate symbol from the displayed list.

The name of the new symbol displays in the message line.

The *Quickview* window displays different data associated with your selection. If you chose the *Text* button, the name of the alternate symbol appears. If you chose the *Graphics* button, a graphic of the alternate symbol appears.

2. Proceed with the remainder of the steps in [Placing Symbols](#).

Related Topics

- [place manual](#)
- [Place Manual Command: Options Panel](#)
- [Placement Dialog Box](#)
- [Placing Components](#)
- [Rotating During Placement](#)
- [Mirroring During Placement](#)

Placing Modules

When you place modules containing logic, net names within the module are prefixed with the module names. Since net names are limited to 31 characters, module placement fails if adding the module name plus an underscore (_) results in the net name going over the 31 character limit. You MUST insure that net names in a module do not exceed the 31 character limit when it is later prefixed with the module name! Module names are a letter followed by two digits, so your maximum net name within a module must be 27 characters.

Here is the calculation for the effect of module names on your net names and an example:

```
max. module net name = net name max. - module name - underscore
```

```
27 = 31 - 3 - 1
```

1. Run `place manual`.
 - o If you run this command from the menu path, the Placement dialog box appears. Continue with step 2.
 - o If you run this command from the console command line, note the syntax. The design element is attached to the cursor by its symbol origin. Continue with step 4.
2. Configure the *Advanced Settings* tab to display the appropriate lists and to set the general options.
3. Choose the module instance and/or definition name(s) you want to place.
The module is attached to the cursor at the point of origin defined in the module.
4. Change the orientation of the module, if necessary.
Rotate and *Mirror* options are available in the pop-up menu. See [Rotating During Placement](#) and [Mirroring During Placement](#) for details.
5. When you are satisfied with the placement, click to place the module.
A fill-in box appears when the element being placed is a module definition. If it is a module instance, it already has an instance name and the fill-in does not appear.
6. Type in a module instance name to define the new instance of the module definition, and click *OK*.
The module is placed and locked in the design canvas. Dynamic shapes within a module are converted to static shapes and any structure that is part of a module is flattened into its design elements.

⚠ You can unlock the module either using right-click option *Unlock* or by removing the LOCKED property using [Edit – Properties \(property edit\)](#) command.

7. If *AutoNext* is enabled in the *Advanced Settings* tab and you have checked more than one module (or the entire group), continue placing modules until all your chosen objects have been placed, or until you choose *Done* from the pop-up menu.

Related Topics

- [place manual](#)
- [Place Manual Command: Options Panel](#)
- [Placement Dialog Box](#)
- [Placing Components](#)
- [Placing Symbols](#)
- [Mirroring During Placement](#)

Rotating During Placement

The *Rotate* option in the placement pop-up lets you rotate the symbol attached to the cursor, so the symbol shows the correct rotation before you position it in the design. You can rotate the component or symbol by any angle increment, in either a clockwise or a counterclockwise direction.

To rotate the component or symbol attached to the cursor during a placement operation:

1. Choose *Rotate* from the pop-up menu.
A line appears connecting the cursor to the symbol to let you control rotation.
2. If required, change the angle increment at which rotation occurs in the *Angle* field on the *Options* panel of the control panel or in the *Design Parameter Editor*.
3. Move the cursor in a clockwise or counterclockwise direction to rotate the design element by the increment specified in the *Angle* field.
4. When the design element is at the required angle, click left.
When you move the cursor, the rotated symbol reappears attached to the cursor. You can now continue placing the element.

Related Topics

- [place manual](#)
- [Place Manual Command: Options Panel](#)
- [Placement Dialog Box](#)
- [Placing Components](#)
- [Placing Symbols](#)
- [Placing Alternate Symbols](#)

Mirroring During Placement

1. As you place a design element, you can mirror it to the other side of the design. If *Mirror* is chosen on the *Options* panel or in the *Design* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* (`prmed` command).

Choose the *Display* tab placement automatically uses a component in its mirrored state.

For information about mirroring with alternate symbols, see the *Placing the Elements* user guide in your documentation set.

To mirror the component or symbol during a placement operation:

1. Choose *Mirror* from the pop-up menu. OR choose the *Mirror* option on the *Options* panel.

The item you are placing is mirrored. When you move the cursor, the mirrored component or symbol reappears attached to the cursor. Continue placement. If you have used the *Color* option to request the display of items on the opposite side of the board/substrate, the item appears in color for that side when you have completed placement.

For information on using the `mirror` command, see *Preparing for Layout* in the user guide.

Reversing Mirroring

To change the symbol back to the original side of the design:

1. Choose *Mirror* again from the pop-up menu while the component or symbol is still attached to the cursor.

Examples

These are examples of using the `place manual` command with its optional arguments.

This example attaches to the cursor the symbol assigned to u1:

```
place manual refdes u1
```

This example attaches to the cursor the package symbol dip14:

```
place manual package dip14
```

Related Topics

- [place manual](#)
- [Place Manual Command: Options Panel](#)
- [Placement Dialog Box](#)
- [Placing Components](#)
- [Placing Symbols](#)
- [Placing Alternate Symbols](#)
- [Placing Modules](#)

placement

The `placement` command is a batch program that executes the automatic placement process in a Unix window. Before executing placement, you must assign values, where necessary, to the automatic placement parameters. The program completes the number of passes you assigned to the Max Pass parameter, to a maximum of 10. The best results obtained from any of the passes are saved, in the following manner.

When the first execution is completed, the results are placed in the `output.brd` file. At the end of the next execution, it compares the new results to those in the `output.brd` file. If they are better, `output.brd` is over-written; if worse, the new results are discarded. This process is repeated for each designated pass, assuring that the final results in `output.brd` are the best achieved by all the passes. If you do not designate `output.brd`, the results are placed in `input.brd`, overwriting the original design.

Results based on the greatest number of critical connections completed and the shortest manhattan length are considered. Critical connections are defined in a `crit.dat` file, described in see the *Placing the Elements* user guide in your documentation set.

Syntax

- Without optional arguments

`placement input.brd [output.brd]`

- With optional arguments

-d	Sets debug on.
-a	Improves while (percent < ACCEPT).
-w	Weights the complete graph edges.
-p	Prints the connection matrix.
-version	Prints the version.
drawing_name	Name of drawing without ext.
save_name	Name of modified drawing with ext.

Executing Automatic Placement in a Unix Window

Follow these steps to perform automatic placement in a unix window:

1. Type `placement` at your operating system prompt.

If you enter the command name without arguments, you are prompted to enter input and output file names.

If you want to refine the parameters of the command, enter the appropriate arguments on the command line.

2. Press the Return/Enter key to run the program.

placementedit

The placement edit application mode provides an environment lets you perform tasks relevant during placement. An application mode provides an intuitive environment in which commands used frequently in a particular task domain, such as placement, are readily accessible from right-mouse-button popup menus, based on a selection set of design elements you have chosen. These tasks include:

- accessing a dockable placement window tab
- fine-tuning component placement and alignment using the [align components](#) command, available on a pop-up menu
- replicating circuits based on common connectivity and devices using a schematic-independent template model using the [place replicate create](#) command, available on a pop-up menu
- using a component's alternate symbols with the [use altsym](#) command, available on a pop-up menu

This customized environment maximizes productivity when you use multiple commands on the same design elements or those in close proximity in the design. Application mode configures your tool for a specific task by populating the right-mouse-button popup menu only with commands that operate on the current selection set.

In conjunction with an active application mode, your tool defaults to a pre-selection use model, which lets you choose a design element (noun), and then a command (verb) from the right-mouse-button popup menu. This pre-selection use model lets you easily access commands based on the design elements you've chosen in the design canvas, which the tool highlights and uses as a selection set, thereby eliminating extraneous mouse clicks and allowing you to remain focused on the design canvas.

Use *Setup – Application Mode – None* ([noappmode](#) command) to exit from the current application mode and return to a menu-driven editing mode, or verb-noun use model, in which you choose a command, then the design element.

For more information on the placement-edit application mode, see *Using Placement Application Mode* in the *Placing the Elements* user guide or the *Getting Started with Physical Design* user guide in your documentation set.

Access Using

- Menu Path: *Setup – Application Mode – Placement Edit*



- Toolbar Icon:

Accessing Command Help for Right Mouse Button Options within an Application Mode

To access help command help for right mouse button options within an application mode, follow these steps:

1. Type `helpcmd` in the Allegro X PCB Editor console window.
The Command Browser dialog box appears.
2. Enable the *Help* radio button at the top of the dialog box to place the browser in Help mode.
3. Scroll the command list and select (double-click) the command you want help on.
The command documentation displays in the Cadence Help documentation browser momentarily.

place param

The `place param` command displays the Automatic Placement dialog box where you can do the following:

- Set the parameters that control automatic placement
- Run automatic placement in automatic mode

Related Topics

- [Setting Parameters for and Running Automatic Placement in Automatic Mode](#)
- [Examples of the Place Param Command](#)

Automatic Placement Dialog Box

Access Using

- *Menu Path: Place – Autoplace – Parameters*

The Automatic Placement dialog box sets the parameters that control automatic placement. It can also run automatic placement in automatic mode.

Initial settings in the dialog box reflect system defaults. Each parameter in the dialog box guides placement within the active placement area: a room, a window, or the entire design.

Because automatic placement completes quickly, one way to become more familiar with the placement parameters is to run automatic placement using the default settings on the Automatic Placement dialog box. After you study the results, adjust one placement parameter at a time interactively, and rerun automatic placement for the same placement area.

<i>Algorithm</i>	Specifies one of the following placement algorithms:	
	<i>Discrete</i>	Treats components, such as capacitors and resistors, as if they are discrete. Typically, you use this algorithm to place small-body components.
	<i>IC</i>	(Default) Treats all components as if they were DIPs. Typically, you use this algorithm to place large-body components, even large discrete.
	<i>Array</i>	Places components in a regular pattern, such as that for bypass capacitors, terminating SIP packs, and spares. Automatic placement runs more quickly with this algorithm than with the other two. It does not care about connectivity or parameter weights, except for rotation weights.
<i>Origin</i>	Specifies the point on each component to use for the anchor point. When you choose a component for placement, the program looks at the possible grid choices, selects the most appropriate, and aligns the specified origin with the intersection. Choose from the following component origins:	
	<i>Body Center</i>	Defines the component center by adding a text point at that location. The text point must be of class PACKAGE GEOMETRY/PART GEOMETRY, subclass BODY_CENTER. If you do not define the component center on the symbol drawing, the system automatically uses PLACE_BOUND_TOP to calculate the body center and uses that point for placement.
	<i>Symbol Origin</i>	Indicates that the origin of the component is (0,0) on the symbol drawing.
	<i>Pin 1</i>	Indicates the pin that is defined as Pin 1 in the symbol drawing is positioned on a placement grid point. If no pin is defined as Pin 1, the component is placed on the symbol origin.
<i>Weights</i>	All the parameters in this section use weights between 0 and 100 to indicate their relative importance in the placement process. The weights have the following relative impact on the tool:	
	<i>0</i>	Do not consider the parameter.
	<i>50</i>	Consider the parameter, but do not favor it.
	<i>100</i>	Give a heavy preference to the parameter.
	For more details, see the <i>Placing the Elements</i> user guide in your documentation set.	
<i>Direction</i>	Tells the system where you want a component to be placed in relation to its most highly connected partner. Weighting indicates the direction that you prefer. The tool interprets the directions the same way that you look at the design:	
	<i>North</i>	Above its highest connected partner
	<i>South</i>	Below its highest connected partner
	<i>East</i>	Right of its highest connected partner
	<i>West</i>	Left of its highest connected partner
	The system default is 50 for each direction, which means that each direction is an equally likely choice. For a comparison of direction weights, see Setting Direction Weights.	

P Commands

P Commands--place param

<i>Rotation</i>	Defines how you want your components rotated relative to their library orientation (0, 90, 180, and 270). The system default is 50 for the <i>0 Rotation</i> parameter and 0 for each of the others. These defaults assume that you built your symbol with the preferred orientation. To see how weights are applied to rotation, see <i>Setting Rotation Weights</i> .	
<i>Straight</i>	Specifies that you want interconnected components placed so that the connections between them are straight rather than diagonal. These are values that can guide you to the setting you want:	
	0	Indicates that you prefer diagonal connections over straight
	50	Indicates that straight and diagonal are equally acceptable
	100	Indicates that you prefer straight connections
	For an example of how to use this parameter, see <i>Specifying Weight for Straight or Diagonal Connections</i> .	
<i>Mirror</i>	Specifies on which side of the design you prefer a component to be placed in relation to the component with which it is most interconnected. This parameter applies only when you have chosen placement areas and defined grids on both sides of the design. These are values that can guide you to the setting you want:	
	0	Indicates the components are to be placed on the side of the layout as the components with which they are most interconnected.
	50	Indicates that it makes no difference whether the components are placed on the same side of the layout or on the side opposite the components with which they are most interconnected.
	100	Indicates that the components are to be placed on the side of the layout opposite the components with which they are most interconnected.
	For an example of using this parameters, see <i>Specifying Mirror Weight</i> .	
<i>Leftovers</i>	Tells placement what to do with components that it cannot place—for example, when there is not enough space in the placement area. When the option is on (the default), it places all leftovers in a row starting in the lower left corner of the design. When off, it leaves them as unplaced components.	
<i>Overlap</i>	<p>Specifies whether components can overlap. There are two reasons for allowing components to overlap:</p> <ul style="list-style-type: none"> When there is not enough room within the package/part keepin for all the components to be added. Automatic placement must overlap the components or leave the excess components unplaced. When there is enough room within the package/part keepin but you want to see the components' preferred locations. If you set this parameter when placing every component, however, you will probably end up with all of the components on top of one another. <p>When this option is on, automatic placement overlaps components. When off, automatic placement handles excess components according to the <i>Leftovers</i> setting.</p>	
<i>Soft boundary</i>	Specifies whether components can be placed outside a room created for placement. When this option is on, they can be placed outside the room if it runs out of space. Automatic placement never places components outside the package/part keepin and always tries to fit them within the room, even with soft boundaries. When off, any components that cannot be placed are handled according to the <i>Leftovers</i> setting.	
<i>Clock redistribution</i>	Reserved.	
<i>Cluster</i>	<p>Specifies whether automatic placement should try to cluster all heavily connected components in the placement area. When this option is on, automatic placement considers all placed components influencing the selection of unplaced components. When off, autoplace considers connectivity to components that already are placed inside the placement area components to be outside the area only if the PLACE_TAG property has been specified for them. If you have not enabled <i>Cluster</i> during automatic placement, components with PLACE_TAG properties are examined as follows:</p> <ul style="list-style-type: none"> Unplaced components – Selected for placement Placed components – If outside the current placement area, consider connectivity to this component when choosing the next component to be placed. 	
<i>No rat</i>	Specifies how the system should handle ratsnesting during automatic placement. Running without ratsnesting enables the automatic program to execute faster. When this option is on, ratsnesting is turned off during automatic placement, then repainted when the execution is complete. When off, ratsnesting continues to occur dynamically during automatic placement. If you run automatic placement in interactive mode (place interactive command), ratsnesting displays regardless of the selection made for this parameter.	

P Commands

P Commands--place param

Remove TAG	Specifies what the system should do at the end of automatic placement with the PLACE_TAG properties that are attached to placed components. When this option is on, the PLACE_TAG property is removed from each component that was placed during the current execution of automatic placement. When off, the PLACE_TAG property is left attached to each component. The critical factor in deciding which setting to use is whether or not <i>Cluster</i> is turned on: <ul style="list-style-type: none">• If <i>Cluster</i> is on, leaving <i>Remove TAG</i> off affects the placement of your next area. (See the <i>Cluster</i> description earlier in this table.)• When <i>Cluster</i> is off, all placed components are looked at, both inside and outside the designated placement area, when choosing and positioning the next component.
Max pass	Specifies how many times automatic placement repeats its evaluation to improve placement. The default is 1; the maximum is 10. After the command runs each improvement evaluation, it checks whether the changes that were evaluated would actually improve the overall placement and, if so, takes that as the actual placement. Therefore, you are guaranteed the best of all passes being executed. When you request more than one pass, the tool creates a new file called <code>BSTPLC.brd</code> to store the results of the best execution. For details, see the <i>Placing the Elements</i> user guide in your documentation set. <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"><p>⚠ Each pass takes approximately the same length of time, so the total time increases directly with the number of passes specified.</p></div>
Place	Saves the settings and runs automatic placement in automatic mode.
Close	Saves the settings and closes the dialog box.

Related Topics

- [Examples of the Place Param Command](#)

Setting Parameters for and Running Automatic Placement in Automatic Mode

To run automatic placement in automatic mode and set placement parameters, follow these steps:

1. Run `place param` to display the Automatic Placement dialog box.
2. Edit the dialog box as required.
3. Click *Place* to apply the parameters and place the components in automatic mode. –or– Click *Close* to apply the parameters and close the dialog box.

As the automatic placement process occurs within a placement area, the weighted parameters help automatic placement make decisions based on connection to the seed ecomponent.

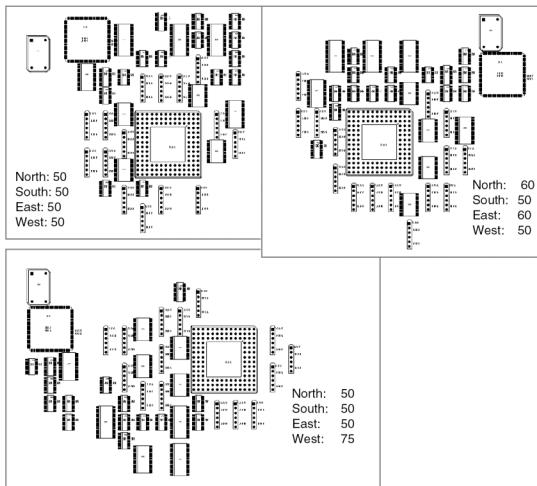
Related Topics

- [place param](#)

Examples of the Place Param Command

Setting Direction Weights

This figure shows the differences in the same placement area with different direction weights applied.

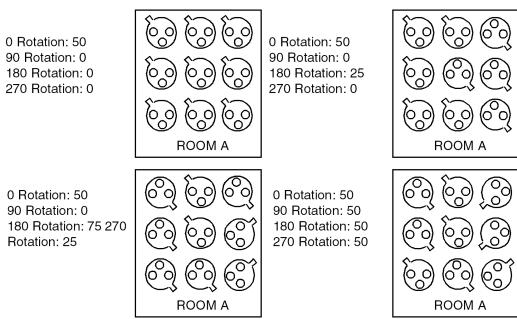


When the other directions are set to 50, the following rules apply to the *North* parameter:

- Setting *North* to 0 indicates that components should not be placed north of their highest connected partner.
- Setting *North* to a value less than 50 means that, relative to the other directions, it is usually better to place components to the south, east, or west than to the north of their highest connected partner.
- A value of 50 means that it makes no difference whether the components are placed to the north, south, east, or west of their highest connected partner.
- Setting *North* to a value greater than 50 means that, relative to the other directions, it is better to place components to the north of their highest connected partner than to the south, east, or west.

Setting Rotation Weights

This figure shows how weights are applied to rotation.

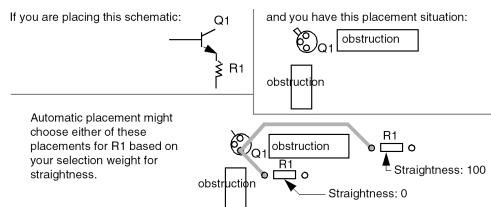


When you set the other Rotation parameters to 50, the following rules apply to the *0 Rotation* parameter:

- Setting rotation to 0 means does not rotate the component 0 degrees.
 - Setting rotation to a value of less than 50 means that, relative to other rotations, it is better to rotate the components to some rotation other than 0 degrees.
 - Setting rotation to 50 means that, relative to the other rotations, components are as likely to be rotated 0 degrees as any other rotation.
 - Setting rotation to a value greater than 50 means that, relative to the other rotations, it is better to rotate the components 0 degrees.
- The WEIGHT and COMPONENT_WEIGHT properties also affect swapping.

Specifying Weight for Straight or Diagonal Connections

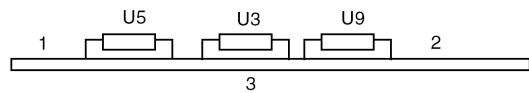
The following example illustrates the effect of the straightness weight.



Specifying Mirror Weight

This figure illustrates how the editor is likely to react when various weights are applied to the mirror parameter.

If U10 is strongly connected to U3, you have the following placement situation:



You can expect automatic placement to place U10 in the following slots based on your decision on mirroring:

Mirror: 0 -- Slot 1 or 2
Mirror: 50 -- Slot 1, 2, or 3
Mirror: 100 -- Slot 3

Related Topics

- [place param](#)
- [Automatic Placement Dialog Box](#)

place replicate apply

The `place replicate apply` command overlays the "seed" data contained in a place replicate module definition database (`.mdd`) file onto elements in the current selection set and then replicates them.

When a circuit is replicated using this command, the replicated symbols and etch, such as clines, shapes, vias, text, and structures are grouped into a module instance that can then be manipulated as a single database object. The module instance is named `CR_<instance_name>_<1-n>`, where CR refers to circuit replicate, and instance name is derived from the `.mdd` file name specified during creation of the place replicate module. (The `undo` command is supported; the `redo` command is not.)

 Place replicate modules are those created with the suite of place replicate commands and are differentiated from traditional modules, which are driven by the `REUSE_MODULE` property definition.

Module created by this command are locked by default. Setting the `disable_module_auto_lock` variable disables automatic locking of the module. You can set this variable in the *Placement – General* category of the User Preferences Editor. A right-click option `Unlock` (`unlock module`) is also available to remove the lock from any module definition. You can also unlock a module by deleting the `LOCKED` property using `Edit – Properties` (`property edit`) command.

If the place replicate module contains dynamic shape it gets converted to static shape when applied. This is done by default to prevent shape-related conflicts between module and target design. Set `disable_module_shape_convert` environment variable to disable shape conversion,

Place replicate module with structures when applied flattens the structure into its design elements. The command ignores the structure symbol definition and matches its content.

If placement or routing changes are made to a single place replicate module instance, the `place replicate update` command can be used to propagate those modifications to all other instances across the design with the same base name.

`place replicate apply` neither adds any symbols or components to the design nor changes any net/pin assignments. To replicate, the command matches module based on symbol and component data as well as connectivity. A match occurs if symbol name, device name, and values match and all pins connected to each other in the module are in the same net on the board. The command considers alternate symbols as match:

- If a symbol in the target replicated circuit is an alternate definition of a symbol in the seed circuit, or
- If a symbol in the target replicated circuit has an alternate symbol defined in the seed circuit

 When the command matches a symbol and an alternate symbol, the footprint of the symbol in the target replicated circuit changes to match with the symbol in the seed circuit, ensuring that the replicated circuit matches the seed circuit.

If at least one complete match is not found, [Place Replicate Unmatched Component Interface Dialog Box](#) is displayed.

Available only in the placement edit application mode, this command functions in a pre-selection use model, in which you choose an element first, then right-click and execute the command. Valid element is Symbols.

Related Topics

- [Place Replicate Component Swap Interface Dialog Box](#)
- [Place Replicate Layer Mapping Dialog Box](#)
- [Running the Circuit Replication Flow Command](#)
- [Resolving Unmatched Components between the Seed and Targeted Replicated Circuits](#)

Place Replicate Unmatched Component Interface Dialog Box

This dialog box only displays when components in the seed circuit cannot be matched in the targeted replicated circuit, and assists in troubleshooting these situations.

<i>Seed Circuit -> Match</i>	Lists components in the seed circuit for which a match exists in the target replicated circuit, as well as those for which a match cannot be made and are unmatched.
<i>Similar Components: RefDes (device; value)</i>	By default, shows the unmapped reference designators of symbols whose symbol name, device name, and value match those of the reference designator chosen in the <i>Seed Circuit -> Match</i> list. Click an unmatched component in the <i>Seed Circuit -> Match</i> list to populate this section with appropriate matches for the chosen component, depending on whether <i>Device Name</i> or <i>Value</i> are enabled.
<i>Match</i>	Specifies the criteria used to locate matches for components in the <i>Seed Circuit -> Match</i> list. If neither <i>Device Name</i> or <i>Value</i> is enabled, both can be selected at the same time.
<i>Unmatch</i>	Click to customize the mapping of components between the seed circuit and the replicated circuit. For instance, if C164 -> C167 appears in the <i>Seed Circuit -> Match</i> list, click on it, then click <i>Unmatch</i> . The component C167 appears in the <i>Similar Components</i> section, and you can change the component mapping for C164.
<i>Unmatch all</i>	Click to customize the mapping of all the components between the seed circuit and the replicated circuit.
<i>Component details Refdes (Symbol, Device, Value)</i>	Displays the details of the component selected from <i>Seed Circuit</i> list.
<i>OK</i>	Saves any changes made during the session and closes the dialog box.
<i>Cancel</i>	Closes the dialog box without saving the settings.

Related Topics

- [Place Replicate Layer Mapping Dialog Box](#)
- [Running the Circuit Replication Flow Command](#)
- [Resolving Unmatched Components between the Seed and Targeted Replicated Circuits](#)

Place Replicate Component Swap Interface Dialog Box

This dialog box appears only when substitutes exist for components in the selection set, such as capacitors for instance. Components are only considered for swapping if the integrity of the circuit is maintained.

Next Circuit:	Displays a subset of circuits in the current selection set that match the seed circuit. When you click OK, these components attach to your cursor and are replicated once you click to place them. When you click OK, the component group currently listed here attaches to the cursor.
Swappable:	Lists components in the selection set for which substitutes exist. Symbol and component definitions must be identical and pins must connect to the same nets. For instance, the schematic may specify C1 and C2 to be placed sequentially, but the tool may choose to replicate C3 rather than C2 as the next circuit that matches. In this case, use this field to swap C2 for C3 in the component group targeted for replication.
Swap with:	Lists components that are eligible substitutes for the selected component in the <i>Swappable</i> field, meaning their symbol and component definitions and net connectivity are identical. Clicking a reference designator here swaps it with the selected component.
Ok	Click to attach the subset of circuits in the current selection set that match the seed circuit to your cursor and replicate them. Every time you click <i>OK</i> , the component group currently shown in <i>Next Circuit</i> appears on your cursor, one instance at a time. The tool continues to locate circuits in the selection set that match the seed circuit, and present the dialog box to allow you to swap one component for another, unless you choose <i>Hide Form</i> .
Hide Form	Click to prevent the <i>Place Replicate Component Swap Interface</i> dialog box from appearing. Doing so, however, prevents you from swapping one component for another and allows the tool to choose for you.

Related Topics

- [place replicate apply](#)
- [Running the Circuit Replication Flow Command](#)
- [Resolving Unmatched Components between the Seed and Targeted Replicated Circuits](#)

Place Replicate Layer Mapping Dialog Box

This dialog box is shown if there are differences in the source and destination stackup.

<i>Source Stackup</i>	Lists the layers in the source.
<i>Mapped StackupForm</i>	Lists the layers in the source that are mapped to the destination.
Destination Stackup	Lists the layers in the destination.

Drag layers to change mappings.

Related Topics

- [place replicate apply](#)
- [Place Replicate Unmatched Component Interface Dialog Box](#)
- [Resolving Unmatched Components between the Seed and Targeted Replicated Circuits](#)

Running the Circuit Replication Flow Command

Follow these steps to run the circuit replication flow command:

1. Choose *Setup – Application Mode – Placement Edit*.
2. Place and route the initial (seed) circuit.
3. Choose all components that comprise the seed circuit by left-clicking, shift-clicking, or window selecting.
The console window prompts you to select/de-select the additional etch elements for the seed circuit. After you have completed the selection, right-click and choose *Done*.
4. Right-click and choose *Place Replicate Create* from the pop-up menu that appears.
The console window prompts you to select/de-select the additional etch elements for the seed circuit. After you have completed the selection, right-click and choose *Done*.
5. The console command window prompts to Pick origin or use right-click to *Use Snap to* functionality:
 - Click to pick the origin, or
 - Right-click to *Use Snap to*, which lets you accurately position pick points with snapping modes.

The *Placement Replicate Create* dialog box appears.

6. Enter a name for the place replicate module definition database file (*.mdd*).
7. Click *Save the File* to save the *.mdd* file to disk.
8. Click *OK* to save the *.mdd* file to the current design database and close the dialog box.
9. Window select or choose the components targeted for replication. Limiting the selection to relevant members reduces processing time.
10. To apply the seed circuit *.mdd* file to other common circuits, right-click and choose *Place Replicate Apply* from the pop-up menu.
11. Choose the seed circuit *.mdd* file from the pop-up menu, or choose *Browse* to locate an existing file on disk. One of the following occurs:
 - If components match exactly, click *OK* to attach the resultant circuits to your cursor and place them. Right click to rotate or mirror components as necessary.
 - If a component can be substituted with another component, typically decoupling capacitors, the *Place Replicate Component Swap Interface* dialog box appears, and the *Next Circuit section* lists a subset of circuits by reference designator in the current selection set that match the *.mdd* criteria.
 - Click the component in the *Swappable* list, and eligible substitutes for it appear in the *Swap with* list.
 - Click the component in the *Swap with* list to make the substitution; the substituted component appears in *Next Circuit*.
 - Click *OK*. The component group shown in the *Next Circuit* attaches to the cursor, then left click to place the replicated circuit. Every time you click *OK*, the component group currently shown in the *Next Circuit* attaches to the cursor, and the tool then continues to locate circuits in the selection set that match the seed circuit. The *Place Replicate Component Swap Interface* dialog box redisplays each time you click *OK* if substitutions for a component exist, unless you choose *Hide Form*. Doing so, however, prevents you from substituting one component for another and allows the tool to choose for you.
 - Right-click and choose *Done* from the pop-up menu that appears to finish placing replicated circuits.
 - If a component in the seed circuit cannot be matched in the targeted replicated circuit, the *Place Replicate Unmatched Component Interface* dialog box displays, which allows you to place an altered version of the seed circuit. See [Resolving Unmatched Components Between the Seed and Targeted Replicated Circuits](#) for additional procedural information.
12. Use the [place replicate update](#) command as required after modification of one instance of a replicated circuit to apply those modifications to all other instances with the same base name across the design.

Related Topics

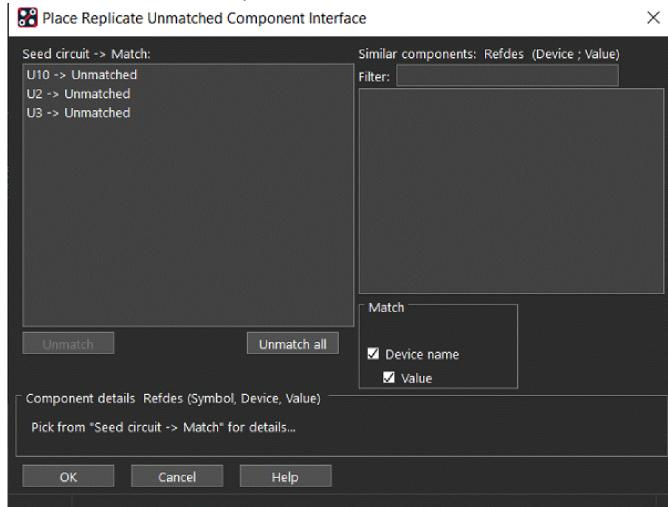
- [place replicate apply](#)
- [Place Replicate Unmatched Component Interface Dialog Box](#)
- [Place Replicate Component Swap Interface Dialog Box](#)

Resolving Unmatched Components between the Seed and Targeted Replicated Circuits

The *Place Replicate Unmatched Component Interface* dialog box only displays when a component in the seed circuit cannot be matched in the targeted replicated circuit, and assists in troubleshooting these situations.

Follow these steps to resolve unmatched components:

1. Choose *Setup – Application Mode – Placement Edit*.
2. After placing the initial (seed) circuit and creating the *.mdd* file from it, window select the components targeted for replication. Limiting the selection to relevant members reduces processing time.
3. To apply the *.mdd* file, right-click and choose *Place Replicate Apply* from the pop-up menu and choose the *.mdd* file or *Browse* to locate an existing file on disk.
- The *Place Replicate Unmatched Component Interface* dialog box displays when components in the seed circuit cannot be matched in the targeted replicated circuit.
4. Review the components that appear in the *Seed Circuit -> Match* list.
5. Click an unmatched component in the *Seed Circuit -> Match* list to populate the *Similar Components: RefDes (device; value)* list with appropriate matches for the chosen unmatched component.

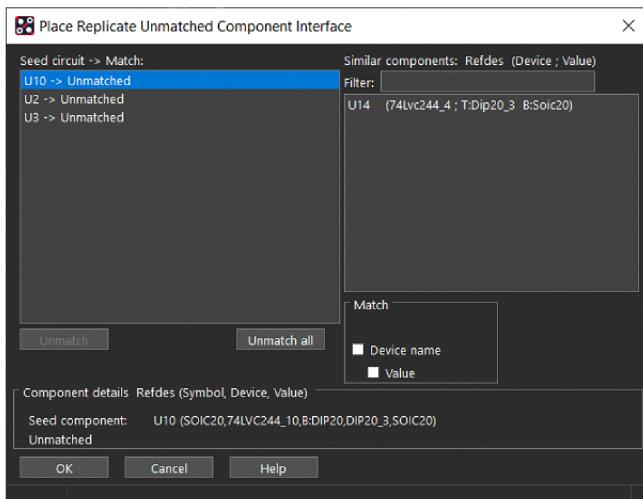


Choose either of the following, enable both, or disable both:

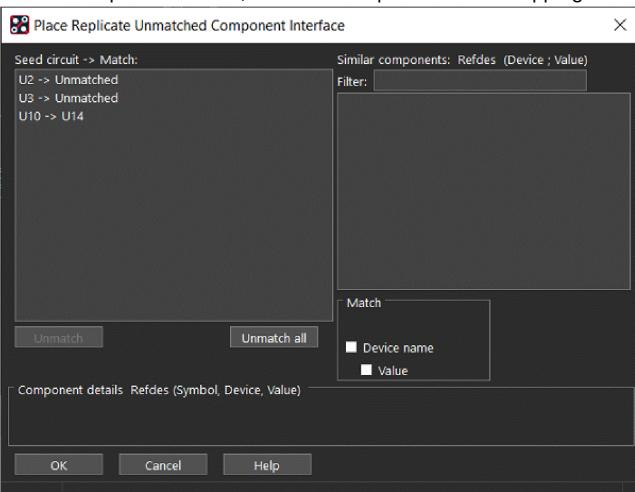
- a. *Device Name* to match components by device name in the *Similar Components* section. Otherwise the device name is not used for matching.
 - b. *Value*: to match components by value in the *Similar Components* section. Otherwise the value is not used for matching.
 - c. If neither *Device Name* or *Value* is enabled, only the symbol name is matched.
6. From the *Similar Components: RefDes (device; value)* list, click on a match for the unmatched component you chose in the *Seed Circuit -> Match* list. For instance, U14 appears as a candidate match for U10.

P Commands

P Commands--place replicate apply



7. The chosen component then displays in the *Seed Circuit -> Match* list as U10 ->U14. To customize an existing mapping of components between the seed circuit and the replicated circuit, choose a component with a mapping from the *Seed Circuit -> Match* list, click on it, then choose *Unmatch*.



Related Topics

- [place replicate apply](#)
- [Place Replicate Unmatched Component Interface Dialog Box](#)
- [Place Replicate Component Swap Interface Dialog Box](#)
- [Place Replicate Layer Mapping Dialog Box](#)

place replicate create

The `place replicate create` command designates a circuit as a "seed," and creates a place replicate module definition database (`.mdd`) file from it, essentially a template containing pre-placed symbols you have chosen and their associated logic, routed etch, shapes, vias, and structures to be used to replicate additional circuits.

You then use the `place replicate apply` command to overlay the "seed" data onto the components currently in your selection set. Circuits are replicated when their symbol definitions, component definitions, and net connectivity match the `.mdd`.

 Place replicate modules are those created with the suite of place replicate commands and are differentiated from traditional modules, which are driven by the REUSE_MODULE property definition.

Available only in the placement edit application mode, this command 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

Related Topics

- [Creating a Place Replicate Module File \(.mdd\) from a Seed Circuit](#)

Placement Replicate Create Dialog Box

<i>Enter new place replicate name</i>	Specifies a name for a place replicate module definition database (.mdd) file.
<i>Or pick name to overwrite</i>	Displays a list of existing .mdd files and overwrites its contents.
<i>Save to File</i>	Saves the .mdd file to disk for use on other boards.
<i>Ok</i>	Saves the .mdd file to the current design database and closes the dialog box.
<i>Cancel</i>	Closes the dialog box without saving the settings.

Creating a Place Replicate Module File (.mdd) from a Seed Circuit

Perform these steps to generate a place replicate module file from a seed circuit:

1. Choose *Setup – Application Mode – Placement Edit*.
2. Create and place the initial (seed) circuit.
3. Preselect all components associated with the seed circuit by left-clicking, shift-clicking, or window selecting.
4. Right-click and choose *Place Replicate Create* from the pop-up menu that appears.
The console window prompts you to select/de-select the additional etch elements for the seed circuit. After you have completed the selection, right-click and choose *Done*.
5. The console command window prompts to Pick origin or use right-click to *Use Snap to* functionality:
 - Click to pick the origin, or
 - Right-click to *Use Snap to*, which lets you accurately position pick points with snapping modes.
6. The *Placement Replicate Create* dialog box appears. Enter a name for the place replicate module definition database file (.mdd).
7. Click *Save the File* to save the .mdd file to disk for use on other boards.
8. Click *OK* to save the .mdd file to the current design database and close the dialog box.
9. Window select the components targeted for replication. Limiting the selection to relevant members reduces processing time.
10. To apply the .mdd file, right-click and choose *Place Replicate Apply* from the pop-up menu. See the [place replicate apply](#) command for additional procedures.

Related Topics

- [place replicate create](#)

place replicate update

The `place replicate update` command lets you modify one instance of a replicated circuit created with the [place replicate apply](#) command and propagate the etch element and placement location modifications to all other instances with the same base name across the design.

Available only in the placement edit application mode, this command functions in a pre-selection use model, in which you choose an element first, then right-click and execute the command. Valid elements are:

- Module Instances

 The command also updates modules that are locked.

Modifying Replicated Circuits

To modify replicated circuits, follow these steps:

1. Choose *Setup – Application Mode – Placement Edit*.
2. Select Groups in the *Find* filter.
3. Hover your cursor over the modified replicated circuit to be used as a template.
4. Right-click and choose *Place Replicate Update* from the pop-up menu that appears. Any modifications made to the template circuit propagate to all other instances with the same base name across the design.

place set bottomgrid

The `place set bottomgrid` command generates grids on a design's BOARD GEOMETRY/SUBSTRATE GEOMETRY class, PLACE_GRID_BOTTOM subclass. You can mirror this grid up to the PLACE_GRID_TOP subclass.

To use this command, you must have a package/part keepin in your design.

Access Using

- *Menu Path: Place – Autoplace – Bottom Grids*

When you run `place set bottomgrid`, two dialog boxes display, prompting you for the X and Y increments of the grid.

Generating Grids on a BOARD GEOMETRY/SUBSTRATE GEOMETRY Class, PLACE_GRID_BOTTOM Subclass

If you have not created a package/part keepin area for the automatic placement area, create the package/part keepin from the *Setup – Areas – Package Keepin* menu.

Follow these steps to generate grids on a BOARD GEOMETRY/SUBSTRATE GEOMETRY class, PLACE_GRID_BOTTOM subclass:

1. Run `place set bottomgrid`.
All grid lines are erased from the subclass BOTTOM and a dialog box prompts you for the X increment of the grid.
 2. Enter a value for the spacing between the grid lines in the X direction.
Another dialog box prompts you for the Y increment of the grid.
 3. Enter a value for the spacing between the grid lines in the Y direction.
- ⚠ If you enter 0 for the X and Y grid spacing, the current grid is deleted.
4. Pick a location for a grid point within the package/part keepin.
The program places the intersection of one pair of horizontal and vertical grid lines at the point you select, then adds the remaining horizontal and vertical grid lines, spaced at the specified increments, to fill the package/part keepin.
The grid lines are highlighted.
 5. If the lines are not positioned as you want, pick another location.
The grid lines are erased and added again, starting at the new origin.
 6. Continue to pick locations until they are positioned appropriately.
- ⚠ Grid lines must run vertically or horizontally across the package/part keepin. If they have odd angles or jogs, automatic placement does not run and errors are reported.
7. When you are satisfied with the grid, choose *Done* from the pop-up menu.

place set topgrid

The `place set topgrid` command generates grids on a design's BOARD GEOMETRY/SUBSTRATE GEOMETRY class, PLACE_GRID_TOP subclass. You can mirror this grid down to the PLACE_GRID_BOTTOM subclass.

To use this command, you must have a package/part keepin in your design.

Access Using

- *Menu Path: Place – Autoplace – Top Grids*

When you run `place set topgrid`, two dialog boxes display, prompting you for the X and Y increments of the grid.

Generating Grids on a BOARD GEOMETRY/SUBSTRATE GEOMETRY Class, PLACE_GRID_TOP Subclass

If you have not created a package/part keepin area for the automatic placement area, create the package/part keepin from the *Setup – Areas – Package Keepin* menu.

Follow these steps to generate grids on a BOARD GEOMETRY/SUBSTRATE GEOMETRY class, PLACE_GRID_TOP subclass:

1. Run `place set topgrid`.
All grid lines are erased from the subclass TOP and a dialog box prompts you for the X increment of the grid.
 2. Enter a value for the spacing between the grid lines in the X direction.
Another dialog box prompts you for the Y increment of the grid.
 3. Enter a value for the spacing between the grid lines in the Y direction.
- ⚠ If you enter 0 for the X and Y grid spacing, the current grid is deleted.
4. Pick a location for a grid point within the package/part keepin.
The program places the intersection of one pair of horizontal and vertical grid lines at the point you select, then adds the remaining horizontal and vertical grid lines, spaced at the specified increments, to fill the package/part keepin.
The grid lines are highlighted.
 5. If the lines are not positioned as you want, pick another location.
The grid lines are erased and added again, starting at the new origin.
 6. Continue to pick locations until they are positioned appropriately.
- ⚠ Grid lines must run vertically or horizontally across the package/part keepin. If they have odd angles or jogs, automatic placement does not run and errors are reported.

7. When you are satisfied with the grid, choose *Done* from the pop-up menu.

place structure

The `place structure` command lets you place structures in a design. You can snap structures to pins, vias, clines, or free place anywhere.

You can also refresh and replace structures. See the [refresh via structure](#) and [replace via with structure](#) commands for details.

For additional information on correct design methodologies associated with creating and using structures, see *Defining and Developing Libraries* in the user guide.

Related Topics

- [Placing a Structure](#)

Place Structure Dialog Box

Access Using

- Menu Path: Route – Structures – Place

This dialog box appears when you run the `place structure` command. It contains the controls for placing structures to your design.

<i>Select Structure</i>	<ul style="list-style-type: none"> • <i>Pick from design</i> lets you choose a defined structure from your current design. The selected structure is then highlighted in the dialog box. • <i>Pick from list</i> lets you choose from the list of existing structures in the list box. The structures listed in the box may come from one or both of the following sources: <ul style="list-style-type: none"> ◦ <i>database</i> displays only the structures present in the current design file ◦ <i>library</i> displays all the structures present in directories defined in the environment variable \$PAD_PATH <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p>⚠ Any structures in the library that share names with those already in the database will be excluded.</p> </div>
<i>Filter by name</i>	Specify name of structure to select
<i>Export Selected Structure</i>	Lets you export any structure that you select from your design or database/library list, and save the structure in a .xml-formatted file.
<i>Name</i>	Displays lists of all structures available in database or library
<i>Layer stackup for selected via structure</i>	Lists the layers on which the structure resides. This is for information-purposes only.
<i>Assign return path net</i>	Lets you select a net to assign to the return path via nets. Available only if the selected high-speed via structure has return path vias.
<i>Options</i>	<i>Include unassigned:</i> When checked, lets you attach the structure to elements that are not assigned to a net. <i>Free place structures:</i> When checked, allows you to place the structure wherever you click on the design. The aligned rotation option does not apply in this scenario.
<i>Rotation</i>	<i>Specify angle</i> lets you specify the placement angle of the structure. Default values are in 90-degree increments, but you can define an angle between 0 and 360 degrees. <i>Aligned</i> sets the rotation of the structure at the same angle as the element to which it is attached. This is typically used when you attach the structure from the end of the connect line segment. Note: Alignment is based on the angle of the selected element, either pins, vias, or connect lines. For pins and vias, the tool determines the angle by its rotation; for connect lines, the tool determines the angle by the segment to which the structure is connected.
<i>Description</i>	Displays details of above options.
<i>OK</i>	Completes the command and closes the dialog box.
<i>Cancel</i>	Exits the command without saving any changes.

Placing a Structure

To place a structure in your design:

1. Choose *Route – Structures – Place*.

The Place Structure dialog box appears.

2. From the dialog box, choose from what source you want to place the structure:

- *Pick from design* lets you choose a structure present in your design.
- *Pick from list* lets you choose a structure from the list in the dialog box.
 - *database* lists the structures defined in the current design
 - *library* lists the structures defined in a library of archived .xml files defined in the environment variable \$PAD_PATH

In either case, the selected structure attaches to your cursor at the structure's origin point.

3. Specify an angle of rotation.

4. To allow the structure to be attached to elements that are not assigned to a net, check *Include unassigned* in the dialog box.

5. Select a net to assign to the return path via nets if the selected structure has return path vias identified.

6. To snap structure to pins, vias, or clines, in the *Find filter* choose the element types to which you can attach the structure. The choices are *Pins*, *Vias*, and *Clines* (connect line segments).

In canvas, choose elements to which to attach the structure.

Do this by drawing a border around the selected elements, or by using *Temp Group* in the right -button pop-up menu. The structure attaches to your cursor as you go to place it.

If the padstack layers in the structure you are attempting to add differ in number or type from the layer structure of the active design, the *Via Structure Text In Layer Alignment* dialog box appears.

- a. Select either *Align to top layer* or *Align to bottom layer*.

7. Alternatively, to place a structure anywhere in design (without snapping to pins, vias, or clines), enable *Free place structures* checkbox. Click anywhere in canvas to place structure.

8. When all structures have been placed, click *OK* to close the dialog box and exit the command.

All instances placed will have the **LOCKED** property to prevent unintentional changes.

 You can edit the property by choosing *Edit – Properties*.

To edit individual objects of a via structure, right-click and choose *Unlock to Edit*. Similarly, you can right-click and choose *Lock Via Structure* to lock the structure, which is recommended to avoid unintentional changes.

Related Topics

- [place structure](#)

plane generator

The `plane generator` command creates Power and Ground planes based on the locations of pins or vias within the target area.

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

The plane boundary can range from a flood net to cover the layer where no other plane is poured, to regions defined by a bounding shape (convex hull) surrounding pin groups, to rectangular regions such as blocks around regular clusters of pins that identify power domains inside of the chip.

 You can only select the power and ground nets with Ratsnest Schedule and Voltage properties.

You can continue to change parameters as the design progresses and regenerate all the planes as and when needed.

Power Delivery Solution Generator Dialog Box

Access Using

- Menu Path: *Si Layout – Power Delivery – Power Delivery Generator.*

Control	Description
Select Plane Boundary	Select the region in the design as a shape.
Sequence	Specify the sequence in which shapes should be added to layers. If empty, shapes will not be created in the layer.
Layer	Specify the layer name.
Flood Net	Specify net to be used to fill areas without any nets.
Nets	Specify ordered list of nets to create power/ground regions. Dynamic fill priority decreases in sequence from the beginning of the list.
Style	Specify the style of shapes be created.
Corner	Specify the outline corner.
Grid Snap	Select to snap each vertex in the shape to the defined etch grid.
Pad Reference	Specify the layer to reference when grouping elements to define plane boundaries.
Pad Clearance	Specify the clearance between pad outline and the plane shape edge.
Minimum Pads	Specify the minimum pins required for pad or net square patterns. Pins are grouped to the minimum blocks and the blocks are combined recursively to form larger regions.
Pad Squares Pin Pitch	Specify the pad pitch for pin and net square patterns. The default is <AUTO>, where the value is automatically calculated.
Generate Planes	Click to generate power and ground planes for the target region and layers configured.
Remove Planes	Click to remove all existing planes from the active region and layers that are not fixed.
Save Planes	Click to save the last generated plane set by adding the FIXED property. Planes will not be regenerated, modified, or removed until unfixed.
Import	Import spreadsheet to specify via configuration.
Export	Generate a spreadsheet that can be modified externally and then imported using the Import option. Import using <i>Simple</i> for a minimal configuration and using <i>Complete</i> for detailed editing.

plc chglyr

An internal Cadence engineering command.

plctxt

The `plctxt` batch command lets you place a design from an ASCII text file of component positions and orientations, or generate a text file of placed component positions and orientations from a design. Each component is identified by X, Y location, rotation, mirrored status, and symbol name in a file whose name defaults to `place_txt.txt` created in or read from the current directory.

You can use the contents of a generated `place_txt.txt` file as input to either a new or an existing design. If you are updating an existing design and do not want to overwrite the design file, specify the `<out_design>` argument.

The command operates in two modes:

- Creates a `place_txt.txt` file containing the refdes, origin, rotation, mirror status, and embedded layers of all placed parts in a design.
- Uses an existing `place_txt.txt` file to place a board without affecting parts already placed.

 The command neither saves nor places mechanical or format symbols.

If you run the command without the `-r` switch, it extracts the placed components that appear in `<in_design>` into the `place_txt.txt` file.

For more details, see the *Placing the Elements* user guide in your documentation set.

For information on interactively placing or exporting components using a text file, see [plctxt in](#) or [plctxt out](#).

Syntax

```
plctxt [-r] [-s] [-c|-p] [-d] [-o] [-m] [-z]<in_design> [<out_design>] [<place_text_file>]
```

<code>-r</code>	Reads from a <code><place_text_file></code> of placement positions and places components as specified.
<code>-s</code>	If an error is detected while reading in a design, <code>plctxt</code> continues processing and saves the design. If you do not specify this argument, <code>plctxt</code> does not save the design.
<code>-c</code>	Optional. Places a component so its body center or subclass BODY_CENTER point occurs at the point specified in the <code><place_text_file></code> . If not specified, then uses symbol origin.
<code>-p</code>	Optional. Places symbol by origin of its Pin 1 location. If not specified, uses symbol origin.
<code>-m</code>	Optional. Move existing components. By default, add unplaced components.
<code>-z</code>	Optional. Ignores FIXED property.
<code>-o</code>	Optional. Creates <code><place_text_file></code> in an old format supported.
<code>-d</code>	Optional. Places symbol by default symbol present on component or if hard a component by symbol based its location.
<code>in_design</code>	Required. Identifies the design being placed or read from.
<code>out_design</code>	Optional. When <code><place_text_file></code> serves as input to a design, saves the resulting placed design in <code><out_design></code> . If not specified, overwrites <code><in_design></code> .
<code>place_text_file</code>	Default name is <code>place_txt.txt</code> . If you specify a name here, you must also specify an <code><out_design></code> name.

Running the Plctxt Command

1. Run `plctxt` from your operating system prompt.

If you enter the command without arguments, you are prompted for an existing design file name, an output file name, and a *<place_text_file>* name.

plctxt in

The `plctxt in` command lets you place components in a new or existing design, using the ASCII text file `place_txt.txt` that specifies component positions and orientations from another existing design. This command displays the Import Placement dialog box.

For more details, see the *Placing the Elements* user guide in your documentation set.

This command can be run in batch mode as [plctxt](#).

Related Topics

- [Importing Component Placement Information](#)

Import Placement Dialog Box

Access Using

- *Menu Path: File – Import – Placement*

<i>Placement File</i>	The ASCII text file <code>place_txt.txt</code> is shown. If you have more than one <code>place_txt.txt</code> , you must specify the full path to the directory that contains the desired file.
<i>Placement Origin</i>	<i>Symbol Origin</i> : Indicates you want to place the components using the symbol origin. This is the default. <i>Body</i> : Indicates you want to place the components using the component body center. <i>Pin 1</i> : Indicates you want to place the components using pin 1 of the component.
<i>Placement Options</i>	<i>Add unplaced components</i> : Add unplaced components at the specified origin. This is the default. <i>Add and move</i> : Add unplaced component and move the already placed component to the location specified in the <code>place_txt.txt</code> . <i>Ignore FIXED property</i> : Does not move components with FIXED property. This option is enabled by default. <i>Place by component's symbol</i> : Add unplaced components by Jedec type and ignores the symbol name specified in the <code>place_txt.txt</code> . If enabled, this option ignores alternate symbols if defined in the <code>place_txt.txt</code> .
<i>Import</i>	Runs the program.

Importing Component Placement Information

Follow these steps to import component placement information:

1. Run `plctxt in` to display the Import Placement dialog box.
2. Enter the name of the placement file you want to import.
3. Choose an option for *Placement origin*.
4. Specify *Placement options*.
5. Click *Import*.

When the program is completed, the following message appears:

Import completed.

The program places the components in the design file using the coordinate information in the placement file.

If errors are encountered, the log file displays.

1. Click *Close*.

You can review the results of the program in the `plctxt.log` log file in the current working directory.

Related Topics

- [plctxt in](#)

plctxt out

The `plctxt out` command generates an ASCII text file, `place_txt.txt`, of placed component positions and orientations from an existing design. You can use the placement file to re-create the design later, as required. When you choose this command, the Export Placement dialog box displays.

For more details, see the *Placing the Elements* user guide in your documentation set.

This command can be run in batch mode as `plctxt`.

Related Topics

- [Exporting Component Placement Information](#)

Export Placement Dialog Box

Access Using

- *Menu Path: File – Export – Placement*

<i>Placement File</i>	Indicates the ASCII text file <code>place_txt.txt</code> that is created from your design and is placed in your current working directory. If you want to place the file in another directory, you must specify the full path to the directory.
<i>Placement Origin</i>	<i>Symbol Origin:</i> Indicates the component locations are to be written to the file using the component symbol origin. This is the default. <i>Body:</i> Indicates the component locations are to be written to the file using the component body center. <i>Pin 1:</i> Indicates the component locations are to be written to the file using the component pin 1 location.
<i>Export</i>	Runs the program.

Exporting Component Placement Information

Follow these steps to export component placement information:

1. Run `plctxt out` to display the Export Placement dialog box.
2. Enter the placement file name.

 If you create more than one file using the default name (`place_txt.txt`), make sure they are in different directories.

3. In *Placement Origin*, choose how the component locations are referenced in the text file.

4. Click *Export*.

When the program is completed, the following message appears:

Export completed.

If errors are encountered, the log file displays.

5. Click *Close*.

You can review the results of the program in the `plctxt.log` log file in the current working directory.

Related Topics

- [plctxt out](#)

plot

The `plot` command lets you preview a plot as it will look when printed. When you choose `plot preview`, the user interface changes to preview the active design as it will plot based on the setup parameters in the *Plot Setup* dialog box and/or the Windows *Print Setup* dialog box.

- (i) Windows and Unix operating systems handle plotting differently. See the sections on plotting that are appropriate to the operating system you use.

On Unix operating systems, successful plotting involves correct set-up and the creation of IPF and control files, as well as the `.cdsplotinit` plotter configuration file, which lists available printers/plotters. The `.cdsplotinit` file must reside in `<install_path>/tools/plot`, the current working directory, or your home directory. See *Preparing Manufacturing Data* in the user guide.

On Unix, the layout editor recognizes the *Vectorize text* setting on the *Plot Setup* dialog box, available with the `plot setup` command, to permit direct plotting of nonvectorized text with the *File – Plot* command.

For detailed documentation on various aspects of plotting, see *Preparing Manufacturing Data* in the user guide . For information on discrete plotting commands, see [allegro_plot](#), [create plot](#), [gbplot](#), and [plot setup](#).

Related Topics

- [Plotting Prerequisites on a Unix Workstation](#)
- [Plotting Your Design on Unix](#)
- [Plotting Your Design on Windows](#)

Print Dialog Box

Access Using

- *Menu Path: File – Plot*

The `plot` command on Windows runs the standard *Windows Print* dialog box.

On Unix, the `plot` command runs the *Plot* dialog box, that contains the following controls:

<i>Print To File</i>	Indicates the plot file is to be sent to the named file.
<i>Printer Name</i>	Indicates the name of the plotter the plot file is to be sent to.
<i>Pen Numbers</i>	Displays the <i>Plot Preference</i> dialog box for assigning colors to pens.
<i>Cancel</i>	Ignores input and closes the dialog box.
<i>OK</i>	Creates the plot and closes the dialog box.

Related Topics

- [Plotting Your Design on Unix](#)
- [Plotting Your Design on Windows](#)

Plotting Prerequisites on a Unix Workstation

To run the `plot` command or the `allegro_plot` program on a Unix workstation, a plotter configuration file named `.cdsplotinit` must reside in `<install_path>/tools/plot`, the current working directory, or your home directory.

If a `.cdsplotinit` file resides in multiple locations, the program looks down each path in turn and adds any new information or replace any old information as it is found.

The `.cdsplotinit` file contains information vital to the operation of the `allegro_plot` program such as:

- the name of the output device,
- the output format to be used for the device,
- the paper sizes available for the device,
- the maximum number of pages allowed on the device,
- the Unix commands for spooling jobs to the queue,
- checking the jobs in the queue
- removing jobs from the queue on the device
- other device specific information

The following is a sample `.cdsplotinit` file entry:

```
bos1|Apple LaserWriter II NT/NTX: \
:manufacturer=Apple Computer: \
:spool=lpr -Pbos1: \
:query=lpq -Pbos1: \
:remove=lprm -Pbos1 $3: \
:type=postscript1: \
:maximumPages#30: \
:resolution#300: \
:paperSize="A" 2400 3150 75 75: \
:paperSize="A4" 2332 3360 60 60:
```

For detailed information on setting up the `.cdsplotinit` file, refer to the *Plotter Configuration User Guide*, available on Cadence Online Support.

Related Topics

- [plctxt out](#)
- [Plotting Your Design on Windows](#)

Plotting Your Design on Unix

To plot your design on unix, follow these steps:

1. Before running `plot`, you must set up your plotting parameters as described in the procedure section of [plot setup](#). You must also have created a plotter configuration file, as described in the section above.
2. When setup is complete, run the `plot` command to display the *Plot* dialog box.
3. To direct output to a file, choose *Print to file*. To write the design to the current working directory, enter only a filename. To direct the design file to another location, enter the full path.
—or—
To direct output to a plotter, choose *Printer name* and choose the printer name from the drop-down menu.
4. If necessary, click *Pen Numbers* to make color-to-pen assignments in the Plot Preference dialog box.
5. For each pen assignment you want to change, highlight the pen number and enter a new number.
Each number corresponds to a pen on your plotter. You assign each color in the palette to a corresponding pen number. If there are more colors in your drawing than there are pens in your plotter, assign more than one color to each pen. You should not have a number on your palette higher than the pen numbers in your plotter.
6. Click *OK* to close the dialog box.
7. In the *Plot Setup* dialog box, click *OK* to print or create the design file.

Related Topics

- [plctxt out](#)
- [Print Dialog Box](#)

Plotting Your Design on Windows

Follow these steps to plot your design on windows:

1. Run the `plot` command to display the *Print* dialog box.
2. Choose the print resolution in the *Print quality* field.
If you want to direct output to a file, check *Print to file*. To write the design to the current working directory, enter only a filename. To direct the design file to another location, enter the full path.
3. If necessary, click *Setup* to set additional printing options in the Windows *Print Manager* dialog box.
4. Click *OK* to print or create the design file.

Related Topics

- [plctxt out](#)
- [Print Dialog Box](#)
- [Plotting Prerequisites on a Unix Workstation](#)

plot preview

The `plot preview` command lets you preview how an active design appears when sent to a printer. The preview conforms to the settings in the *Plot Setup* and the Windows *Print Setup* dialog boxes. This feature temporarily replaces the editor user interface with a representation of the plotted design.

 This feature is available only on Windows operating systems.

Access Using

- *Menu Path:* *File – Plot Preview*

Previewing an Active Design

To preview an active design:

1. Run `plot preview`.
The user interface of the tool is replaced with a representation of the plotted design.
2. Zoom in to a specific area of the design preview by positioning the cursor at the desired location. To enlarge the display without regard for cursor position, click *Zoom In*. Click *Zoom Out* to reduce the display size.
3. To close the preview window and return to the tool editor, click *Close*. Do not use the Microsoft control button to close the window; doing so exits you from the tool.

plot setup

The `plot setup` command lets you set parameters for plotting a design. (See [plot](#) for additional details on plotting.) Although plotting procedures vary according to the operating system you are running, the procedure for `plot setup` is the same for Unix and Windows.

Related Topics

- [Setting Parameters for Plotting a Design](#)

Plot Setup Dialog Box

Access Using

- *Menu Path: File – Plot Setup*

⚠ The .ini file retains parameters set in the *Plot Setup* dialog box. Therefore, they remain in effect for every database you open until you change the parameters.

General Tab

Plot scaling	<i>Fit to page</i> : Indicates the plot file is to be scaled to fit the entire plotted page. <i>Scaling factor</i> : Indicates the scale of the finished plot. <i>Default line weight</i> : Converts any zero width line to a width proportional to the setting. Aids in displaying very thin lines on high-resolution output.
Plot orientation	<i>Auto center</i> : Centers the design on the plot page. This control automatically invokes when you choose the Fit to page setting. <i>Mirror</i> : Flips the design end-for-end about the Y axis. Useful for viewing top and/or bottom layers.
Plot method	<i>Color</i> : Directs the output to print in color. Color is determined by the method specific to the platform you are using. On Unix, color is read from a user-supplied stipples file (<i>allegro_plot_param.stipples</i>); if a stipples file is not found, plotter color defaults are used. On Windows, color selection is determined by setting in the Color dialog box. <i>Black and white</i> : Directs the output to print in black and white.
Plot contents	<i>Screen contents</i> : Prints/plots the contents of what is currently displayed in the design area of the user interface. <i>Sheet contents</i> : Prints/plots the extents of all currently-visible graphics within the design (not the drawing extents).
IPF setup	<i>Vectorize text</i> : Specifies that text output to the IPF file is broken into line vectors. The environment variable PLOT_VECTORIZE_TEXT determines whether the <i>Vectorize text</i> option is enabled or disabled by default. <i>Width</i> : specifies the width for simulating the text characters. The width used is established with the environment variable PLOT_VECTEXT_WIDTH. The default is 0. When <i>Vectorize text</i> is enabled, and a negative value is entered in the <i>Width</i> field, any other width setting of 0 or greater causes photoplot widths to be ignored, and all text is uniformly stroked with the same specified width. The <i>Vectorize text</i> and <i>Width</i> settings apply as specified when the <code>create plot</code> command executes. Environment variables are the initial default settings for the Plot Setup dialog box. If you modify the settings in the dialog box, the new settings override any environment variable settings that you may have specified. When you exit the editor, the current dialog box settings are saved in the .ini file. These .ini file settings are then used in the next session, and again override any specified environment variable settings. If you modify the environment variable settings after changing settings in the <i>Plot Setup</i> dialog box, the editor does not use these new environment variable settings. You must delete the .ini file, and then the editor can use the new environment variable settings. You can set environment variables using procedures, based on the platform you are running. You can also set these environment variables in the editor using <i>Setup – User Preferences</i> .
OK	Saves the settings and closes the dialog box.
Cancel	Closes the dialog box without saving the settings.

Windows Tab

Only appears on the Windows platform. The .ini file retains all settings between design sessions.

Non-vectorized Text Control

Non-Vectorized Text	Choose to generate plot files with true font text, which lets you generate PDF-format plot output with searchable text.
Font	Specifies a font to use; defaults to Courier.
Font Height	Enter a percentage scaling factor for the character height to closely match font text with that of the layout editor's normal vectorized text display/plot.
Font Width	Enter a percentage scaling factor for the character width to closely match font text with that of the layout editor's normal vectorized text display/plot.
View Available Fonts	Click to review the available text fonts for the plot device.

Margin Control

<i>Margin Width</i>	Specify the desired margin width in user units. The default equates to 0.25 inches, or 0.0 if the <code>noplotmargins</code> environment variable is set.
<i>OK</i>	Saves the settings and closes the dialog box.
<i>Cancel</i>	Closes the dialog box without saving the settings.

Setting Parameters for Plotting a Design

Follow these steps to set parameter for plotting a design:

1. Adjust the visibility of the display layer and the view (zoom) level.
2. Run `plot setup` to display the Plot Setup dialog box.
3. Set plot parameters as described in the section above. Parameters that you set in Plot Setup are retained in the `.ini` file. Therefore, they remain in effect for every database you open until you change the parameters.
4. Click *OK* to save the settings.

Related Topics

- [plot setup](#)

plotwint

An internal Cadence engineering command.

polar

The `polar` command lets you enter a selection (pick) point (P) specified by polar coordinates. Specify polar coordinates by radius, then angle. Radius is the distance from the origin to P; angle, is the angle from the positive x axis to the ray joining the origin with P. Polar coordinates make it easier, for example, to add a fixed radius arc to the drawing and symbols at odd angles.

You cannot use a polar coordinate for entering the first point in a command; polar coordinates must have an origin point, which must first be chosen or typed in. You can enter the `polar` command with the values for radius and angle in the command line or enter the command without values, in which case it prompts you for the radius and angle.

Syntax

```
polar <radius> <angle>
```

Entering Selection Points Specified by Polar Coordinates

Perform these steps to enter selection points specified by polar coordinates:

1. Run a command
2. Type `polar` in the user interface command console.
Dialog boxes prompt you for the radius and angle.
or
Type `polar` followed by radius and angle values.

Example

<code>polar</code> <code>600 45</code>	The first number (<code>600</code>) is the radius (distance between the first point, already specified (the origin) and the second point (P) (the point to be created), and the second number (<code>45</code>) is the angle in degrees from the x axis of the line from the origin to P.
---	---

pop add

The `pop add` command is used in conjunction with the `package height` command. It allows you to add a shape while the command is active.

Access Using

Pop commands are accessed through the right-click pop-up menu that is used in conjunction with various interactive commands.

pop addrect

The `pop addrect` command is used in conjunction with the `package height` command. It allows you to add a rectangle while the command is active.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop align

Use the `pop align` command in conjunction with the `spin` command to designate an element with which the elements you spin must align.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Using the Pop Align Command

To align the rotation of components with respect to other components in the design:

1. Choose Set Rotate Angle from the pop-up menu or type `pop align` in the user interface command console.
The command prompts you to

Select element to align with.

1. Choose the element to control the alignment.

The command only finds elements to align with that have a rotation angle associated with them. These include package, mechanical, and format symbols, lines and connect lines, text, and filled and unfilled rectangles. The command does not recognize arcs, circles, or shapes, since it has no way of deriving an alignment angle from them. The command highlights the element you chose to be the one with which the rotated elements are to be aligned and sets the spin command in the *Options* panel field *Rotation Type* to Absolute and the *Angle* field to the angle of the alignment element.

1. Choose elements and they spin to that absolute angle.

pop alt module

Not in use.

pop alt symbol

Use the `pop alt symbol` command during interactive placement to substitute an alternate package symbol type for placement. You must already have specified the alternate symbol name(s) by attaching the ALT_SYMBOLS property to the component definition.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Substituting an Alternate Package Symbol

Follow these steps to substitute an alternate package symbol:

1. Choose Alternate Symbol from the pop-up menu or enter `pop alt symbol` in the user interface command console.
The alternate symbol definition is loaded into the cursor buffer.
2. Choose Alternate Symbol again.
Each time you invoke the Alt Symbol , the `place manual` command cycles to the next alternate symbol defined for that device type. When it has cycled through all alternate symbols, it loads the original definition, so you can cycle repeatedly.
3. Once the alternate symbol is in the cursor, choose the location for placement.
The `place manual` command adds the alternate symbol to the layout for that reference designator.

pop apply

Saves modifications you made with an interactive command.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop autovoid

The `pop autovoid` command automatically generates voids for static positive shapes on positive etch layers. Voids are created around any elements that fall inside the shape, such as connect lines, pins, and vias (based on antipad definitions in the padstacks for your pins and vias).

Executes when you choose *Shape – Manual Void – Element* (`shape void element` command), right mouse click, and choose *Void All* from the pop-up menu that displays.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop bbdrill

The `pop bbdrill` command adds a blind/buried via at the point chosen when creating a line using `add connect`. The tool swaps the active layer with the layer defined in the `bbvia padstack` form. If there is more than one defined `bbvia` that includes the current active layer, a dialog box lists the choices; choose the one you want by selecting from the list. (See also [pop drill](#).)

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Adding a Blind/Buried Via to the Design

To add a blind/buried via to your design, follow these steps:

1. Run `add connect`.
2. Create a cline.
3. Choose Add Via from the pop-up menu or by typing `pop bbvia` in the user interface command console.
A blind/buried via is added to the chosen point of the line.

pop change layer

An internal Cadence engineering command.

pop controltrace

The `pop controltrace` command is only available when group routing or routing differential pairs. During group routing, it allows you to change the control trace that the editor chose. 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.

Access Using

Choose *Route – Connect*. While performing group routing or routing differential pairs, click the right mouse button to display the pop-up menu. Choose *Change Control Trace*.

pop cut

The `pop cut` command cuts line segments during the `slide`, `delete`, or `change` commands. In `slide`, the chosen line segment becomes dynamic and the command prompts for a destination point. When you choose the destination, the command redraws the line segment, according to the parameters specified in the *Options* panel.

In `delete`, the line segment highlights. Selecting either Done or selecting another element completes the deletion and the highlighted segment disappears. Once segments have been deleted they cannot be retrieved.

In `change`, the line segment immediately redefines itself according to the parameters specified in the *Options* panel.

The `pop cut` command has effect for one piece of line segment only. To use `pop cut` again, you must choose it again.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Using the Pop Cut Command

Follow these steps to use the `pop cut` command:

1. Choose Cut from the pop-up menu or type `pop cut` in the user interface command console.
2. Click the cut start point on a line segment.
The command prompts for a second selection along the line segment.
3. Make the selection.
The command executes on the chosen piece of segment—sliding, deleting, or changing it.

pop datum

Use `pop datum` during the `dimension datum` command to establish a new datum point.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Creating a New Datum Point

Perform these steps to create a new datum point:

1. Run the `dimension datum` command.
2. Choose Change Datum from the pop-up menu or enter `pop datum` in the user interface command console.
The command prompts you to

Pick element or point for NEW DATUM location.

1. Choose an element or point.
The `dimension datum` command treats the point you choose as the new datum reference point.

pop delete all voids

The `pop delete all voids` command deletes all user-defined voids in a dynamic shape and all voids in a static shape.

Executes when you choose *Shape – Manual Void – Delete* ([Shape void delete](#) command), right mouse click, and choose *Delete All Voids* from the pop-up menu that displays.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop delete vertex

Use the Delete Vertex command in the Edit Vertex command to delete a vertex you have already chosen for editing.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Deleting a Vertex

To delete a vertex:

1. Run `vertex`.

2. Choose a vertex.

3. Choose Delete Vertex from the pop-up menu or by type `pop delete vertex` in the user interface command console.

The command deletes the chosen vertex and joins the far end points of the affected segments with a single segment. Connect line connectivity and shape/void closure remain after vertex deletion.

pop dimension value

The `pop dimension value` command is used in conjunction with the dimension/draft commands. It allows you to add a dimension value at a location you select.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop drc window

An internal Cadence engineering command.

pop drill

The `pop drill` command adds a through hole via at the most recent point chosen when creating a connect line using `add connect`. The command uses the via chosen in the *Options* panel for the through hole, then swaps the active and alternate layers. Any segments added after the through hole add on the "new" active layer. (See also [pop bbdrl](#)l.)

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Adding a Through Via Hole to the Design

Activate `pop drill` by selecting Add Via from the pop-up menu or by typing `pop drill` at the command line.

pop dyn_option_select

An internal Cadence engineering command.

pop finish

The `pop finish` command completes the current action of various interactive commands. It performs the same function as the *Finish* item in the right-button pop-up menu. This command does not return you to the idle state.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop fix

Refer to the [fix](#) command, which assigns the FIXED property to elements without requiring the use of the *Edit Property* dialog box. A database element that is "fixed" is restricted from additional modification. For example, mechanically placed components or critical high-speed nets often are fixed to prevent accidental movement or deletion. Fixed nets are not ripped up during auto-routing, nor updated during glossing.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop flip

Use the `pop flip` command during the `boundary` command to reverse the sense (from clockwise to counterclockwise or vice versa) of the tangent arc rubberband at the starting selection.

The Flip Rubberband command exists because, when Link Lock in the *Options* panel is set to Arc , Edit Boundary starts arc segment edits with a tangent arc rubberband turning in the same sense as the edited arc. This can be either clockwise to counterclockwise.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Using the Pop Flip Command

Follow these steps to use the `pop flip` command:

1. Run `boundary`.

You are prompted to pick an edit starting point on a shape or void boundary.

2. Choose the starting point.

You are prompted to pick the next vertex or closing point on the shape.

3. Make the selection.

4. If you want to edit the arc in the opposite direction, choose **Flip Rubberband** from the pop-up menu or type `pop flip` in the user interface command console.

The `boundary` command responds by reversing the direction of the rubberband arc from clockwise to counterclockwise or vice versa.

pop in

Use `pop in` during the `zoom points` command to display a smaller area rather than windowing into a smaller part of the displayed area. Activate `pop in` by selecting `In` from the pop-up or entering `pop in` on the command line, before either the first or second selection of the command. The `zoom points` command displays a dynamic window with dynamic rays to the corners of the layout display. The dynamic window part of this pattern identifies the area that the entire current window occupies when the re-display completes. The command redisplays after the second selection.

pop jinput

The `pop jinput` command is used in conjunction with the `add text` and `edit text` commands. It allows you to enter Kanjii characters by way of a special user interface.

 This feature is available only for Unix users on Sun and Hewlett-Packard platforms.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop lasso

An internal Cadence engineering command.

pop mirror

The `pop mirror` command mirrors the active component in the cursor buffer during the `place manual` command. Use the `pop mirror` command to mirror a component that is already placed.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Mirroring a Selected Component

Follow these steps to mirror selected components in your design:

1. Run `place manual`, and choose an element for placement.

The element displays in the cursor buffer with the orientation defined in the *Options* panel.

1. To change the mirroring before placing the active component, choose Mirror from the pop-up menu, or type `pop mirror` at the command line.

The symbol and any associated text or etch appear mirrored in the buffer.

1. Choose a placement location.

The mirrored symbol is added to the layout.

pop mirror geometry

The `pop mirror geometry` command is similar to `pop mirror`. It allows you to specify how objects should be mirrored; for example, on the same layer, non-3D (symbols, vias, and pins).

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop move

The `pop move` command is used in conjunction with active commands and is typically used in scripts and for debugging purposes.

pop neck

With `add connect active`, the `pop neck` command adds the segment resulting from the next selection at the neck width defined in the Net Layer Rule for the active layer. Activate Neck by selecting Neck from the pop-up menu or by typing `pop neck` at the command line.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop net list

This command is obsolete.

pop net name

This command is obsolete.

pop new target

While `add connect` is active, processing the `pop new target` command causes the automatic router to allow the choice of a new target pin. You can invoke the automatic router to complete the connection by selecting **Finish** from the pop-up menu. If you want the automatic router to make the connection to a pin other than the one it is rubberbanded to, then choose **New Target** or type `pop new target` at the command line before selecting **Finish**. If you choose a target pin not in the netlist as part of that net, then the program displays a dialog box with the message

Not authorized to execute router.

You must confirm the message. The program does not try to route to that pin.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop no target

The `pop no target` command is used in conjunction with `add connect`. It disables the routing command's ability to create a ratsnest line to the default target.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop out

Use `pop out` during the `zoom points` command to display a larger area rather than windowing into a larger part of the displayed area. Activate `pop out` by selecting Out from the pop-up or entering `pop out` on the command line, before either the first or second selection of the command. The `zoom points` command displays a dynamic window with dynamic rays to the corners of the layout display. The dynamic window part of this pattern identifies the area that the entire current window occupies when the re-display completes. The command redisplays after the second selection.

pop path

An internal Cadence engineering command.

pop pickgroup

This command is obsolete.

pop prmed

Refer to the [prmed](#) command, which displays the Design Parameter Editor, a centralized location for editing parameters that are saved and stored in the database. In the Design Parameter Editor, select tabs for *Display*, *Design*, *Text*, *Shapes*, *Flow Planning*, *Route*, and *Mfg Applications* and edit the specific parameters in each of these categories.

You may also access the Design Parameter Editor by right-clicking anywhere in the design canvas to display the *Quick Utilities* pop-up menu from which you may choose *Design Parameters*.

pop properties

This command is obsolete.

pop readfile

The `pop readfile` command is used in conjunction with `add text`. It lets you specify a file name and read it into the design database.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop refdes list

This command is obsolete.

pop refdes name

This command is obsolete.

pop region list

An internal Cadence engineering command.

pop route_from_target

The `pop route_from_target` command is used in conjunction with `add connect`. It swaps 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.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop routespace

The `pop routespace` command is executed when you choose the *Route Spacing* from the pop-up menu during group routing. Use this command with command line arguments and the `alias` command to support space changes during group routing. When you provide the additional arguments, the *Route Spacing* dialog box does not appear.

Access Using

Choose *Route – Connect*. While performing group routing, click the right mouse button to display the pop-up menu. Choose *Route Spacing* and then choose a spacing mode.

Syntax

```
pop routespace <spacing mode> [spacing value]
```

<code>spacing mode</code>	The choices are: <i>initial</i> – Sets the mode to Initial Space. <i>mindrc</i> – Sets the mode to Minimum DRC. <i>user</i> – Sets the mode to User-defined.
<code>spacing value</code>	Applicable only if the <i>spacing mode</i> is <i>user</i> . If provided, the value is used to update the user-defined spacing value. If not provided, the user-defined spacing value remains unchanged.

Example

Typically, you use this command with the `alias` command in your global environment file to choose spacing modes during group pair routing. You can also use this command in a script.

This example shows how to create an alias for the `pop routespace` command. To create an alias for the current editor session:

1. Create an alias, using the `alias` command, the function key you are specifying, and the `pop routespace` command. For example, in your `env` file, type:

```
funckey u 'settoggle routespace mindrc user current; pop routespace $routespace; echo routespace = $routespace'
```

This example sets up the `SF10` function key to sequence the spacing mode among these options: *mindrc*, *user-defined*, and *initial space*.

2. Open a design that contains a bus.
3. Choose *Route – Connect* and choose the nets that you want to route.
4. Press `u` to change the spacing.

Each time you press `SF10`, the spacing for group routing changes according to the values specified for the spacing modes.

5. Continue routing.

pop scale

The `pop scale` command is used in conjunction with `load plot`. It allows you to set parameters in design units to scale the incoming plot.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop shape change type

The `pop shape change type` command changes shape fill type from Static Solid to Dynamic Copper or visa versa. When you uprev legacy boards, their shapes' shape fill type is Static Solid. You can change the shape fill type for more than one board at a time.

i You may also change shape fill type from Dynamic Copper to Static Solid at the end of production to preserve its current state. When you do so, however, the following important shape information is lost:

- original shape boundary
- dynamic shape parameter settings
- user-defined (manual) void information

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop shape copy

The `pop shape copy` command creates duplicates of shapes in your design.

Executes when you choose *Shape – Select Shape or Void*, right mouse click, and choose and *Copy* from the pop-up menu that displays.

Access Using

Choose *Shape – Select Shape or Void*. Click the right mouse button to display the pop-up menu. Choose *Copy*.

pop shape copy layers

The `pop shape copy layers` command replicates a shape on the chosen subclass layers.

Executes when you choose *Shape – Select Shape or Void*, right mouse click, and choose and *Copy to Layers* from the pop-up menu that displays.

Access Using

Choose *Shape – Select Shape or Void*. Click the right mouse button to display the pop-up menu. Choose *Copy to Layers*.

pop shape defer dyn fill

The `pop shape defer dyn fill` command prevents a currently chosen shape from dynamically updating.

Executes when you choose *Shape – Select Shape or Void*, right mouse click, and choose *Defer Dynamic Fill* from the pop-up menu that displays.

Access Using

Choose *Shape – Select Shape or Void*. Click the right mouse button to display the pop-up menu. Choose *Defer Dynamic Fill*.

Deferring Dynamic Copper Fill for a Single Dynamic Shape

Perform these steps to defer dynamic copper fill for a particular dynamic shape:

1. Choose *Shape – Select Shape or Void* ([shape select](#) command). The command window prompt displays the following message:
Pick a shape or void to edit
2. Choose the shape for which you want to defer automatic update and filling.
3. Right mouse click and choose *Defer Dynamic Fill* from the pop-up menu that displays. The command window prompt displays the following message:
Unfilling shape in progress
The shape immediately unfills.

pop shape delete vertex

The `pop shape delete vertex` command deletes the vertex in a shape and restores the original line. For a rectangle, the shape resembles the original rectangle but its type remains as a polygon.

For additional related information on working with shapes, see Preparing for Layout in the user guide.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop shape edit boundary

The `pop shape edit boundary` command redefines the boundary of the copper area shape or its voids. You can edit a polygonal shape or void boundary, circular void, and arcs. You can define the new boundary inside or outside the old boundary, but you cannot cross any shape or void boundary with the new definition. A gravitation priority mechanism ensures that if you pick near a corner or an edge, the system automatically snaps to the corner if it's closer, or to the boundary edge. Snapping to the edge occurs on the intersection of a normal vector to the edge.

Executes when you choose *Shape – Select Shape or Void*, right mouse click, and choose *Edit Boundary* from the pop-up menu that displays.

Access Using

Choose *Shape – Select Shape or Void*. Click the right mouse button to display the pop-up menu. Choose *Edit Boundary*.

Changing a Shape or Void Outline

Perform the following steps to change the outline of a shape or a void:

1. Choose *Shape – Select Shape or Void*.

The command window prompt displays:

Pick shape or void to edit.

1. Choose a shape by clicking on it. Handles appear on the chosen shape. The command window prompt displays the following message:

Pick vertex or segment to edit

2. Right mouse click and choose *Edit Boundary* from the pop-up menu that displays.
3. Choose a starting point on the boundary of the chosen shape, or any of its voids.

The command window prompt displays:

Please pick edit starting point on shape boundary.

1. Choose the next point of the new boundary, and continue choosing points until the edit is complete.

2. To complete the edit, choose a closing point on the boundary.

The command deletes the original boundary section and replaces it with the new one.

3. Right mouse click to display the pop-up menu and choose:

Done to exit the command.

Oops to undo last segment. If no segments remain, undoes the pick location that started edit boundary operation. *Oops'ing* back past the first pick of edit boundary undoes previous edit operations on the active shape.

Cancel to terminate the edit boundary process and revert the boundary to its prior state.

Next to edit another shape boundary.

pop shape move

The `pop shape move` command repositions the entire shape you chose. Right mouse click again to rotate the chosen shape by choosing *Rotate* from the pop-up menu that displays.

Executes when you choose *Shape – Select Shape or Void*, right mouse click, and choose *Move* from the pop-up menu that displays.

Access Using

Choose *Shape – Select Shape or Void*. Click the right mouse button to display the pop-up menu. Choose *Move*.

Moving an existing shape or void

Perform the following steps to move a shape or a void in the design:

1. Choose *Shape – Select Shape or Void* ([shape select](#) command).The command window prompt displays the following message:
Pick a shape or void to edit
2. Right mouse click and choose *Move* from the pop-up menu that displays.
3. To rotate the chosen shape, right mouse click again and choose *Rotate* from the pop-up menu that displays.The command window prompt displays the following message:
Spin the element(s)
4. Move the mouse as required to rotate the shape's position.
5. Right mouse click and choose *Done* from the pop-up menu that displays.

You can also use the *Edit – Move* ([move](#) command) to move the shape or enable the *shape_drag_move* board level environment variable in the *User Preferences* dialog box, available by running the [enved](#) command.

pop shape param

The `pop shape param` command specifies shape outline parameters on a shape-specific basis and displays the *Shape Instance Parameters* dialog box. For additional related information on working with shapes, see Preparing for Layout in the user guide.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop shape raise priority

The `pop shape raise priority` command assigns the higher priority to a dynamic shape during dynamic filling and voiding when two shapes overlap, causing the higher-priority shape to plow into the other shape. By default, the first dynamic shape added to a design has the highest priority. Use this command to override this default assignment. You can only choose one dynamic shape at a time. The layout editor lists the shape priorities in the *Shape Report (Tools – Reports)* or by using *Display – Element (show element)* command on the chosen shape.

Executes when you choose *Shape – Select Shape or Void*, right mouse click, and choose *Raise Priority* from the pop-up menu that displays.

Access Using

Choose *Shape – Select Shape or Void*. Click the right mouse button to display the pop-up menu. Choose *Raise Priority*.

Assigning a Higher Voiding Priority to a Dynamic Shape

To assign a higher voiding priority to a dynamic shape, follow these steps:

1. Choose *Shape – Select Shape or Void*, then right mouse click and choose *Raise Priority* from the pop-up menu that appears. The command window prompt displays the following message:

Pick dynamic shape

2. Click on the overlapping dynamic shape to which to assign a higher priority.

The shape's priority is immediately updated

pop shape report

The `pop shape report` command produces the *Dynamic Shape* report that lists shape settings; generation results, including number of dynamic etch/conductor shapes and their areas; shape fill type; thermal relief connects; void controls; and clearance settings in your design. The information can be printed or displayed on your terminal screen as individual reports, or you can combine individual reports into a single appended `output.rpt` file and view the results.

For additional related information on working with shapes, see Preparing for Layout in the user guide.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop shape select

The `pop shape select` command executes when you choose *Shape – Select Shape or Void* ([shape select](#) command) and lets you choose a shape, void or filled rectangle for editing or changing parameters at the shape instance level. When you choose a shape, void or filled rectangle, edit handles appear, which are small rectangles or circles at all vertices of the shape boundary, allowing you to move and resize it. Double clicking the left mouse button on any edge also chooses a shape.

For additional related information on working with shapes, see [Preparing for Layout](#) in the user guide.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop shape update

An internal Cadence engineering command.

pop show

This command is obsolete.

pop skip

The `pop skip` command is used in conjunction with `place manual`. It lets you skip the next element in the placement list.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop swap

With `add connect active`, the `pop swap` command swaps the active layer for the alternate layer.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Swapping the Active Layer for an Alternate Layer

To swap the active layer for an alternate layer in your design:

1. Run `add connect`.
2. Create a cline segment.
3. Choose Swap from the pop-up menu or type `pop swap` in the user interface command console.
4. The active layer swaps to the alternate layer, as shown in the *Options* panel.

pop swap pin assignment

With `add connect active`, the `pop swap pin assignment` command swaps the active pin for the alternate pin.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Swapping an Active Pin for the Alternate Pin

Perform the following steps to swap an active pin for an alternate pin:

1. Run `add connect`.
2. Create a pin.
3. Choose Swap from the pop-up menu or type `pop swap` pin assignment in the user interface command console.
4. The active pin swaps to the alternate pin, as shown in the *Options* panel.

pop unfix

Refer to the [unfix](#) command, which lets you quickly remove the FIXED property from objects without opening the Edit Properties dialog box. The command works with the Find Filter settings in the Control Panel. The Find Filter retains the button values from the last command used. You can use the previously set values or change them. To specify the objects, use either single picks or draw a window around them.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

pop unfixall

The `pop unfixall` command deletes the FIXED property from all objects.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Deleting the FIXED Property from All Objects

To delete the FIXED property from all objects in the design: follow these steps:

1. Run the unfix command.
2. Click the right mouse button in the Design window and choose Unfix All from the pop-up menu. The following message appears in the command window:

Deleted <number> FIXED properties.

Click the right mouse button and choose Done from the pop-up menu.

pop window

The `pop window` command enables an Edit command to operate on multiple design elements which you specify by enclosing them in a window.

Access Using

Pop commands are accessed through the right-button pop-up menu that is used in conjunction with various interactive commands.

Using the Pop Window Command

To use the pop window command:

1. During an `edit` command, choose Window from the pop-up menu or type `pop window` in the user interface command console.
2. Specify the window by selecting two opposite corners.
Any element wholly or partially included in the window highlights and is operated on by the active `edit` command.

pop wirebond add

Refer to the [wirebond add](#) command. Run the `wirebond select` command, choose *Pins* in the Find Filter, and choose the *Add* option from the pop-up menu.

pop wirebond add guide

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Comps* in the Find Filter, and choose the *Add Guide* option from the pop-up menu.

pop wirebond add jumper

For internal use only.

pop wirebond add route

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Advanced – Add Routing Channels* option from the pop-up menu.

pop wirebond adjust drc

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Finger* in the Find Filter, and choose the *Adjust Min DRC* option from the pop-up menu.

pop wirebond auto bond

Once you select *Components* in the Find Filter, type this command at the prompt to perform automatic auto bonding to your design.

pop wirebond best fit path

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Advanced – Create Best Fit Path* option from the pop-up menu.

pop wirebond center

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Center Wires* option from the pop-up menu.

pop wirebond chg group

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* or *Bond wires* in the Find Filter, and choose the *Change Group* option from the pop-up menu.

pop wirebond copy finger left

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, right-click and choose the *Copy Left* option from the pop-up menu.

pop wirebond copy finger right

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, right-click and choose the *Copy Right* option from the pop-up menu.

pop wirebond copy guide

Refer to the [wirebond select](#) command. Run the `wirebond manage guide paths` command, choose *Bond wires* in the Find Filter, and choose the *Copy* option from the pop-up menu.

pop wirebond create ring

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Advanced – Create Ring* option from the pop-up menu.

pop wirebond create unwired

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Lines* in the Find Filter, right-click and choose the *Create Non-wired Finger* option from the pop-up menu.

pop wirebond cut shape

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Shapes* in the Find Filter, and choose the *Cut* option from the pop-up menu.

pop wirebond default

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Pins*, *Fingers*, *Bond wires*, *Lines*, or *Shapes* in the Find Filter, and choose the *Set/Unset Default Action* option from the pop-up menu.

pop wirebond delete

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Delete* option from the pop-up menu.

pop wirebond delete wires

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Bond wires* in the Find Filter, and choose the *Delete* option from the pop-up menu.

pop wirebond del route

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Advanced – Delete Routing Channels* option from the pop-up menu.

pop wirebond edit guide

Refer to the [wirebond select](#) command. Run the `wirebond manage guide paths` command, choose *Lines* in the Find Filter, and choose the *Edit* option from the pop-up menu.

pop wirebond heal shape

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Shapes* in the Find Filter, and choose the *Heal* option from the pop-up menu.

pop wirebond merge fingers

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Advanced – Create Merge Finger Shape* option from the pop-up menu.

pop wirebond move

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Move* option from the pop-up menu.

pop wirebond move guide

Refer to the [wirebond select](#) command. Run the `wirebond manage guide paths` command, choose *Lines* in the Find Filter, and choose the *Move* option from the pop-up menu.

pop wirebond move wires

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Bond wires* in the Find Filter, and choose the *Move* option from the pop-up menu.

pop wirebond net assign

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Shapes* in the Find Filter, and choose the *Assign Nets* option from the pop-up menu.

pop wirebond pause

The `pop wirebond pause` command is used while interactive placement such as adding or moving wire bonds. While paused, the tool does not process any cursor movements until the tool returns to the active state.

pop wirebond popul guide

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Lines* in the Find Filter, select the guide path, and choose the *Populate with Fingers* option from the pop-up menu.

pop wirebond recon

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Pins* in the Find Filter, and choose the *Reconnect* option from the pop-up menu.

pop wirebond settings

Refer to the [wirebond select](#) command

pop wirebond space

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Space Evenly* option from the pop-up menu.

pop wirebond split fingers

Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Advanced – Remove Merge Finger Shape* option from the pop-up menu.

pop wirebond split guide

Run the `wirebond select` command, choose *Lines* in the Find Filter, and choose the *Redistribute Fingers* option from the pop-up menu.

pop wirebond swap

Run the `wirebond select` command, choose *Fingers* in the Find Filter, and choose the *Swap* option from the pop-up menu.

pop wirebond swap wires

Refer to the [wirebond select](#) command. Run the `wirebond select` command, choose *Bond wires* in the Find Filter, and choose the *Swap* option from the pop-up menu.

power integrity

The `power integrity` command can be used to design, model, and analyze the power delivery system of your PCB design, which comprises many features and multiple functionality.

Preparing your Design to Satisfy Power Integrity Setup Requirements

To prepare your design to meet the power integrity setup requirements, follow these steps:

1. Type *power integrity* in the Command window and press Enter.
If your design has not been previously been set up for Power Integrity simulation and analysis, the Power Integrity Setup Wizard appears. Otherwise, the Power Integrity Design and Analysis dialog box displays.
2. With the Power Integrity Setup Wizard displayed, click the *Next* button to begin the setup procedure.
The Board Outline dialog box appears.
3. In the Board Outline dialog box, enter height and width values for your board outline, then click *Create Board Outline* to create a board outline from scratch. A representation of the specified outline appears in the upper-right corner of the dialog box.
– or –
Click *Import Outline* to import a board outline from another design. A file browser appears. Choose a design from the file window and click *Open*. A representation of the outline from that board appears in the upper-right corner of the dialog box. Complete step 2 or click *Next* to proceed to the Stack-up dialog box.
4. Optional Step:
Click *Edit Outline* and modify the board outline to suit your needs. A board outline confirmation window appears as the current board outline opens in the floorplanner view. In the confirmation window, click *Edit* and then click and drag the resize handles to change the size or the shape of the outline. In the confirmation window, click *OK* to accept the new outline. Click *Next* to proceed to the Stack-up dialog box.
5. In the Stack-up dialog box, check to see that a valid stack-up exists. If a stack-up does not exist or if it does not contain plane layers, a message appears in the upper-right corner of the dialog box. In the case of an existing stack-up, verify that the stack-up characteristics are suitable by viewing details in both the spreadsheet and the stack-up graphic. Complete step 4 or click *Next* to proceed to the DC-Net – Plane Association dialog box.
6. Optional Step:
Click *Edit stack-up* and modify the existing stack-up to suit your needs. The Edit Stack-up dialog box appears. Choose a stack-up layer from the listbox, then use the options in the Selected Layer area to edit the layer characteristics. Click either *Apply* or *OK* to accept the new changes. Click *Next* to proceed to the DC-Net – Plane Association dialog box.
– or –
Click *Import stack-up* to import a stack-up from another design. A File Browser appears. Choose a design from the file window and click *Open*. A representation of the stack-up appears in the dialog box. Click *Next* to proceed to the DC-Net – Plane Association dialog box.
7. In the DC-Net – Plane Association dialog box, assign a DC voltage to each plane shape. If a stack-up does not exist or if it does not contain plane layers, a message appears in the upper-right corner of the dialog box. In the case of an existing plane layers, verify that the DC voltage assignments are suitable by viewing details in both the spreadsheet and the stack-up graphic. Complete one or more of the optional steps below or click *Next* to proceed to the Power Pair Setup dialog box.
8. Optional Step:
Click *DC nets* and specify power pins and voltage levels. The Identify DC Nets dialog box appears. Click *Next* to proceed to the Power Pair Setup dialog box.
Because you can change the voltage directly in the wizard, you do not have to use this dialog box unless you are interested in reviewing all the nets that exist in the design as the wizard shows only those nets that have a VOLTAGE property attached to it.
9. Optional Step:
Click *Edit shape* and create, modify, move or delete a plane shape in the Floorplanner view. The Plane Outline dialog box appears. choose an operation, enter the shape data, click either *Apply* or *OK*. In the case of Edit or Move operations, click on the plane handles in the Floorplanner view to move or edit the plane shape. Click *Next* to proceed to the Power Pair Setup dialog box.
10. Optional Step:
Click *Edit nets* and create a new net or change the pins in an existing net. The Edit Nets dialog box appears.
Because you can change the net or create a new net directly in the wizard, you do not have to use this dialog box unless you are interested in changing the pins in a net.
11. In the Power Pair Setup dialog box, you match power and ground planes as a requirement for power distribution analysis. In the Planes area, choose two planes to be a matched pair, then click *Add* to add the plane pair to the Power Plane Pairs list box. Repeat as required until all plane pairs have been matched. Complete one or more of the optional steps below or click *Next* to proceed to the Library Setup dialog box.
It is permissible to share the same ground plane with one or more power rails (as separate plane pairs) however, only one plane pair can be analyzed at a time.
12. Optional Step:
Modify an existing power plane pair. Choose a different plane in the Planes area. Choose the power plane pair to modify from the listbox, then click *Modify*. The plane pair IDs change to reflect the newly chosen planes for the matched pair. Click *Next* to proceed to the Library Setup dialog box.
13. Optional Step:
Delete an existing power plane pair. Choose the power plane pair to be deleted from the Power Plane Pair area, then click *Delete*. The plane pair is

deleted and removed from the listbox. Click *Next* to proceed to the Library Setup dialog box.

14. In the Design Device setup dialog box, you select devices that are already in the board design for use by Power Integrity. (Selected devices appear in the Board directory of the Library Setup dialog box tree view window.) You can alter these to display those used solely for power integrity or for signal integrity. Devices already in use, by at least one plane pair, are in red. Devices in green are not currently in use. Click *Next* to proceed to the Library Format dialog box.
15. In the Library Format dialog box, you choose a method to add decoupling capacitors. You can select Project Library to add PTF-formatted decaps from the Component Browser or DCL Library to add 3rd party DCL-formatted decaps.

You cannot backannotate to your schematic the decoupling capacitors you are adding in this step. When you add decaps from the Component Browser, the design database is branded as HDL. Additional information on the injected properties that define model parameters in PTF-formatted decoupling capacitors are covered in Library Format in the *Allegro PCB PI User Guide*.

Click *Next* to proceed to the Library Setup dialog box.

16. In the Library Setup dialog box, you choose a decoupling capacitor library and choose decoupling capacitors for power plane pairs. Click the arrow at the end of the Power plane pair field and choose a power plane pair. In the library navigation pane, choose a capacitor library containing capacitors whose resonant frequencies cover your design's operating frequency band.
When searching for decoupling capacitor models, Allegro PCB PI option looks to the directories associated with the `DCL_PATH` environment variable.
17. Click on the label of a decoupling capacitor and verify, using the diagram, that its resonant frequency is acceptable for your design. To accept the capacitor for the power plane pair, click on its check box. Choose additional capacitors for the power plain pair as required. Complete step 14 or click *Finish* to exit the Setup Wizard once decoupling capacitors have been chosen for all power plane pairs in your design.

Optional Steps:

1. Click *Library*. The Signal Analysis Library Browser dialog box appears. You can use this dialog box to manage your decoupling capacitor libraries as well as modify a decoupling capacitor model.
2. Click *Component Browser*.
 - a. If the design has not previously been branded, you must select the HDL option in the Choice Selection popup.
 - b. Select a HDL design file (`.cpm`) of library parts. (These files are created by Project Manager during creation and setup of a project. See the *Allegro Design Entry HDL User Guide* for details.)
The Component Browser dialog box appears. You can use this dialog box to
 - Search/browse for parts
 - View details of parts, including symbols and footprint
 - Add/replace parts

preferences

This command is obsolete.

prepopup

With another command active, the `prepop-up` Scommand lets you change the position of a right mouse button pop-up menu by entering x and y coordinates. Although available in most tools, it is used primarily in SigExplorer.

pring wizard

The `pring wizard` command launches the Power/Ground Ring Wizard. This command lets you define and place one or more shapes in the form of a ring around the die. The Power/Ground Ring Wizard creates up to 12 rings (shapes) at a time. If you require more rings, you can run the Power/Ground Ring Wizard as many times as needed. This command displays a wizard in which you can specify the:

- Number of rings to generate
- The creation of the first ring as a die flag
If you create a die flag and the first ring is the same net as the flag, you can enter a negative distance to overlap the ring and die flag.
- Placement of the rings from a specified origin
 - Origination point
 - Distance from the edge of the die
 - Distance from the nearest die pin on each die side
- The reference designator of the die with which the rings will be used
- Distance between rings
- Width of each ring
- Corner types on each ring
 - Arc
 - Chamfer
 - Right-angle
- Radius of corner if arc (or Length if chamfer)
- Assigned netname for each ring
- Label for each ring

The rings are basic in nature. For other shape geometries, choose *Shape – Polygon* or *Shape – Compose/Decompose Shape* from the menu bar. For information on how to split rings, see [Creating a Set of Split Rings Around a Complex Wire Bond Die](#).

Depending on the options you select, the dialog boxes for the Power/Ground Ring Wizard change, representing how the rings will be created. You should verify the dialog box settings to ensure that you create the rings you intended.

Related Topics

- [Ring Count and Placement Dialog Box](#)
- [First Ring/Die Flag Definition Dialog Box](#)
- [Next Ring Definition Dialog Box](#)
- [Result Verification Dialog Box](#)
- [Creating a Set of Split Rings Around a Complex Wire Bond Die](#)

Power/Ground Ring Wizard Dialog Boxes

Access Using

- *Menu Path: Route – Power/Ground Ring Generator*
- *Menu Path: Generate – Power/Ground Ring Generator*

The dialog boxes and controls in this section step you through the wizard process. Each dialog box lets you set specific parameters. The controls common to each dialog box are:

<i>Next</i>	Moves you to the next step of the wizard process.
<i>Back</i>	Returns you to the previous step of the wizard process. (Not available for the Ring Count and Placement screen.)
<i>Cancel</i>	Closes the dialog box and exits you from the command.

Related Topics

- [First Ring/Die Flag Definition Dialog Box](#)
- [Next Ring Definition Dialog Box](#)
- [Result Verification Dialog Box](#)
- [Creating a Set of Split Rings Around a Complex Wire Bond Die](#)

Ring Count and Placement Dialog Box

This dialog box lets you specify:

- The number of rings you want to create
- If you want to create the first ring as a die flag
- How to place them
- Set the basic configuration

If the current design has no die component, you only can specify the number of rings you want, which are placed in reference to the origin specified on a subsequent dialog box. If the current design has a die, you can specify the number of rings and whether they are placed in reference to the origin, to the edge of the die, or to the nearest die pin.

Number of Rings	Options are between 1 and 12 rings Note: You can type a number exceeding 12 in this field, but the wizard defaults to 12 when you attempt to move to the next screen. Values less than 12 revert back to the previous value.
Place with respect to...	Indicates how to calculate the distances of the rings. The options are: <ul style="list-style-type: none"> • <i>Origin point</i> (a specified point in the design) • <i>Specified distance from edge of die</i> (relative to the die symbol's outline) • <i>Specified distance from nearest die pin</i> (relative to the die pin's edge).
Basic configuration	Lets you specify the distance from the inner ring to the origin point, the edge of the die, or the nearest die pin. The parameters that you set vary, depending on the method you select to place the rings.
Origin point	<p>Specified distance from edge of die</p> <ul style="list-style-type: none"> • <i>RefDes</i>: If there is more than one die in the package, use this to select the die to use as the reference to create the rings. • <i>Distance D1</i>: Specifies the distance of the inner edge of the first ring, relative to the die symbol's outline.
Specified distance from nearest die pin	<ul style="list-style-type: none"> • <i>RefDes</i>: If there is more than one die in the package, select the die to use as the reference to create the rings. • <i>Distance D1</i>: Specifies the distance of the inner edge of the first ring, relative to the nearest die pin's outline on each die side.
Create first ring as die flag	Lets you specify the first ring as a die flag. A die flag is a metal structure, which varies in shape, that is the contact point between the back of a wire bond die and the top of the substrate. It is used for adhesion and thermal transfer. If the die flag requires a cutout, for example part of the die cannot be tied to the assigned net, use the shape editing tools.
Create ring(s)/flag as static shapes, not dynamic	Lets you determine whether the tool creates the rings or flag static or dynamic shapes.
Group ring(s) and flag with die	If you enable this checkbox, the shapes created by the wizard are added to the die's IC group. When you modify the die, the rings and flag are also acted on in the same way.

Related Topics

- [pring wizard](#)
- [Next Ring Definition Dialog Box](#)
- [Result Verification Dialog Box](#)
- [Creating a Set of Split Rings Around a Complex Wire Bond Die](#)

First Ring/Die Flag Definition Dialog Box

This dialog box lets you specify information about the first ring or set the die flag parameters.

Ring 1 Parameters	
<i>Layer</i>	Selects the Conductor subclass on which to place the inner ring (shape). Click the browse button next to the <i>Layer</i> drop-down list to open the Die Flag Layers dialog box. Here you can click the check boxes to specify multiple layers on which to create rings.
<i>Constraints for layer</i>	Displays the chosen constraint values for reference.
<i>Corner Type</i>	Displays from the drop-down menu a corner type for your rings. Options are <i>Arc</i> , <i>Right Angle</i> , and <i>Chamfer</i> .
<i>Shape-shape spacing</i>	Displays the shape-to-shape spacing value established for the chosen layer.
<i>Radius R1/Length</i>	Defines the radius of the inside corner of the first ring (inner edge) or the distance from the die corner to the inner edge of the chamfer if your corner type is <i>Chamfer</i> . The larger the number, the more oval the ring becomes. The smaller the number, the more rectangular the ring becomes. The radius/length must be less than one-half the smaller of the width or height. This parameter is not applicable to right-angle corners.
<i>Minimum line width</i>	Displays the minimum line width for the chosen layer.
<i>Width W1</i>	Defines the width of the inner ring.
<i>Optional Information</i>	
<i>Create soldermask opening</i>	Lets you generate a soldermask shape based on the power and ground rings and die flags as the layout tool creates them in the ring generator.
<i>Clearance</i>	A positive value creates a soldermask shape that is larger than the actual ring. A negative value creates an exposure smaller than the actual ring boundary.
<i>Label</i>	Defines a text string as a label for the inner ring. This attaches the <i>LABEL</i> property, equal to the value entered. You can use this information in the Package Report feature.
<i>Net</i>	Defines the net name that you want to associate with the inner ring (shape). Type in the net name or select the <i>Browse</i> button to obtain a list of net names in the design. If the net name you typed does not exist in the design, then the wizard creates a new net with that name.
<i>Next</i>	Creates the ring in the drawing area, as indicated by the status message 'Ring 1 created' at the console command prompt, then moves you to the next step of the wizard process.
<i>Back</i>	Returns you to the previous step of the wizard process.

If you checked the *Create first ring as die flag* box in the Ring Count and Placement dialog box, these parameters appear.

Die Flag Parameters	
<i>Layer</i>	Selects the Conductor subclass on which to place the inner ring (shape).
<i>Constraints for layer</i>	Displays the chosen constraint values for reference.
<i>Corner Type</i>	Displays from the drop-down menu a corner type for your rings. Options are <i>Arc</i> , <i>Right Angle</i> , and <i>Chamfer</i> .
<i>Shape-shape spacing</i>	Displays the shape-to-shape spacing value established for the chosen layer.

P Commands

P Commands--pring wizard

<i>Radius R1/Length</i>	Defines the radius of the inside corner of the first ring (inner edge) or the distance from the die corner to the inner edge of the chamfer if your corner type is <i>Chamfer</i> . The larger the number, the more oval the ring becomes. The smaller the number, the more rectangular the ring becomes. The radius/length must be less than one-half the smaller of the width or height. This parameter is not applicable to right-angle corners.
<i>Minimum line width</i>	Displays the minimum line width for the chosen layer.
<i>Label</i>	Defines a text string as a label for the inner ring. This attaches the <i>LABEL</i> property, equal to the value entered. You can use this information in the Package Report feature.
<i>Net</i>	Defines the net name that you want to associate with the inner ring (shape). Type in the net name or click <i>Browse</i> to obtain a list of net names in the design. If the net name you typed does not exist in the design, then the wizard creates a new net with that name.
<i>Next</i>	Creates the die flag in the drawing area, as indicated by the status message ‘Die Flag created’ at the command window prompt, then moves you to the next step of the wizard process.
<i>Back</i>	Returns you to the previous step of the wizard process.

Related Topics

- [pring wizard](#)
- [Power/Ground Ring Wizard Dialog Boxes](#)
- [Result Verification Dialog Box](#)
- [Creating a Set of Split Rings Around a Complex Wire Bond Die](#)

Next Ring Definition Dialog Box

Use this dialog box to specify information about succeeding rings. The wizard creates each additional ring concentrically around the preceding ring.

<i>Ring (x) Parameters</i>	
<i>Layer</i>	Selects the Conductor subclass on which to place the inner ring (shape).
<i>Constraints for layer</i>	Displays the chosen constraint values for reference.
<i>Corner Type</i>	Displays from the drop-down menu a corner type for your rings. Options are <i>Arc</i> , <i>Right Angle</i> , and <i>Chamfer</i> .
<i>Shape- shape spacing</i>	Displays the shape-to-shape spacing value established for the chosen layer
<i>Radius Rx/Length</i>	Defines the radius of the inside corner of the ring (inner edge) or the distance from the die corner to the inner edge of the chamfer if your corner type is <i>Chamfer</i> . The larger the number, the more oval the ring becomes. The smaller the number, the more rectangular the ring becomes. The radius/length must be less than one-half the smaller of the width or height. This parameter is not applicable to right-angle corners.
<i>Minimum line width</i>	Displays the minimum line width for the chosen layer.
<i>Width Wx</i>	Defines the width of the inner ring.
<i>Distance Dx</i>	Defines the distance between rings. You can set a negative value for the first ring to overlap the ring and the die flag if the flag and the flag are the same net.
<i>Optional Information</i>	
<i>Label</i>	Defines a text string as a label for the inner ring. This attaches the <i>LABEL</i> property, equal to the value entered. You can use this information in the Package Report feature.
<i>Net</i>	Defines the net name that you want to associate with the inner ring (shape). Type in the net name or click <i>Browse</i> to obtain a list of net names in the design. If the net name you typed does not exist in the design, then the wizard creates a new net with that name.
<i>Next</i>	Creates the ring in the drawing area, as indicated by the status message 'Ring x created' in the command console of your user interface. The Next Ring Definition dialog box appears again for the number of rings you specified in the Ring Count and Placement dialog box, letting you set the parameters for the next ring to be created. The default settings for the current ring are based on the settings established for the previous one.
<i>Back</i>	Returns you to the previous step of the wizard process.

Related Topics

- [pring wizard](#)
- [Power/Ground Ring Wizard Dialog Boxes](#)
- [Ring Count and Placement Dialog Box](#)
- [Creating a Set of Split Rings Around a Complex Wire Bond Die](#)

Result Verification Dialog Box

This dialog box lets you accept the rings as viewed in the layout and generate the power and ground rings by writing them to the database. It then indicates how many new DRCs might be created by the rings.

Up to this point, power and ground rings have been placed in the drawing but have not been written to the database. You can modify the rings by clicking the *Back* button.

- ⓘ To change a power or ground ring already written to the database, you must delete the existing ring and re-create it with the Power and Ground Ring Wizard.

<i>Back</i>	Click this button to return to the previous dialog box to make changes.
<i>Finish</i>	Click this button to accept the rings as generated in the layout.
<i>Cancel</i>	Click this button to abort the generation of the rings. The rings, as viewed in the layout, are removed.
<i>Help</i>	Click this button to get access to the on-line help files.

⚠ The rings created are simple shapes with a void. The size of the void is the size of the shape minus the width of the ring. If you need to edit the rings, select *Shape–Manual Void–Circular* (`shape void circle`) from the menu. Be careful because the void may be removed, leaving a solid filled shape and not a ring as intended.

Related Topics

- [pring wizard](#)
- [Power/Ground Ring Wizard Dialog Boxes](#)
- [Ring Count and Placement Dialog Box](#)
- [First Ring/Die Flag Definition Dialog Box](#)

Creating a Set of Split Rings Around a Complex Wire Bond Die

The following is the recommended flow for creating split rings in a package design.

Goal	Using the Power and Ground Ring Wizard and dynamic shape functionality, create a set of split rings for use around a complex wire bond die in a package.
Conditions	You have an existing package design containing the IC around which you wish to create the split ring pattern. This flow assumes you already know where the major ring breaks (where the rings change nets) will be.

To create a set of split rings around a complex wire bond die, perform these steps:

1. Open the design.
2. Create a set of whole rings which you will cut up to form your split ring pattern.
 - a. Choose *Route – Power/Ground Ring Generator* ([pring wizard](#) command).
 - b. In the wizard, specify the number of whole rings you want to create.
 - c. Select the reference designator for the die around which you want the rings.
 - d. For each ring, specify the width, corner type, and radius. Specify also the gap between each ring (or, for the first ring, from the edge of the die). Summary information on the **Result Verification** page of the wizard informs you if DRCs were created as a result of ring creation.
 - e. When you are satisfied with the results of the ring generation, click *Finish* to complete the wizard process and instantiate the rings into the design.
3. Correct any DRCs that occurred following creation of the rings.
4. Ensure that the shape suppression value is less than the minimum ring piece size you will create.
 - a. Choose *Shape – Global Dynamic Params* ([shape global param](#) command) *from the menu bar*.
 - b. In the **Global Shape Parameters** dialog box, bring forward the Void controls tab and set the *Suppress shapes less than* field to the required minimum value. If you are unsure what this value should be, enter 0 for now. This prevents any shapes in your design being suppressed.

 Record the original suppression value so you can reset it toward the end of this process.
5. Use the line drawing commands of your choice (*Add – Line*, *Add – Circle*, *Add – Arc*, and so on) to create lines on the same layer as the ring shapes. It is recommended that you set your shape-to-line DRC spacing to be half the desired width of the gap between ring sections and set your line width to 0. This ensures the exact gap you require. Record the original shape-to-line DRC value so you can reset it toward the end of this process.

 The lines must cut through the shapes completely in order to split the ring section into two distinct pieces.
6. Convert each ring into a static shape.
 - a. Run *Shape – Change Shape Type* ([shape change type](#) command).
 - b. In the *Options* panel, set the *Shape Fill* type to *static solid*.
 - c. Select the ring segment you want to convert.

A warning message informs you that you will lose the original shape boundary, parameter settings, and defined voids on the BOUNDARY class.
 - d. Click *Yes* to acknowledge the warning.
 - e. Repeat steps **a** through **d** for each ring you want to convert.
 - f. Choose *Done* from the right-button pop-up to complete the command and return to the idle state.
7. Delete the marker lines you used to split the rings.
8. Reset the shape suppression and DRC values that you changed in steps 4 and 5 to their original settings.
9. Assign each ring segment to a net of your choice with *Logic–Assign Net* ([assign net](#) command).
10. Reconvert each ring segment into a dynamic shape.
 - a. Run *Shape – Change Shape Type* ([shape change type](#) command).
 - b. In the *Options* panel, set the *Shape Fill* type to *dynamic copper*.
 - c. Select the ring you want to convert.
 - d. Repeat steps **a** through **c** for each ring you want to convert.

e. Choose *Done* from the right-button pop-up to complete the command and return to the idle state.

The system can now dynamically void the ring segments to add or remove clearances around vias, routing, and other design elements.

Related Topics

- [pring wizard](#)
- [Power/Ground Ring Wizard Dialog Boxes](#)
- [Ring Count and Placement Dialog Box](#)
- [First Ring/Die Flag Definition Dialog Box](#)
- [Next Ring Definition Dialog Box](#)

print

The `print` command lets you print a file from the command console of your user interface.

Syntax

```
print <file_name> <printer_name>
```

Printing a File from the Command Line

To print a file from the command line:

1. Type `print` and the name of the file you want printed, followed by the name of the printer.
You must type in a full path name to the file if it is not in your current working directory.
2. Press Enter/Return to print the file.

printform

This command is designed for internal Cadence development.

prmed

The `prmed` command displays the Design Parameter Editor, which centralizes commonly used parameters. In the Design Parameter Editor, select tabs for *Display*, *Design*, *Text*, *Shapes*, *Flow Planning*, *Route*, and *Mfg Applications* and edit the specific parameters in each of these categories.

You may also access the Design Parameter Editor by right-clicking anywhere in the design canvas to display the *Quick Utilities* pop-up menu from which you may choose *Design Parameters*.

 With Allegro Free Viewer and Allegro Viewer Plus, only the *Display*, *Design*, and *Text* tabs are available.

Related Topics

- [Changing Display Parameters](#)
- [Changing Design Parameters](#)
- [Changing Text Parameters](#)
- [Changing Shapes Parameters](#)
- [Changing Flow Planning Parameters](#)
- [Changing Route Parameters](#)
- [Changing Manufacturing Applications Parameters](#)

Design Parameter Editor Dialog Box

Access Using

- *Menu Path:* Setup – Design Parameters



- *Toolbar Icon:*

Use this dialog box to edit the parameters you want to apply to the design. Hover your cursor over each parameter and a description of its functionality displays in the Parameter Description area of the dialog box. The parameters are grouped under the following tabs:

<i>Display</i>	Lists parameters that control the display of the design.
<i>Design</i>	Lists parameters that control the drawing size and extents, line lock and text controls.
<i>Text</i>	Lists parameters that control the display of text.
<i>Shapes</i>	Lists parameters that control how shapes are defined.
<i>Flow Planning</i>	Lists global parameters that control interconnect flow planning. (The <i>Flow Planning</i> tab is only available with Allegro PCB Designer and above.)
<i>Route</i>	Lists parameters that control etch editing.
<i>Mfg Applications</i>	Lists parameters that control the processing of manufacturing applications, such as testability, thieving and silkscreen.

Related Topics

- [Changing Design Parameters](#)
- [Changing Text Parameters](#)
- [Changing Shapes Parameters](#)
- [Changing Flow Planning Parameters](#)
- [Changing Route Parameters](#)
- [Changing Manufacturing Applications Parameters](#)

Changing Display Parameters

 Via span labels are disabled in Allegro PCB Design L, OrCAD, and Allegro PCB Performance option L.

To change the display parameters, perform these steps:

1. Click the *Display* tab.
 When you hover your cursor over a parameter, the full description of that parameter is shown in the *Parameter description* group box.
2. In the *Display* group box, enter new values for the parameters you want to change.
3. In the *Enhanced Display Modes* group box, enable or disable the check boxes to either display or hide particular objects.
4. In the *Display net names* group box, enable *clines*, *shapes* or *pins* to display the embedded net names.

 To display net names enable OpenGL and set transparency to less than *Solid* level in the *Color Dialog* box.

1. In the *Grids* group box, enable *Grids on* to display the grids. Click *Setup Grids* to display the Define Grid dialog box and specify the grid spacings you want to use for different layers.
2. Click *Apply* to apply the changes.

Related Topics

- [prmed](#)
- [Changing Text Parameters](#)
- [Changing Shapes Parameters](#)
- [Changing Flow Planning Parameters](#)
- [Changing Route Parameters](#)
- [Changing Manufacturing Applications Parameters](#)

Changing Design Parameters

To change the design parameters, perform these steps:

1. Click the *Design* tab.

 When you hover your cursor over a parameter, the full description of that parameter is shown in the *Parameter description* group box.

2. In the *Size*, *Extents*, *Move Origin*, *Symbol* and *Drawing Type* group boxes, enter new values for the general design parameters you want to change.
3. In the *Line Lock* group box, enter new values for *Lock direction*, *Lock mode* and *Minimum radius*.
4. Click *Apply* to apply the changes.

Related Topics

- [prmed](#)
- [Design Parameter Editor Dialog Box](#)
- [Changing Shapes Parameters](#)
- [Changing Flow Planning Parameters](#)
- [Changing Route Parameters](#)
- [Changing Manufacturing Applications Parameters](#)

Changing Text Parameters

To change the text parameters, perform these steps:

1. Click the *Text* tab.

 When you hover your cursor over a parameter, the full description of that parameter is shown in the *Parameter description* group box.

2. In the *Size* group box, enter new values for *Justification*, *Parameter block* and *Text marker size*.
3. Click *Setup Text Sizes* to display the Text Setup dialog box and specify new parameters for the text blocks.
4. Click *Apply* to apply the changes.

Related Topics

- [prmed](#)
- [Design Parameter Editor Dialog Box](#)
- [Changing Display Parameters](#)
- [Changing Flow Planning Parameters](#)
- [Changing Route Parameters](#)
- [Changing Manufacturing Applications Parameters](#)

Changing Shapes Parameters

To change the shapes parameters, perform these steps:

1. Click the *Shapes* tab.
2. Click *Edit global dynamic shape parameters* to display the Global Dynamic Shape Parameters dialog box. Make changes to the parameters as needed. You can also run *Shape – Global Dynamic Params* (`shape global param`) to access the Global Dynamic Shape Parameters dialog box.
3. Click *Edit static shape parameters* to display the Static Shape Parameters dialog box. Make changes to the parameters as needed (similar to selecting a shape, then entering the command `shape param`).
4. Click *Edit split plane parameters* to display the Split Plane Params dialog box. Select a new style (similar to entering the command `split plane param`).

 Any changes you make to split plane parameters will only apply if you have positive planes defined in your design.

Related Topics

- [prmed](#)
- [Design Parameter Editor Dialog Box](#)
- [Changing Display Parameters](#)
- [Changing Design Parameters](#)
- [Changing Route Parameters](#)
- [Changing Manufacturing Applications Parameters](#)

Changing Flow Planning Parameters

 The *Flow Planning* tab is only available with Allegro PCB Designer and above.

To change the flow planning parameters, perform these steps:

1. Click the *Flow Planning* tab.
 -  When you hover your cursor over a parameter, the full description of that parameter is shown in the *Parameter description* group box.
2. In the *Category* group box, click the folder for the parameters (*General*, *Default Bundle Properties*, *Auto Bundle*, *Plan Invocation*, *Routing Controls*, *Layer Controls*) you want to modify.
The parameters associated with that folder appear in the right-hand pane.
3. Choose the value for the specific parameter you want to modify from the corresponding drop-down list box, or enter a value. For some parameters that do not have values, enable the corresponding radio button or check box to activate the parameters.
 -  For *Layer Controls*, click the check boxes to enable or disable the layers under the *Random Logic* column, then choose the *Direction* for each layer from the drop-down list boxes.
4. Click *Reset* to restore the displayed general parameters you are modifying. *Reset* restores the parameters to either the most recent previous setting, or to the default setting if no prior change has been made.
5. Click *Apply* to apply the changes.

Related Topics

- [prmed](#)
- [Design Parameter Editor Dialog Box](#)
- [Changing Display Parameters](#)
- [Changing Design Parameters](#)
- [Changing Text Parameters](#)
- [Changing Manufacturing Applications Parameters](#)

Changing Route Parameters

To change the route parameters, perform these steps:

1. Click the *Route* tab.

 When you hover your cursor over a parameter, the full description of that parameter is shown in the *Parameter description* group box.
2. In the *Commands* group box, click the folder for the route command (*Add Connect*, *Delay Tune*, *Edit Vertex*, *Slide*, *Auto-I. Phase Tune*, *Auto-I. Delay Tune*, *Timing Vision*, *Auto-I. Convert Corner*, *Gloss*, or *Create Fanout*) you want to modify.
The parameters associated with that command appear in the right-hand pane.

 The *Gloss* command is not available when you are working on a `.dpf` (design partition file). In the Symbol Editor, only the *Add Connect* and *Slide* commands are available.
3. Choose the *Value* for the specific *Parameter* you want to modify from the corresponding drop-down list box, or enter a value. For some parameters that do not have values, enable the corresponding radio button or check box to activate the parameters.

 For the *Gloss* command, click *View glossing applications* to display the Glossing Controller dialog box, then click on different applications and edit glossing parameters in the associated dialog box for each application. For the *Create Fanout* command, click *Create Fanout Parameters* to display the Create Fanout dialog box, then specify the parameters for the new fanout.
4. Enable the *Show* check box to specify whether the corresponding parameter should appear in the *Options* panel of the Control Panel. If you disable the *Show* check box, the parameter will not appear in the *Options* panel. By default, the *Show* check box is enabled for every parameter. Alternately, click the *Show* button to enable or disable all of the check boxes at one time.
5. Click *Reset* to restore the displayed general parameters for the particular etch edit command you are modifying. *Reset* restores the parameters to either the most recent previous setting, or to the default setting if no prior change has been made.
6. Click *Apply* to apply the changes.

Related Topics

- [prmed](#)
- [Design Parameter Editor Dialog Box](#)
- [Changing Display Parameters](#)
- [Changing Design Parameters](#)
- [Changing Text Parameters](#)
- [Changing Shapes Parameters](#)

Changing Manufacturing Applications Parameters

To change the manufacturing applications parameters, perform these steps:

1. Click the *Mfg Applications* tab.
2. Click *Edit testprep parameters* to access the Testprep Parameters dialog box, also available by running *Manufacture – Testprep – Parameters* (`testprep prmed` command).
3. Click *Edit thieving parameters* to access the Thieving Parameters dialog box. These parameters are also available in the *Options* panel of the Control Panel by running *Manufacture – Thieving* (`thieving` command).
4. Click *Edit silkscreen parameters* to access the Auto Silkscreen dialog box, also available by running *Manufacture – Silkscreen* (`silkscreen param` command).
5. Click *Edit drafting parameters* to access the Drafting dialog box, available by running *Manufacture – Dimension/Draft – Parameters* (`draft param` command).

Related Topics

- [prmed](#)
- [Design Parameter Editor Dialog Box](#)
- [Changing Display Parameters](#)
- [Changing Design Parameters](#)
- [Changing Text Parameters](#)
- [Changing Shapes Parameters](#)
- [Changing Flow Planning Parameters](#)

prompt

The `prompt` command extends the capabilities of your Cadence tool or lets you evaluate problems related to scripting. For example, typing:

```
prompt script
```

at the command window prompt displays the text in a window. The script is paused until you close the window.

property edit

The `property edit` command assigns properties to design elements, or changes or deletes existing property values.

To quickly add or remove properties and edit override values on a net without invoking Constraint Manager, you can also launch the Edit Property dialog box using:

- *Setup – Constraints – Physical Net Overrides*, which displays the physical properties relevant for that net in the Edit Property dialog box.
- *Setup – Constraints – Spacing Net Overrides*, which displays the spacing properties relevant for that net in the Edit Property dialog box.

This command functions in a pre-selection use model, in which you choose an element first, then right-click and execute the command. Elements ineligible for use with the command generate a warning and are ignored. Valid elements:

- Groups
- Components
- Symbols
- Nets
- Pins
- Vias
- Clines
- Lines
- Shapes
- Voids
- Figures
- Rat Ts

Related Topics

- [Assigning a Property to a Design Element](#)
- [Editing a Property on a Design Element](#)
- [Deleting a Property from a Design Element](#)
- [Attaching a Property to a Room Boundary](#)
- [Creating an Inherited Property](#)

Edit Property Dialog Box

Access Using

- *Menu Path: Edit – Properties*
- *Menu Path: Setup – Constraints – Physical Net Overrides*
- *Menu Path: Setup – Constraints – Spacing Net Overrides*
- *Menu Path: Setup – Edit Properties (Allegro PCB SI)*

Use this dialog box to choose the properties you want to add, edit, or delete. When you choose a property from the *Available Properties* list, or type in the property name, it appears in the panel on the right-hand side of the dialog box.

<i>Available Properties</i>	Lists all of the properties that are attached to the chosen elements.
<i>Name</i>	Lets you type the name of the property you want to delete or change instead of choosing it from the Available list.
<i>Delete</i>	Marks the chosen property for deletion from the chosen set of elements.
<i>Property</i>	Identifies the property that you are adding, deleting, or modifying.
<i>Value</i>	Lets you enter an alphanumeric character string as a value or to turn the property into an enabling switch by entering <code>TRUE</code> to assign the property to an element. If you leave this field empty, the property is not attached to any element when you click <i>Apply</i> . If you defined a user-defined property, you can store a web link as the value of the property. For additional information, see <i>Creating Design Rules</i> in the user guide.
<i>Reset</i>	Resets the dialog box, which clears the right-hand panel.
<i>Apply</i>	Adds or deletes marked properties on the chosen list to or from the chosen layout elements.
<i>Show</i>	Displays instances of the chosen property. Display options: <i>Sort by Element</i> : displays the Show Properties dialog box. <i>Sort by Prop</i> : displays the Show Properties dialog box. <i>Inherited Properties</i> : displays the Show Inherited Properties dialog box. <i>Turn off Show</i> : closes the Show Properties dialog box.

Show Properties Dialog Box

Lists all elements that you chose, along with their attached properties and values.

Related Topics

- [Editing a Property on a Design Element](#)
- [Deleting a Property from a Design Element](#)
- [Attaching a Property to a Room Boundary](#)
- [Creating an Inherited Property](#)

Assigning a Property to a Design Element

To assign a property to a design element:

1. Hover your cursor over an element or window select to choose a group of elements. The tool highlights the element and a data tip identifies its name.
2. Right-click and choose *Property Edit* from the pop-up menu.
3. The Edit Property dialog box appears.
4. In the Edit Property dialog box, choose a property from the list of Available Properties or enter a property name into the *Name* text field. (You can enter the property name in uppercase or lowercase characters.)
The property that you choose appears in the right side of the dialog box. You can choose any number of properties.
5. Specify a property value.
The type of value (boolean, integer, string, pre-defined) varies depending on the chosen property.
6. Click *Apply*.
The Show Properties window updates to display the current property values on the elements.
7. Click *OK* to close the Edit Property dialog box.

Related Topics

- [property edit](#)
- [Deleting a Property from a Design Element](#)
- [Attaching a Property to a Room Boundary](#)
- [Creating an Inherited Property](#)

Editing a Property on a Design Element

You can edit the value of existing properties in your design by following these steps:

1. Hover your cursor over an element or window select to choose a group of elements. The tool highlights the element and a data tip identifies its name.
2. Right-click and choose *Property Edit* from the pop-up menu.
3. The Edit Property dialog box appears.
The *Edit Properties* and *Show Properties* windows appear. The *Show Properties* window lists properties attached to the design element.
4. In the Edit – Property dialog box, click on a property from the list of *Available Properties* or enter a property name into the *Name* text field. (You can enter the property name in uppercase or lowercase.)
The property that you choose appears in the right side of the dialog box. You can choose any number of properties.
5. Edit the value type, depending on the chosen property (boolean, integer, string, pre-defined).
6. Click *Apply*.
The Show Properties window updates to display the current property values on the elements.
7. Click *OK* to close the Edit Property dialog box.

Related Topics

- [property edit](#)
- [Edit Property Dialog Box](#)
- [Attaching a Property to a Room Boundary](#)
- [Creating an Inherited Property](#)

Deleting a Property from a Design Element

You can remove a property from a chosen design element by performing these steps:

1. Hover your cursor over an element or window select to choose a group of elements. The tool highlights the element and a data tip identifies its name.
2. Right-click and choose *Property Edit* from the pop-up menu.
3. The Edit Property dialog box appears.
The *Edit Properties* and *Show Properties* windows appear. The *Show Properties* window lists properties attached to the design element.
4. In the Edit – Property dialog box, click on the property from the list of *Available Properties* or enter a property name into the *Name* textfield. (You can enter the property name in uppercase or lowercase.)
The property that you choose appears in the right side of the dialog box. You can choose any number of properties.
5. Click Delete to the left of the property name.
6. Click *Apply*.
The Show Properties window updates to display the current property values on the elements.
7. Click *OK* to close the Edit – Property dialog box.

Related Topics

- [property edit](#)
- [Edit Property Dialog Box](#)
- [Assigning a Property to a Design Element](#)
- [Creating an Inherited Property](#)

Attaching a Property to a Room Boundary

To attach a property to a room:

1. Hover your cursor over a shape. The tool highlights the element and a data tip identifies its name.
2. Right-click and choose *Property Edit* from the pop-up menu.
The Edit Property dialog box appears.
3. Choose the room boundary outline.
The Show Properties and Edit Property dialog boxes appear.
4. In the Property form under Available Properties, select ROOM_TYPE, or in the box next to Name type: ROOM_TYPE.
The right side of the form reflects the property.
5. In the Value box, type HARD and click *Apply*.
The Show Properties dialog box reflects the information you entered.
6. Click *Done* to close the dialog boxes.
7. Choose *Done* from the pop-up menu.

Related Topics

- [property edit](#)
- [Edit Property Dialog Box](#)
- [Assigning a Property to a Design Element](#)
- [Editing a Property on a Design Element](#)

Creating an Inherited Property

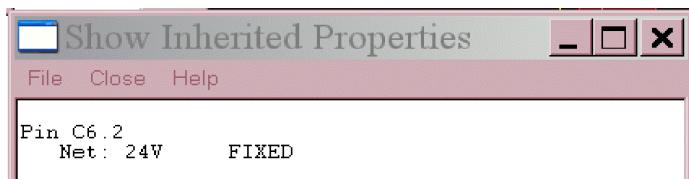
You create an inherited property with the `property edit` command in the same way that you attach any property to an element. You can also use the `fix` command to add the `FIXED` property.

Follow these steps to create an inherited property:

1. Hover your cursor over an element—for example, `net 24V`. The tool highlights the element and a data tip identifies its name.
2. Right-click and choose *Property Edit* from the pop-up menu.
3. The Edit Property dialog box appears.
4. Choose *FIXED* from the *Available Properties* list in the Edit Property dialog box.
The property displays on the right side of the dialog box.
5. Click *Apply* to attach the property to the element—in this example, `24V`.
6. Choose a child element (for example, `pin C6.2`) of the parent (`net 24V`).
7. In the Edit Property dialog box, click *Show* and then choose *Inherited Properties*.

The Show Inherited Properties dialog box displays the child element (the pin) and the status (*FIXED*) of its parent (the net), as follows.

Figure: Child Pin and Inherited Property from Parent Net Element



Related Topics

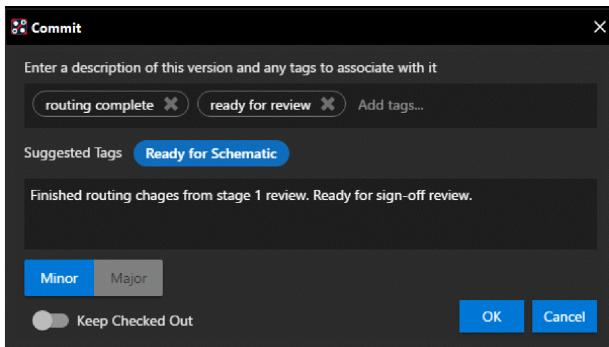
- [property edit](#)
- [Edit Property Dialog Box](#)
- [Assigning a Property to a Design Element](#)
- [Editing a Property on a Design Element](#)
- [Deleting a Property from a Design Element](#)

pulse_commit

The `pulse_commit` command uploads a copy of the current design to the version control database. You can add tags and comments to identify the version of the design being created. The commit action can be marked as a *Major* or *Minor*, which controls how the design version is incremented.

A `Ready for Schematic` tag can be used to synchronize the design for which the schematic is created in Allegro System Capture. For more information, refer to *Working with Pulse-Linked Layout and Schematic Designs*.

In the Enterprise mode, an additional option *Keep Checked Out* is available to keep the design checked out without unlocking the design after commit is successful.



⚠ If you have both read and write permissions for the design, the commit command can also be accessible through the right-click option on the Pulse icon in the status bar of the layout editor.

For more information on Pulse, see *Allegro Pulse Configuration Flow* in the *Allegro Pulse Configuration Guide*.

Access Using

- *Menu Path: File – Commit*

Examples

1. Opens the Commit dialog.

```
pulse_commit
```

2. Saves the design as major update.

```
pulse_commit major
```

3. Initiates a minor commit with comment string "fixes".

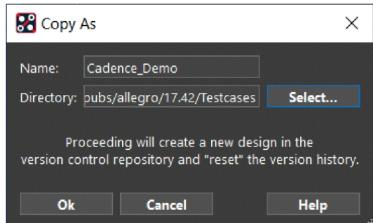
```
pulse_commit minor "fixes"
```

4. Initiates a major commit with comment string "fixes" and tags "tag1" and "tag2".

```
pulse_commit major "fixes" "tag1 tag2"
```

pulse_copy_as

The `pulse_copy_as` command saves a copy of the current design managed by Pulse to a new location and resets its version history.



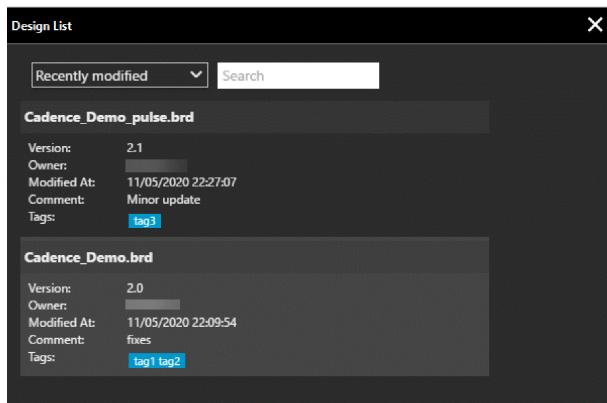
For more information on Pulse, see *Allegro Pulse Configuration Flow* in the *Allegro Pulse Configuration Guide*.

Access Using

- *Menu Path: File – Copy As*

pulse_designs

The `pulse_designs` command displays a list of shared designs that are managed by the Pulse platform and are accessible to you. You can choose from the designs that have been shared with you, or that you own. The selected design, if found on the local disk, opens in the layout editor. Otherwise, if the design is not found locally, the latest version is downloaded from Pulse and opened in the layout editor.



For more information on Pulse, see *Allegro Pulse Configuration Flow* in the *Allegro Pulse Configuration Guide*.

Access Using

- *Menu Path: File – Open From Pulse*

pulse_getrevision

The `pulse_getrevision` command opens a specific version in the layout editor. You can also save the copy of current version of the design as a backup. The revert option is also accessible through the right-click menu on the Pulse icon in the status bar of the layout editor.

Examples

1. Opens design version 1.3.

```
pulse_getrevision 1.3
```

2. Opens current design version.

```
pulse_getrevision current
```

This command reverts all local changes.

3. Opens the most recent version of the design.

```
pulse_getrevision latest
```

pulse_lock

The `pulse_lock` command locks the design and prevents other clients from committing or exporting design data. When successful the design acquire a lock from the Pulse and all other clients become read only. They can only edit the design, but cannot commit or save the locked design. The Pulse icon in the status bar of the layout editor displays a locked icon ().

 This command is available only in the Enterprise mode.

The locking option is also accessible through the right-click menu on the Pulse icon in the status bar of the layout editor.

pulse_reuse_module_mgr

The `pulse_pulse_reuse_module_mgr` command opens the Pulse Module Manager dialog box for managing modules placed in a top-level layout design. This dialog box provides the synchronization status of each module and options to update the managed modules.

Pulse Module Manager Dialog Box

Access Using:

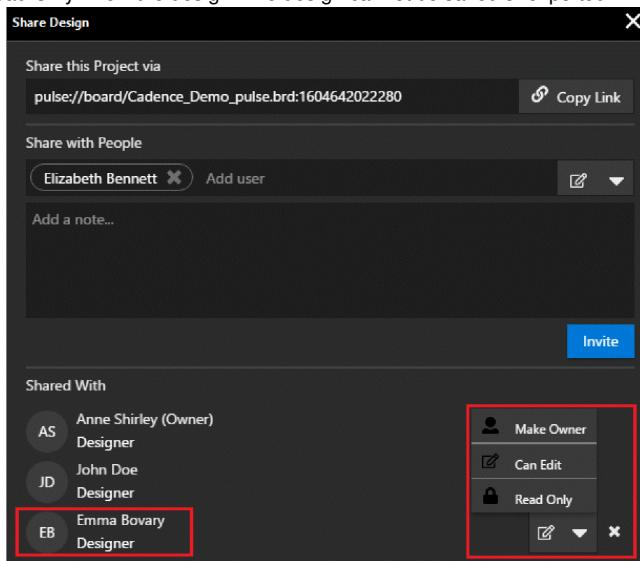
- Menu Path: Place – Pulse Module Manager

Name	Lists both placed and unplaced modules referenced by a project
Placed Instances	Displays the number of module instances placed in the top-level layout
Module Version	Displays version of the module in the top-level layout
Update Available	Indicates and displays the updated version of a module
Logic State	Indicates inconsistencies between the placed module logic and top-level layout logic
Lock State	Displays the check out state of the module. A module is locked if changes are being made to its logic
Module Netlist Status	Indicates the status of a module netlist. If an updated netlist is available to be imported in the module design, the status is displayed as Update Available.
Version Control	Opens Version control tree for the selected module
Open for Editing	Opens the selected module for editing
Import Latest	Refreshes all managed module instances to the latest

pulse_share

The `pulse_share` command is used to share the design with others. Before sharing the designs, you can modify permissions for each user individually. The design can be shared in the following three modes:

- Make Owner: Force unlocks the design and change permissions for other users.
- Can Edit: Edits and saves the version of the design (Commit) to Pulse server.
- Read Only: View the design. The design cannot be saved or exported.



⚠ This command is available only in the Enterprise mode.

For more information on Pulse, see *Allegro Pulse Configuration Flow* in the *Allegro Pulse Configuration Guide*.

Access Using

- Toolbar Icon: 

pulse_unlock

The `pulse_unlock` command unlocks the design. This command can be used by the lock owner to remove the lock. The Pulse icon in the status bar of the layout editor displays an eye icon showing that the design is available for locking.



⚠ This command is available only in the Enterprise mode.

You can unlock the design through the right-click option on the Pulse icon in the status bar of the layout editor.

Examples

1. Release the lock.

```
pulse_unlock
```

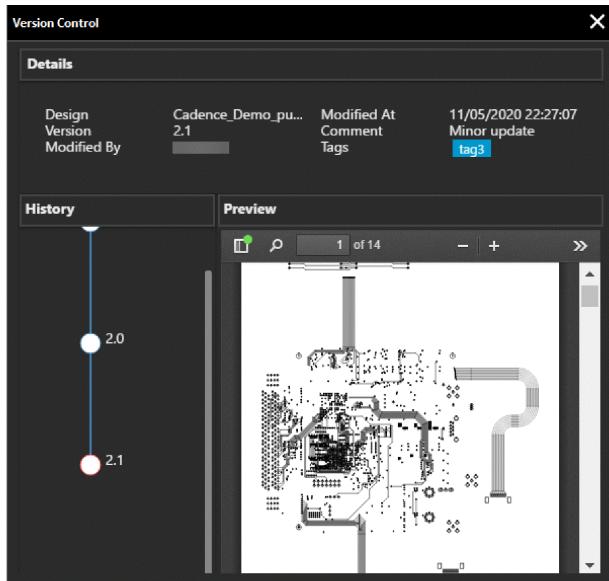
2. Prompts if design owner to forcibly remove the lock.

```
pulse_unlock -f
```

pulse_version_control

The `pulse_version_control` command displays the version history of the design. Selecting a node in the *History* pane displays a preview of the design version on the right and comments and tags in the *Details* section.

You can select and review the design versions without impacting the version of the design loaded in the layout editor. To load a previous version in the layout editor, right-click on a version node and choose *Open Version*.



The command can also be accessible through the right-click option on the Pulse icon in the status bar of the layout editor.

For more information on Pulse, see [Allegro Pulse Configuration Flow](#) in the [Allegro Pulse Configuration Guide](#).

Access Using

- *Menu Path: File – Version Control*

purge unplaced comps

The `purge unplaced comps` command lets you remove unplaced components left in your design database when you remove or replace design objects such as dies and BGAs. It also lets you import objects whose names are identical to objects already in your drawing. These components are not part of your design, but appear in lists of components or in component reports. Because they contain logic, they also remain attached to nets in the design.

Use of the Command

This command now deletes all the information about a die including tile groups, die parameter record, IC group, and LEF/DEF properties. It also updates the design link between the corresponding `.cio` file (for co-design dies) and the parent `.mcm`.

Related Topics

- [Removing Unplaced Components from the Design Database](#)

Purge Unplaced Components Dialog Box

Access Using

- *Menu Path: Logic – Purge Unplaced Components*

The Purge Unplaced Components dialog box lists all unused components by their reference designator (refdes) in the current design.

 Co-design dies appear with an asterisk in this dialog box.

<i>Delete</i>	Lists all unplaced components in the current design for removal. By default, all unplaced components are removed.
<i>Save</i>	Lists the unplaced components you elect to keep in the design database.
<i>Save all -></i>	Moves the entire list of unplaced components to the Save window.
<i><- Delete all</i>	Moves the entire list of saved components to the Delete window.
<i>OK</i>	Removes all the components listed in the Delete window from your design, closes the dialog box, and returns the tool to an idle state.
<i>Cancel</i>	Closes the dialog box without removing any components.

Removing Unplaced Components from the Design Database

To remove unplaced components from the design database, perform these steps:

1. Run `purge unplaced comps` from the command window prompt.

If there are unplaced components in your design, the Purge Unplaced Components dialog box appears with all unplaced component reference designators listed in the left window. If the database contains no unplaced components, that message appears in the command window and the tool remains in the idle state.

2. Determine which unplaced components you want to retain in your design database by:

- Clicking on individual refdes names to move them into the *Save* window
- Clicking *Save all* -> to retain all unplaced components

3. To remove the chosen unplaced components from the design, click *OK*.

4. To see the results of the purge operation, run `viewlog` to display `purgeUnplacedComps.log`.

If the system is unable to purge one or more unplaced components, error messages appear at the command window.

Related Topics

- [purge unplaced comps](#)

purge unused nets

The `purge unused nets` command lets you remove some or all unused nets left in your design database when you remove or replace design objects (such as dies, BGAs, etc.) or import objects whose names are identical to objects already in your drawing. These nets are not associated with any pins, shapes, or other design objects other than properties, but appear in lists of nets or net reports.

You can run `purge unused nets` as a separate command to clean up a design at any time; however, the functionality also runs automatically as part of [die text in](#), [bga text in](#), [deassign net](#), [die editor](#), and [bga editor](#).

Related Topics

- [Removing Unused Nets from the Design](#)

Purge Unused Nets Dialog Box

Access Using

- *Menu Path: Logic – Purge Unused Nets*

The Purge Unused Nets dialog box lists all unused nets in the current design.

<i>Delete</i>	Lists all unused nets in the current design.
<i>Save</i>	Lists the unused nets you elect to keep in the design database
<i>Save all -></i>	Moves the entire list of unused nets to the Save window
<i><- Delete all</i>	Moves the entire list of saved nets to the Delete window
<i>OK</i>	Removes from your design all the nets listed in the Delete window, closes the dialog box, and returns the tool to an idle state
<i>Cancel</i>	Closes the dialog box without changes

An asterisk (*) appearing before the net name indicates that the net has a property attachment.

Removing Unused Nets from the Design

This procedure outlines the use of purge unused nets as a separate command, but it is applicable when run within `deassign net` and the text wizards.

1. Run `purge unused nets` from the command window prompt.
If there are unused nets in your design, the Purge Unused Nets dialog box displays with all unused nets listed in the left window. If the database contains no unused nets, that message displays in the command window and the tool remains in the idle state.
2. Determine which, if any, unused nets you want to retain in your design database by:
 - Clicking on individual net names to move them into the Save window
 - Clicking *Save all* -> to retain all unused nets
3. To remove all unused nets in the design, click *OK*.
4. To see the results of the purge operation, run `viewlog` to display the log file, *purgeUnusedNets.log*.

If the system is unable to purge one or more unused nets, descriptive error messages are displayed at the command window of the user interface.

Related Topics

- [purge unused nets](#)

push connectivity

The `push connectivity` command pushes the nets of selected symbols and pins to all connected objects

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

Dynamic shapes and filled rectangles are not considered part of connectivity. Static shapes are supported.

Related Topics

- [Advanced Selection Filtering Dialog Box](#)

Push Connectivity: Options Panel

Access Using

- *Menu Path: Logic – Push Connectivity.*

Control	Description
Enable advance selection	Select to open the Advanced Selection Filtering dialog box to limit the push to only specific nets only. Not selected by default.
Ignore fixed property on net	Select to ignore the FIXED property attached to any nets and push the change. Selected by default.
Run purge unused nets on done	Select to remove all unused nets. Not selected by default.

Advanced Selection Filtering Dialog Box

Access Using

- *Menu Path:* The *Enable advance selection* option in the *Options* panel for Push Connectivity (*Logic – Push Connectivity*).

In this dialog box, select the nets to which the connectivity should be pushed.

Related Topics

- [push connectivity](#)

pwd

The `pwd` (print working directory) command displays the pathname of the current working directory in the display line. This command is similar to the Unix `pwd` command.

