



S 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	20
S Commands	20
save	23
Saving a Design	24
save_as	25
Saving a File with Another Name	25
Save As Dialog Box	26
save_settings	27
Save Settings Dialog Box	28
Saving the Active UI Settings of the Layout Editor	29
script	30
Scripting Dialog Box	31
Creating a Script	32
Creating a Macro	33
Replaying a Script	34
Replaying a Macro	35
Converting a .jrl File to a Script	36
Recording/Replaying Padstack Scripts	37
scriptmode	38
Controlling the Replay of Scripts	39
sctab	40
Select Objects on the Canvas	41
set	42
Setting Variables	43
set default layers	44
settoggle	45
Changing Environment Variable Values	46
signal setup	47
SI Design Setup Wizard	48
Selecting Setup Categories	49

Selecting Xnets and Nets to Setup	50
Setting Up Search Directories and File Extensions	51
Performing Setup Operations on Selected Categories	54
Setting up Power and Ground Nets	55
Assigning Models to Components	58
Setting up Design Cross-Section	59
Setting Up SI Simulations	60
Completing the Setup	61
Setting Up Differential Pairs	62
Setting up Design to Perform SI Simulations	67
setwindow	68
shadow	69
Color Dialog Box	71
Setting Shadow Mode from the Console Window Prompt	72
shadow toggle	73
shape	74
shape add	75
Shape Add Command: Options Panel	76
Adding a Dynamic Copper Fill Polygon Shape	78
Adding a Static Polygon Shape	80
shape add circle	81
Shape Add Circle Command: Options Panel	82
Adding a Dynamic Copper Fill Circular Shape	83
Adding a Static Circular Shape	84
shape add rect	86
Shape Add Rect Command: Options Panel	87
Adding a Dynamic Copper Fill Rectangular Shape	88
Adding a Static Rectangular Shape	90
shape_app add	92
Shape_app Add Command: Options Panel	93
Adding a Dynamic Copper Fill Polygon Shape to the Design	94
Adding a Static Polygon Shape to the Design	95
shape_app add circle	96
Shape_app Add Circle Command: Options Panel	97
Adding a Dynamic Copper Fill Circular Shape to the Design	98
Adding a Static Circular Shape to the Design	100
shape_app add rect	102

Shape_app Add Rect Command: Options Panel	103
Adding a Dynamic Copper Fill Rectangular Shape to the Design	104
Adding a Static Rectangular Shape to the Design	105
shape_app change type to dynamic	106
Changing Shape Fill From Static Solid To Dynamic Copper Fill	107
shape_app change type to static	108
Changing Shape Fill From Dynamic Copper Fill To Static Solid	109
shape_app check	110
Checking For Narrow Parts In The Design	111
shape assign net	112
Shape Assign Net Command: Options Panel	113
Assigning Nets To Selected Shapes	113
shape change type	114
Shape Change Type Command: Options Panel	115
Changing a Static Shape Fill Type To Dynamic Copper	116
Changing a Dynamic Copper Shape Fill Type to Static	117
shape check	118
Shape Check Command: Options Panel	119
Checking Solid Fill Etch/Conductor Shapes for Vector-based Artwork	120
Checking Dynamic Copper Fill Etch/Conductor Shapes for Vector-based Artwork	121
shape copy layers	122
Shape Copy to Layers Dialog Box	123
Copying Shapes to Selected Subclass Layers	124
shape defer fill	125
shape edit boundary	126
Shape Edit Boundary Command: Options Panel	127
Changing a Shape or Void Outline	127
shape global param	129
Global Dynamic Shape Parameters Dialog Box	129
Deferring Dynamic Copper Fill for All Shapes In a Board Design	137
Specifying Global Parameters for Dynamic Shapes	137
shape layer param	138
shape lower priority	139
Lowering Shape Priority	140
shape merge shapes	141
Shape Merge Shapes Command: Options Panel	142

Merging Overlapping Shapes Connected to the Same Net in the Design	143
shape operations or	144
Merging Multiple Overlapping Shapes to get the Combined Shape	145
shape operations and	146
Merging Multiple Overlapping Shapes to get the Intersection Shape	147
shape operations andnot	148
Merging Overlapping Shapes with ANDNOT Operation	149
shape operations xor	150
Merging Overlapping Shapes with XOR Operation	151
shape param	152
Static Shape Parameters Dialog Box	153
Dynamic Shape Instance Parameters Dialog Box	157
Changing Parameters for a Shape Instance	161
Restoring a Shape-instance Override Value to a Global Parameter Value	162
shape raise priority	163
Raising Priority of Dynamic Shapes	164
shape report	165
Generating a Dynamic Shapes Report	166
shape select	167
Shape Select Command: Options Panel	168
Shape Select Command Tasks	169
Deferring Dynamic Copper Fill for a Single Dynamic Shape	170
Canceling Dynamic Fill	171
Deleting a Vertex	172
Editing User-defined Manual Voids	174
Changing the Net Assigned to the Shape	176
Changing a Filled Rectangle to a Shape	178
Reviewing Shape Instance Parameters	180
Moving an existing shape or void	181
Deleting a Shape	182
Moving a Segment	183
Running a Dynamic Shape Report	184
Identifying Out-of-date Dynamic Shapes	185
Assigning a Higher Voiding Priority to a Dynamic Shape	186
shape to cline	187
Shape To Cline Command: Options Panel	188
Converting Shapes to Clines	189

shape to padstack	190
Shape To Padstack Command: Options Panel	191
Converting Shapes to Padstacks	192
shape to via	193
Shape To Via Command: Options Panel	194
Converting Shapes to Vias	195
shape vertex add	196
shape void circle	197
Shape Void Circle Command: Options Panel	198
Adding a User-Defined Circular Void to a Shape	198
shape void copy	200
Shape Void Copy Command: Options Panel	201
Copying Voids in Rectangular Patterns	203
Copying Voids in Radial Patterns	204
shape void delete	207
Shape Void Delete Command: Options Panel	208
Deleting a Void in a Shape	209
Deleting All Voids	210
Deleting Island Voids	211
shape void element	212
Shape Void Element Command: Options Panel	213
Generating Voids for Static Positive Shapes on Positive Etch/Conductor Layers Automatically	214
Debugging Improperly Voiding Vias In Dynamic Plane Shapes	215
shape void move	216
Shape Void Move Command: Options Panel	217
Moving an Existing Void	218
Moving Multiple Voids	219
shape void polygon	220
Shape Void Polygon Command: Options Panel	221
Adding a Void Area in an Etch/Conductor Shape Interactively	221
shape void rectangle	223
Shape Void Rectangle Command: Options Panel	224
Adding a User-Defined (Manual) Rectangular Void to a Shape	224
shape zigzag	226
Shape Zigzag Command: Options Panel	227
Creating Zigzag Patterns Between Long Metal Shapes	228

shapeupdate	229
shapeedit	230
Shape Edit Application Mode: Options Panel	231
Shape Edit Tasks	233
Sliding a Segment	234
Extending a Segment	235
Adding a Notch	236
Moving Vertex	237
Chamfering Corners	238
Chamfering All Corners	239
Rounding Corners	240
Rounding All Corners	241
Joining Segments	242
Sliding Multiple Segments	243
shape freeze	244
shape unfreeze	245
shell	246
Opening a Window to the Host Operating System	247
shorting via array	248
Shape Shorting Via Array Dialog Box	249
show allpanes	251
Displaying All Foldable Window Panes	252
showhide dcm	253
showhide find	254
Controlling the Visibility of the Find Panel	255
showhide options	256
Controlling the Visibility of the Options Panel	257
showhide text	258
Controlling the Visibility of the Command Window Pane	259
showhide view	260
Controlling the Visibility of the Worldwide Window Pane	261
showhide views1	262
Controlling the Visibility of the Split View Window Pane	263
showhide vis	264
Controlling the Visibility of the Visibility Window Pane	265
showhide_workflow	266
show element	267

Show Element Dialog Box	268
Displaying Design Attributes for an Object	269
Finding an Object by its Property	270
Finding an Object by its Name	271
show measure	272
Measure Dialog Box	273
Measuring Distance between Two Points in the Design	274
show parasitic	275
Parasitics Calculator Dialog Box	276
Measuring Capacitance Between Two Conductor Elements in the Design	277
show property	278
Show Property Dialog Box	279
Finding Elements With A Specific Property/value	280
Displaying Properties Graphically	281
show rlc	282
show symbol drc	283
show waived drcs	283
Showing Waived DRC Error Markers in the Design	284
signal 3dmodel	285
Signal 3D Model Dialog Boxes	287
3-D Interconnect Modeling Dialog Box	288
3-D Modeling Port Group Dialog Box	290
Wire Bond Profile Editor	291
Creating a 3D Package Device Model For PCB-level Simulation	293
Creating a 3D Package Interconnect Device Model For PCB-level Simulation	294
Grouping Pins in a Multiport Net	295
Enabling Mapping Between Package Bump Pads And Die Bump Pads	296
signal atimes	297
Crosstalk Active Times Import Dialog Box	298
Importing Loading Timing Data to the Design Database	299
signal audit	300
Signal Audit Dialog Boxes	301
Signal Audit Tasks	302
Controlling the Tests to Run	303
Selecting Xnets and Nets to Audit	304
Viewing Audit Errors	305
Highlighting Errors	307

Resolving Errors	308
Viewing Error Report	309
Importing Ignored Errors	310
signal bus setup	311
Signal Bus Setup Dialog Boxes	312
Signal Bus Setup Dialog Box	313
Create Simulation Buses Dialog Box	316
Setting Up A Simulation Bus	317
Setting Up A Signal Bus	318
signal bus sim	319
Analysis Bus Simulation Dialog Box	319
Simulating the Source Synchronous Buses in your Design for Analysis	321
signal demiaudit	322
signal pkg_model	323
Signal PKG Model Dialog Boxes	325
Creating a 3D Package Model For PCB-level Simulation	329
signal prefs	330
Signal Prefs Dialog Boxes	331
Analysis Preferences Dialog Box	332
Device Models Tab	332
InterconnectModels Tab	332
Power Integrity Tab	334
EMS2D Preferences Dialog Box	336
Via Model Extraction Setup Dialog Box	337
Advanced Simulator Preferences Dialog Box	342
Advanced Preferences Dialog Box	345
General Tab	345
OATS Tab	345
Near Field Tab	345
Fast/Typical/Slow Simulations Definition Dialog Box	348
General Tab	348
Pin Parasitics Tab	348
Reference Voltages Tab	349
V/I Currents Tab	350
Terminators Tab	352
Thresholds Tab	352

3-D Modeling Parameters Dialog Box	354
Signal Prefs Tasks	359
Setting Layers to be Ignored by the 3D Field Solver	360
Specifying Glitch Settings	361
Setting Adaptive Mesh Settings	362
Setting up a Spectre/Hspice Simulation	363
Running a Spectre/Hspice Simulation	364
Setting Time-domain Voltage Ripple Display Settings	366
Setting Frequency-domain Impedance Settings	367
Setting Geometry Windows At The Drawing Level	368
Setting Transient Simulation Preferences	369
Setting and Detecting Coplanar Waveguides	370
signal snrscreen	372
signal report	373
signal probe	374
Signal Probe Dialog Boxes	375
Signal Analysis Dialog Box	376
Signal Select Browser Dialog Box	377
Analysis Report Generator Dialog Box	378
Analysis Waveform Generator	383
Stimulus Setup Dialog Box	387
Signal Analysis Tasks	387
Choosing a Group of Nets for Analysis	389
Choosing a Single Net or Pin Pair for Analysis	390
Choosing Groups of Violation Nets for Analysis	391
Choosing Xnets for Simulation	392
Setting Custom Stimulus for Report Generation	393
Setting Custom Stimulus for Waveform Generation	394
Creating Waveforms	395
Creating Standard Reports	396
Creating Custom Reports	397
Performing Signal Quality Screening	399
Starting SigXplorer from the Simulator	401
signal init	402
Signal Analysis Initialization Dialog Boxes	403
System Configuration Editor	405
Connect By Component Dialog Box	407

Creating a New Case	408
Editing a Case Description	409
Removing a Case	410
Choosing a System Configuration	411
Choosing the Current Case	412
signal lib audit	413
signal library	414
Signal Analysis Library Dialog Boxes	415
SI Model Browser	416
DML Library Management Dialog Box	418
Set Model Search Path Dialog Box	419
Analog Output Model Editor Dialog Box	420
IBIS Device Model Editor Dialog Box	421
IBIS Device Pin Data Dialog Box	426
Buffer Delays Dialog Box	428
IOCell Editor Dialog Box	429
V/I Curve Editor Dialog Box	431
V/T Curve Editor Dialog Box	432
Set V/I Curve Point Dialog Box	433
Signal Library Tasks	434
Working with Models and Libraries	434
Specifying a Working Device Model Library/interconnect Model Library	436
Adding a Device Library or Index	437
Adding a Standard Cadence Library	438
Deleting a Library From The Search List	439
Creating a Device Model Index	440
Creating a Device Model Index from the Operating System Command Line	441
Reordering the Libraries in the Search List	442
Merging Device Model Libraries	443
Translating Other Device Model Library Formats to DML	444
Working with Device Models	445
Editing an Analog Output Model	446
Creating a Cable Model	447
Editing a Cable Model	448
Creating a DesignLink Model	449
Editing a DesignLink Model	450
Creating an ESpice Device Model	451

Editing an ESpice Device Model	452
Creating an IBIS Device Model	453
Editing an IBIS Device Model	454
Adding a Pin to an IBIS Device Model	455
Editing the Pin Data for an Existing Pin on an IBIS Device Model	456
Creating an IOCell Model	457
Editing an IOCell Model	458
Creating a Package Model by Copying and Editing an Existing Model	459
Editing a Package Model	460
Adding a Package Model to an IBIS Device Model	461
Working with Interconnect Models	462
Editing a Trace, MultiTrace, Pin or Shape Model	463
signal libs audit	463
signal model	464
Signal Model Assignment Dialog Box	465
Create Device Model Dialog Box	468
Create IBIS Device Model Editor Dialog Box	469
Create ESpice Device Model Editor Dialog Box	470
Signal Model Assignment Tasks	471
Assigning Device Models to Components	472
Assigning IOCell Models to Pins	473
Assigning Programmable Buffer Models to Pins	474
Assigning Trace Models to Bondwires	475
Assigning a Trace Model to a Single Bond Wire	476
Assigning a Trace Model to Multiple Bond Wires	477
Assigning a Trace Model to All Bond Wires	478
Removing All Trace Model Assignments	479
Automatically Assigning Device Models to Discrete Components	480
Creating a New Device Model in Model Assignment	481
Editing a Device Model in Model Assignment	482
Removing a Device Model Assignment	483
Setting Up Models in Allegro PCB Design Entry HDL, System Connectivity Manager, or Third-Party Libraries	484
signal modeedit	484
signal model refresh	485
Model Dump/Refresh Dialog Box	486
Generating Device Model Reports	487

signal demiprefs	488
signal demiprobe	489
signal xtalktable	490
Signal Analysis Crosstalk Table Dialog Box	491
Creating a Crosstalk Table	493
Selecting a Crosstalk Table	494
Exporting a Crosstalk Table	495
Importing a Crosstalk Table	496
Deleting a Crosstalk Table	497
signalintegrity	498
Commands Automatically Run in Application Mode	499
Signal Integrity Command Tasks	500
Activating Application Mode	501
Controlling Application Mode Status Icon and Tip	503
Deactivating Application Mode	504
Accessing Frequently-Used SI Commands	505
Highlighting Objects on Mouse Over	507
Using Super Filter	508
Customizing Layout Editor Functions	510
signal stimulus	511
signal topology	512
signal waveform	513
signoise	514
Generating a Crosstalk Table froma Batch Mode Simulation	515
sigxp	516
silkscreen audit	517
Generating Audit Results	518
silkscreen execute	519
Creating Silkscreen Data	520
silkscreen param	521
Auto Silkscreen Dialog Box	522
Creating Silkscreen Data Automatically	523
skill	524
Executing any AXL-SKILL Function on the Command Line	525
slide	526
Slide Command: Options Panel	527
Sliding Connect Line Segments	529

Sliding Vias or Rat Ts	530
smi message detail	531
Help Message Detail Dialog Box	532
snap_rat_t	533
Routing a Connection on a Net Scheduled with T Points	534
soft net	535
Soft Net Partition Assignment Dialog Box	536
Assigning Soft Nets to a Specified Partition	537
source	538
spdif clarity3dlayout	539
Using the SPDIF Clarity3DLayout Command	540
specctra	541
Generating Routing Files	542
Automatic Router Parameters Dialog Box	543
543	
specctra_in	547
Import from Auto-Router Dialog Box	548
Importing Data from Allegro PCB Router to a Design	549
specctra_out	550
Export to Auto-Router Dialog Box	551
Exporting Data from Design to Allegro PCB Router	552
specctra checks	553
Running Router and Alignment Checks on a Design	554
spif	555
SPIF Dialog Boxes	556
Running the Pre-Route Checker	556
spif_batch	558
Performing a Pre-Routing Check on a Design	559
spin	560
560	
Spin Command: Options Panel	561
Rotating a Graphic Element	562
Rotating Multiple Graphic Elements	563
Rotating an RF Clearance Assembly Group	564
Entering Simulation Details, Choosing Report Formats, and Simulating And Generating Reports	565
split plane create	565

Create Split Plane Dialog Box	566
Creating a Split Plane	567
split plane param	568
Split Plane Params Dialog Box	569
Setting Parameters for Split Planes	570
spread between voids	571
Spreading Between Voids	572
spread clines	573
Spread Clines Command: Options Panel	574
Spreading Clines	575
spreadsheet to symbol	576
Symbol Update from Spreadsheet Dialog Box	577
Importing Information from a Spreadsheet	579
stab	580
stacked via report	581
status	582
Status Dialog Box	583
Displaying Status for RF Components	585
step_out Batch Command	587
step out	588
STEP Export Dialog Box	589
Exporting an Allegro Layout as a STEP Model	591
step pkg map	592
Device/Package STEP Mapping Dialog Box	592
Mapping STEP Model to Device/Package	595
Mapping STEP Model to Mechanical Symbol	596
Exporting STEP Models	597
Importing STEP Models	598
stop	599
stopwatch	600
stream_out Batch Command	601
stream out	603
Stream Out Dialog Box	604
Stream Out Edit Layer Conversion File	607
Converting an Allegro Design to GDSII Stream Format	609
Editing the Stream Out Layer Conversion File	610
stream padstacks	611

Stream Structure Import Dialog Box	612
Creating Padstack Definitions for Pins and Vias	612
strip_design	613
IP Strip Design (for Cadence Testcases) Dialog Box	615
Stripping IP from Design Database	616
stroke_editor	617
Removing Strokes from a Stroke File	617
stroke editor	618
Stroke Editor Window	619
Adding New Strokes to a Stroke File	620
Changing Existing Strokes	621
strokefile	622
Specifying a File Containing Your Own Strokes	623
subclass	624
swap	625
swap area design	626
Defining the package/part keepin as the Automatic Swapping Area	627
swap area list	628
swap area room	629
Defining Rooms for Automatic Swapping	630
swap area window	631
Defining Window Areas for Swapping	632
swap components	633
Swap Components Command: Options Panel	634
Swapping Components in Pre-Selection Mode	635
Swapping Components in Verb-Noun Mode	636
swap execute	637
Running the Automatic Swap Process	638
swap functions	639
Swapping Functions or Gates in a Design	640
swap param	641
Automatic Swap Dialog Box	642
Swapping Automatically	643
swap pins	644
Swap Pins Command: Options Panel	645
Swapping Pins in Pre-Selection Mode	646
Swapping Pin Pairs of Two Differential Pair Nets	647

Swapping Pins of a Single Differential Pair Net	648
symbol	649
Choosing a Symbol to Execute Active Command on it	650
symbol_check	651
symboledit	652
Symboledit Dialog Boxes and Options Panel	653
Symbol Edit application mode - Add component: Options Panel	654
Symbol Edit application mode - Add Driver: Options Panel	655
Symbol Edit application mode - Add Keepin/Keepout: Options Panel	656
Symbol Edit application mode - Bump/Ball attributes: Options Panel	657
Symbol Edit application mode - CTE Compensation: Options Panel	658
Symbol Edit application mode - Die properties: Options Panel	659
Symbol Edit application mode - Edit Boundary: Options Panel	660
Symbol Edit application mode - Grid Add: Options Panel	661
Symbol Edit application mode - Pin Add: Options Panel	662
Symbol Edit application mode - Pin Move/Copy/Modify: Options Panel	665
Symbol Edit application mode - Pin Numbering settings: Options Panel	667
Symbol Edit application mode - Pin Pitch Settings: Options Panel	668
Symbol Edit application mode - Pin Text Settings: Options Panel	669
Symbol Edit application mode - Place Driver: Options Panel	670
Die Abstract Write Dialog Box	671
Refresh Co-Design Die: Options Panel	672
Refresh Co-Design Die Finish Form	673
Net for Component Pin Dialog Box	673
Driver Move: Options Panel	674
Align Driver: Options Panel	675
Respace Driver: Options Panel	676
Swap Driver: Options Panel	677
Accessing Command Help	678
Symbol Edit Application Mode Tasks for Various Object Selection	679
Adding a Fully-Customized Component from Scratch	681
Adding a Pin	682
Adding a Grid	683
Specifying Pin Pitch Settings	684
Specifying Pin Numbering Settings	685
Viewing and Editing IC Details	686
Changing Bump/Ball Attributes	687

Editing Die Properties	688
Specifying Pin Text Settings	689
Editing Boundary	690
Adding a Keepin or Keepout	691
Comparing components	692
Writing a die abstract file	693
Swapping Pins	694
Copying a Component	695
Refreshing a Distributed Co-design Die	696
Writing Device File to Disk	697
Writing Symbol Spreadsheets	697
Renaming a Component or Symbol	698
Converting Vias to Pins	699
Specifying CTE Compensation for Components	699
Deleting Pins	699
Moving Pins	700
Copying Pins	700
Changing Pin Attributes	700
Aligning Pins	700
Respacing Pins	701
Converting Pins to Vias	701
Moving I/O drivers in a co-design die	701
Aligning I/O drivers in a co-design die	702
Respacing I/O drivers in a co-design die	702
Swapping I/O drivers in a co-design die	702
Changing I/O driver placement status	703
symbol to spreadsheet	703
Symbol to Spreadsheet Dialog Box	704
Exporting to a Spreadsheet	706
sna param	707
Signal Analysis Parameters Dialog Box	708

S Commands

save	save_as	save_settings
script	scriptmode	sctab
set	set default layers	settoggle
signal setup	setwindow	shadow
shadow toggle	shape	shape add
shape add circle	shape add rect	shape_app add
shape_app add circle	shape_app add rect	shape_app change type to dynamic
shape_app change type to static	shape_app check	shape assign net
shape change type	shape check	shape copy layers
shape defer fill	shape edit boundary	shape global param
shape layer param	shape lower priority	shape merge shapes
shape operations or	shape operations and	shape operations andnot
shape operations xor	shape param	shape raise priority
shape report	shape select	shape to cline
shape to padstack	shape to via	shape vertex add
shape void circle	shape void copy	shape void delete
shape void element	shape void move	shape void polygon
shape void rectangle	shape zigzag	shapeupdate
shapeedit	shape freeze	shape unfreeze
shell	shorting via array	show allpanes
showhide dcm	showhide find	showhide options
showhide text	showhide view	showhide views1
showhide vis	showhide_workflow	show element

S Commands

S Commands

show measure	show parasitic	show property
show rlc	show symbol drc	show waived drcs
signal 3dmodel	signal atimes	signal audit
signal bus setup	signal bus sim	signal demiaudit
signal pkg_model	signal prefs	signal snrscreen
signal report	signal probe	signal init
signal lib audit	signal library	signal libs audit
signal model	signal modeledit	signal model refresh
signal demiprefs	signal demiprobe	signal xtaktable
signalintegrity	signal stimulus	signal topology
signal waveform	signoise	sigxp
silkscreen audit	silkscreen execute	silkscreen param
skill	slide	smi message detail
snap_rat_t	soft net	source
spdif clarity3dlayout	specctra	specctra_in
specctra_out	specctra checks	spif
spif_batch	spin	split plane create
split plane param	spread between voids	spread clines
spreadsheet to symbol	stab	stacked via report
status	step_out Batch Command	step out
step pkg map	stop	stopwatch
stream_out Batch Command	stream out	stream padstacks
strip_design	stroke_editor	stroke editor
strokefile	subclass	swap
swap area design	swap area list	swap area room
swap area window	swap components	swap execute

S Commands

S Commands

swap functions	swap param	swap pins
symbol	symbol_check	symboledit
symbol to spreadsheet	sna param	

save

The `save` command saves the currently active design either with the current name, or, if you choose not to over-write the design, another file name while keeping the design displayed and active.

Co-Design Environment

When saving changes made to co-design dies, the layout tool copies the temporary Open Access (OA) library/cell/view, where the Cadence I/O Planner (IOP) saved changes you made to the IC, over the OA library/cell/view containing the last saved version of the co-design IC.

In addition, if the layout design contains batched changes to a co-design die that have not yet been updated to the IC design, you are asked if the changes should automatically be updated to the IOP design before it saves the layout design. If you do not save at this time, you can always save later during a subsequent save operation, or they are automatically updated to IOP the next time you edit this die.

You need to save the or `.mcm` database containing the Layout, and the OA libraries containing the layout cell views for each co-design IC. References to the OA libraries are stored in the `.mcm` database, but the OA libraries themselves are not saved within the layout database. They are stored separately on disk and must be maintained and archived with the layout database, or the co-design die details are lost.

The `save` command displays a standard confirmmer window and file browser.

Access Using

- *Menu Path:* File – Save
- *Toolbar Icon:* 

Saving a Design

Follow these steps to save a design:

1. Run `save` from the command window prompt, followed by the file name you want the design saved to.
If the design is new (has never been saved), the filename is written to disk.
If the design already exists, a confirmmer window is displayed with the message:
`<path>/<filename>`: File Exists. Overwrite?
2. Click Yes to over-write the earlier saved version of the file – or– No to display a file browser.
3. To save the layout to a file with a different name, enter the new name into the *File Name* field, and click *OK*.
After the current design has been saved under another name, the design changes to the new name.

Example

```
save <filename>
```

save_as

The `save_as` command saves an existing file under another name, to another drive, to another directory, or in a different format, such as a different symbol type.

Co-Design Environment

For information on how this command works in a co-design environment, see the chapter, *Generating a Co-Design Die* in the *Placing the Elements User Guide*.

Syntax

You can run the `save_as` command from the command window prompt. The syntax is:

```
save_as [<new design name>]
```

Example

```
save_as master.brd
```

If you do not provide the `<new design name>` argument, the layout editor opens a browser window in the current directory.

Related Topics

- [Saving a File with Another Name](#)

Saving a File with Another Name

To save a file with a different name, follow these steps:

1. Run the `save_as` command.
The Save As dialog box appears.
2. Click the left button below the *Help* button to display a text preview of the current design.
The preview area appears on the right side of the list box.
3. Click the right button below the *Help* button to display the graphics preview of the design.
4. Enter the new file name in the *File Name* box.
5. If necessary, click the *Save in* list to choose a different directory.
6. Click *Save*.

Related Topics

- [save_as](#)

Save As Dialog Box

Access Using

- *Menu Path: File – Save As*

The `Save As` dialog box is a standard file browser. Two buttons appear below the *Help* button. The left button lets you display a text preview of the current design; the right button lets you display the graphics preview of the design. The preview area appears on the right side of the list box.

save_settings

The `save_settings` command saves the currently active UI settings of the layout editor with a new name.

You can configure user interface of the layout editor for a specific task and save the settings. Multiple settings can be saved based on the different task requirements. For example, you can save settings for placements, settings for routing, settings for design checks, and so on.

UI settings include the state and the location of toolbars and docking panes. The settings are saved in the `Registry Editor` of Windows and `$HOME` of Linux.

 Saving a new setting becomes the current default. Opening a new session of layout editor shows this setting by default.

Related Topics

- [Saving a File with Another Name](#)

Save Settings Dialog Box

Access Using

- Menu Path: *View – UI Settings – Save Settings*

The command displays *Save UI Settings* dialog box to enter a name.

Saving the Active UI Settings of the Layout Editor

Follow these steps to save the currently active UI settings of the layout editor:

1. Run `save_settings` from the command window prompt.
If the name is new (has never been saved), the settings are written to memory.
If the name already exists, following error message is displayed:
`Setting with name:<name> already exists. To save this UI configuration, use a unique name.`
2. Click *OK* to save the settings with a different name.
3. Enter the new name into the field, and click *Save*.

Related Topics

- [save_settings](#)

script

The `script` command records a series of actions. It creates a text file containing the commands that you execute and adds a `.scr` extension to the file name. You can use scripts to perform global tasks such as setting up dialog box options, adding elements to multiple databases at the same location, and duplicating drawings. Using the interactive version of the `script` command that displays the Scripting dialog box, you can also replay the script.

A macro is a script that lets you automate a series of point selections and replay them, starting at another coordinate. When you replay a macro, the layout editor prompts you for a starting point (origin). The macro places the point selections you recorded relative to this starting point. This is useful in performing operations that you need to repeat on a board/design drawing, such as repeating complex geometric operations.

The current settings in your design are recorded in the script or macro. To display the script with different settings, you must change them as part of the script.

 Cadence tools support many script commands that you can use for various purposes. These include `record`, `recordmacro`, `replay`, `stop`, `stopwatch`, `scriptmode`, and `repeat again`. For information on using environment commands in scripts, see `ifvar` and `ifnvar`.

Environment Variables

To keep the Scripting dialog box open, set the `script_keepformopen` environment variable using *Setup – User Preferences*. When you set this variable, the dialog box does not close when you click the *Replay* button. To specify a script to run on startup, set the `script_startup` environment variable.

Scripting in Allegro X Advanced Package Designer

For information on scripting in Allegro X Advanced Package Designer (APD), see *Generating a Co-Design Die* in the *Placing the Elements User Guide*.

Related Topics

- [Creating a Script](#)
- [Creating a Macro](#)
- [Replaying a Script](#)
- [Replaying a Macro](#)
- [Converting a .jrl File to a Script](#)
- [Recording/Replaying Padstack Scripts](#)
- [record](#)
- [recordmacro](#)
- [stop](#)
- [stopwatch](#)
- [scriptmode](#)
- [repeat_again](#)
- [ifvar](#)
- [ifnvar](#)

Scripting Dialog Box

Access Using

- Menu Path: *File – Script*

Script File	
Name	Specifies the name of the file in which you record your actions. the layout editor adds the <code>.scr</code> extension to the file name.
Browse	Displays the script file data browser that lets you choose a script file to replay.
Library	Displays the script file data browser that lets you choose a script file to replay. Opens to your script path location.
Generate	Displays a file browser from which you can choose a <code>.jrl</code> file to convert into a script without having to leave the current environment. To process the journal file and reconstruct the appropriate script outside the layout editor, run: <code>j2script <source_jrl_file> <target_allegro_script></code>
Record/Replay	
Macro record mode	Specifies whether or not you record as a macro. When replaying, a macro requires a starting point.
Record	Starts recording your actions.
Stop	Stops recording your actions or replaying a script.
Replay	Starts replaying a macro or script.
Cancel	Closes the dialog box.
Help	Displays the Help window.

Related Topics

- [Creating a Macro](#)
- [Replaying a Script](#)
- [Replaying a Macro](#)
- [Converting a .jrl File to a Script](#)
- [Recording/Replaying Padstack Scripts](#)

Creating a Script

To create a script, perform these steps:

1. Run the `script` command.
The Scripting dialog box appears.
2. In the *Name* text box, enter a name for the script.
3. Click *Record*.
The Scripting dialog box disappears.
4. Perform the tasks that you want the script to run.
The name of the file and the *Rec* status appears in the Status window.
5. Run `script` again, then click *Stop* in the Scripting dialog box.

Related Topics

- [script](#)
- [Replaying a Script](#)
- [Replaying a Macro](#)
- [Converting a .jrl File to a Script](#)
- [Recording/Replaying Padstack Scripts](#)

Creating a Macro

To create a macro, perform these steps:

1. Run the `script` command.
The Scripting dialog box appears.
2. In the *Name* text box, enter a name for the macro.
3. Click *Macro Record Mode*.
4. Click *Record*.
The Scripting dialog box disappears.
5. Perform the tasks that you want the macro to run.
The name of the file and the *Rec* status appears in the Status window.
6. Run `script` again, then click *Stop* in the Scripting dialog box.

Related Topics

- [script](#)
- [Scripting Dialog Box](#)
- [Replaying a Macro](#)
- [Converting a .jrl File to a Script](#)
- [Recording/Replaying Padstack Scripts](#)

Replaying a Script

To replay a script, perform these steps:

1. Run the `script` command.
The Scripting dialog box appears.
2. In the *Name* *text box*, enter the name of the script that you want to replay.
If necessary, use the *Browse* button to locate the correct file.
3. Click *Replay*.
The script replays.

Related Topics

- [script](#)
- [Scripting Dialog Box](#)
- [Creating a Script](#)
- [Converting a .jrl File to a Script](#)
- [Recording/Replaying Padstack Scripts](#)

Replaying a Macro

To replay a macro, perform these steps:

1. Run the `script` command.
The Scripting dialog box appears.
2. In the *Name* text box, enter the name of the macro that you want to replay.
If necessary, use the *Browse* button to locate the correct file.
3. Click *Replay*.
The script replays.

Related Topics

- [script](#)
- [Scripting Dialog Box](#)
- [Creating a Script](#)
- [Creating a Macro](#)
- [Recording/Replaying Padstack Scripts](#)

Converting a .jrl File to a Script

To convert a .jrl file to a script, perform these steps:

1. Run the `script` command.
The Scripting dialog box appears.
2. Click *Generate*.
A file browser appears.
3. Choose a journal file to convert, which then creates a file of the same name with `.scr` appended to it in the same directory as the source journal file. Once the layout editor generates the file, its name populates the *Name* text box.
4. Repeat for as many journal files as you want to convert.

Related Topics

- [script](#)
- [Scripting Dialog Box](#)
- [Creating a Script](#)
- [Creating a Macro](#)
- [Replaying a Script](#)

Recording/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 in the Padstack Designer dialog box. To define new padstacks that share similar padstack specifications, you can replay the script file and edit the new padstacks as necessary.

Related Topics

- [script](#)
- [Scripting Dialog Box](#)
- [Creating a Script](#)
- [Creating a Macro](#)
- [Replaying a Script](#)
- [Replaying a Macro](#)

scriptmode

The `scriptmode` command provides you with options to control the replay of scripts. You can use this command when nesting scripts. This nesting capability means that when a script is finished, the original values that were in effect when the replay was started, are restored. For example, if you set windows to be invisible in the script, all windows, opened but not closed by the script, are visible after the script ends.

Cadence tools support many script commands that can be used for various purposes. These include `record`, `recordmacro`, `replay`, `stop`, `stopwatch`, `scriptmode`, `repeat` again, `ifvar`, and `ifnvar`.

Syntax

The following are guidelines for the `scriptmode` command:

- Use the + sign to enable the option and the - sign to restore the default.
- You can enter multiple options on a line.
- If you do not set any options, the layout editor uses the current settings.
- If you set this command in a script, when the script ends, the settings at the start of the script are restored.
- When entering options, you can use the option name or just use the first letter.
- The environment variable, `scriptmode`, allows changing default settings at the layout editor startup.

```
scriptmode [-+] [<options>]
```

-/+ f (flush)	Specifies flush. During record, each command is written to the disk script file. If disabled (default for better performance), the layout editor uses a memory buffer that is written to disk when full, or when the script is terminated.
-/+ b (beep)	Inhibits most beeps during script replay.
-/+ c (continue)	Specifies continue. During replay, the script continues if an error is encountered. If disabled (default), the script terminates when an error is encountered.
-/+ e (echo)	Specifies echo. During replay, the script echoes the command to the appropriate window before executing the command. If disabled (default), no echo is performed.
-/+ i (invisible)	Specifies invisible. During replay, forms are not displayed. A message is echoed to the top window with a multi-line status area every time an invisible form window is created. When the script is finished, the layout editor displays all remaining invisible forms. You cannot create an invisible form when not in a script. Disabling the invisible option in scripts does not cause the display of any currently invisible forms, but causes any form in the future to be displayed. Use this option on well debugged script files to increase performance.
-/+ n (noinfo)	If set, the script writes no information level messages to the status area. Use this switch to speed up script replay. This switch cannot block warning and error messages, however.
-/+ s (step)	Specifies step. During replay, the script stops after every command and waits for you to press a mouse button or a key before executing the next command. When this option is enabled, a window appears in the center of the screen to remind you that the script stops after every command.
-/+ w (warnerror)	Inhibits form warning and error messages.

Related Topics

- [record](#)
- [recordmacro](#)
- [replay](#)
- [stop](#)
- [stopwatch](#)
- [repeat again](#)
- [ifvar](#)
- [ifnvar](#)

Controlling the Replay of Scripts

1. Type `scriptmode` and the appropriate arguments, as described above, at the command window prompt.

Scripts run according to the settings you specified.

Following are some examples of the command:

<code>scriptmode +i +c</code>	All windows created during the script are made invisible and the script continues even with errors. Windows that are open when script ends are visible.
<code>scriptmode -echo</code>	The echo command is deactivated.
<code>scriptmode +invisible +noinfo</code>	Windows are not opened and information messages, such as Pick X Y, are not echoed.

sctab

An internal Cadence engineering command.

Select Objects on the Canvas

To select objects on the canvas, you can choose from any of the *Selection Set* pop-up menu item. The options available are:

<i>Clear all selections</i>	Clears all previous selections.
<i>Select by Polygon</i>	Select by drawing a polygon outline. All elements partially or completely contained within the boundary and matching the Find filter settings are selected.
<i>Select by Lasso</i>	Select by drawing a free-form polygon outline. All elements partially or completely contained within the boundary and matching the Find filter settings are selected.
<i>Select on Path</i>	Select by a free-form line. All elements touched by the line and matching the Find filter settings are selected.
<i>Object Browser</i>	Opens the Show Element Dialog Box dialog box.

set

The `set` command lets you temporarily define or replace an environment variable setting. When the current session ends, the variable reverts to its former value. The `unset` command also returns a variable setting to its previous value.

You can also use the `enved` command to define or replace an environment variable.

 Methods for setting environment variables vary according to the shell you are using. If you are using `csh`, for example, you can set variables using the `setenv` command. If you do not know what shell you are using, refer to your operating system documentation or see your system administrator.

Syntax

```
set <variable_name> = value(s)
```

Access Using

- Menu Path: *Tools – Utilities – Env Variables*

Related Topics

- [enved](#)
- [unset](#)

Setting Variables

To set variables in the layout editor, perform these steps:

1. Type `set`, followed by a variable name and value at the command window prompt.
2. Press Return/Enter to set the variable.

Examples

This example sets the database to save your work automatically every 30 minutes:

```
set autosave_time = 30
```

This example sets a path, in this case, the path to the clipboard library:

```
set clippath = . cliplib_test \home\jones\cliplib
```

This command directs the layout editor to look first in the current directory for the clipboard elements (signified by a period), then the directory `cliplib_test`, and finally the directory `\home\jones\cliplib`.

set default layers

The `set default layers` command opens the New Default Layers dialog box, an automatic process that occurs when you run the `new` command.

Related Topics

- [new](#)

settoggle

The `settoggle` command lets you change the value of an environment variable based on its current value and a list of possible values.

Syntax

variable name	The required environment variable name
values [1 - n]	<p>An optional list of possible values for the environment variable. If you specify no optional values and the variable is not set, the variable is set with a value of "", which is equivalent to: <code>set <variable name></code>. If you specify no optional values and the variable is currently set, the variable is unset, which is equivalent to: <code>unset <variable name></code>. If you specify one value and the variable is unset, the variable is set to that of the specified value, which is equivalent to: <code>set <variable name> value 1</code>. If you specify one value and the variable is currently set, the layout editor unsets the variable is unset, which is equivalent to: <code>unset <variable name></code>. If you specify more than one value, <code>set</code> substitutes the value listed immediately after the current environment variable value for the current variable. The comparison is case insensitive. The command sets the environment variable to the first value in the value list when the variable:</p> <ul style="list-style-type: none">• is currently unset• has a value not in the list• has the same value as the last item in the value list <p>This is equivalent to: <code>set <variable name> value 1</code></p>

Changing Environment Variable Values

Follow these steps to change the values of environment variables:

1. Type `settoggle`, followed by a variable name and value at the command window prompt.
2. Press Return/Enter to toggle the variable.

Following are some examples of setting and toggling variables:

Command	Description
<code>unset pcb_cursor</code>	Unsets the <code>pcb_cursor</code> environment variable.
<code>settoggle pcb_cursor infinite cross</code>	Sets the <code>pcb_cursor</code> environment variable to <code>infinite</code> .
<code>settoggle pcb_cursor infinite cross</code>	Sets the <code>pcb_cursor</code> environment variable to <code>cross</code> .
<code>unset display_drcfill</code>	Unsets the <code>display_drcfill</code> environment variable.
<code>settoggle display_drcfill</code>	Sets the <code>display_drcfill</code> environment variable.
<code>settoggle display_drcfill</code>	Unsets the <code>display_drcfill</code> environment variable.

signal setup

The `signal setup` command displays the *SI Design Setup wizard*, which helps you set up the design to perform SI simulations. The wizard assists you in making your board ready to run high-speed analyses. It simplifies the setup by guiding you through the required steps.

Access Using

- Menu Path: *Setup – SI Design Setup*
OR
- *Menu Path: Tools – Utilities – Keyboard Commands.* Choose *signal setup* in Command Browser.

Related Topics

- [Setting up Design to Perform SI Simulations](#)

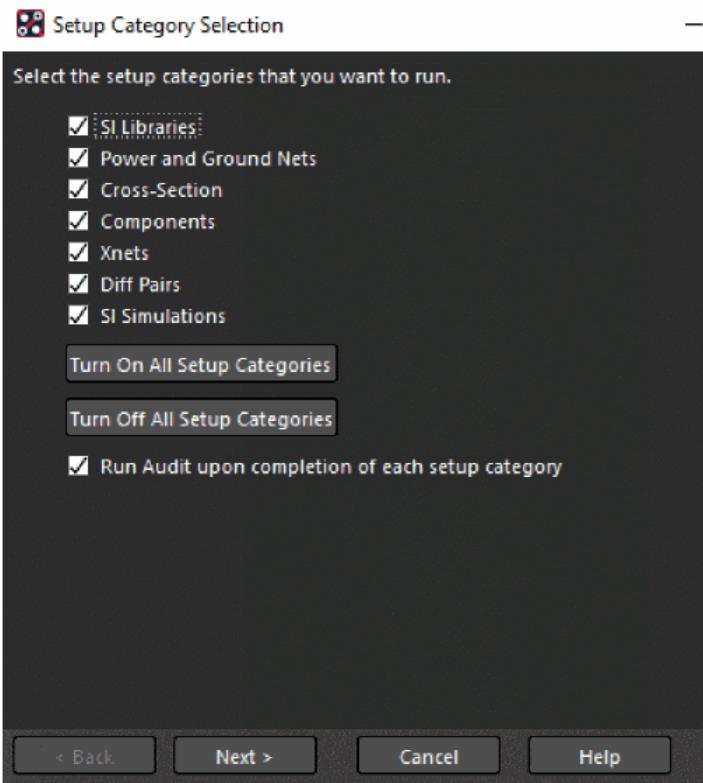
SI Design Setup Wizard

The SI Design Setup wizard provide pages to perform the following functions:

- Selecting Setup Categories
- Selecting Xnets and Nets to Setup
- Setting Up Search Directories and File Extensions
- Setting up Power and Ground Nets
- Setting up Design Cross-Section
- Assigning Models to Components
- Setting Up Differential Pairs

Selecting Setup Categories

The first page of the Setup Category Selection wizard lists the categories on which you can perform setup operations. This list is similar to the test categories used by the *signal audit* command.



You can turn on or off setup operations on all the categories using the *Turn On All Setup Categories* and *Turn Off All Setup Categories* buttons, respectively.

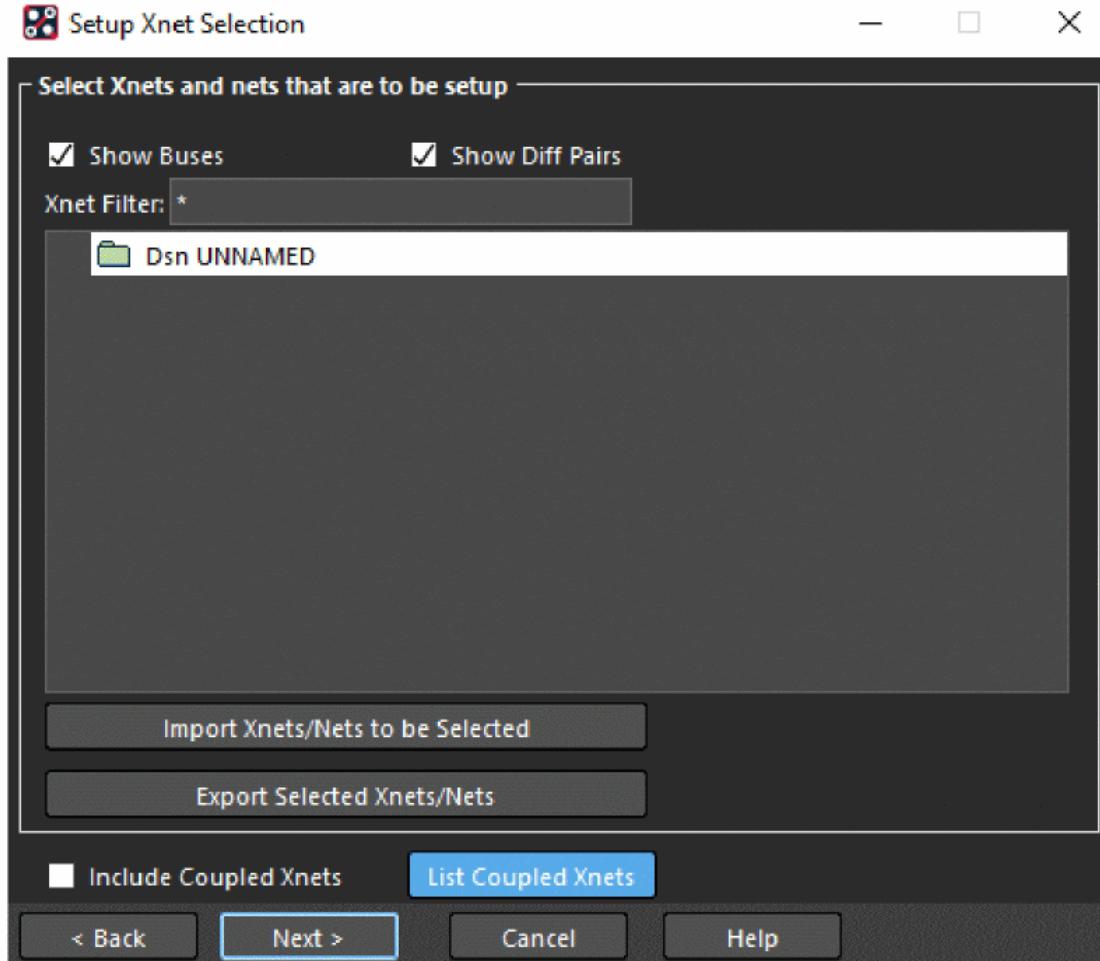
You can optionally run an audit on each category after the setup operations for a category complete. This is the default behavior. If you want to turn it off, deselect the *Run Audit upon completion of each setup category* option.

- After you complete making selections on this page, click *Next*.

Selecting Xnets and Nets to Setup

On the second page of the wizard, you select the Xnets and nets on which the setup operations are to be run. This page is the same as the second page of the SI Design Audit page wizard.

You can expand all the items in the Xnet selection tree. This tree can have several levels: *designs*, *buses*, *differential pairs*, and *Xnets* and it becomes difficult to manage large number of items. Use the *Expand All* and *Collapse All* commands on the right-click pop-up menu to expand or collapse the tree at the top level or at the selected level. *Expand All* expands the tree for the selected item as well as all the sub-items under the selected item. Likewise, selecting *Collapse All* collapses the selected item as well as all of its sub-items.



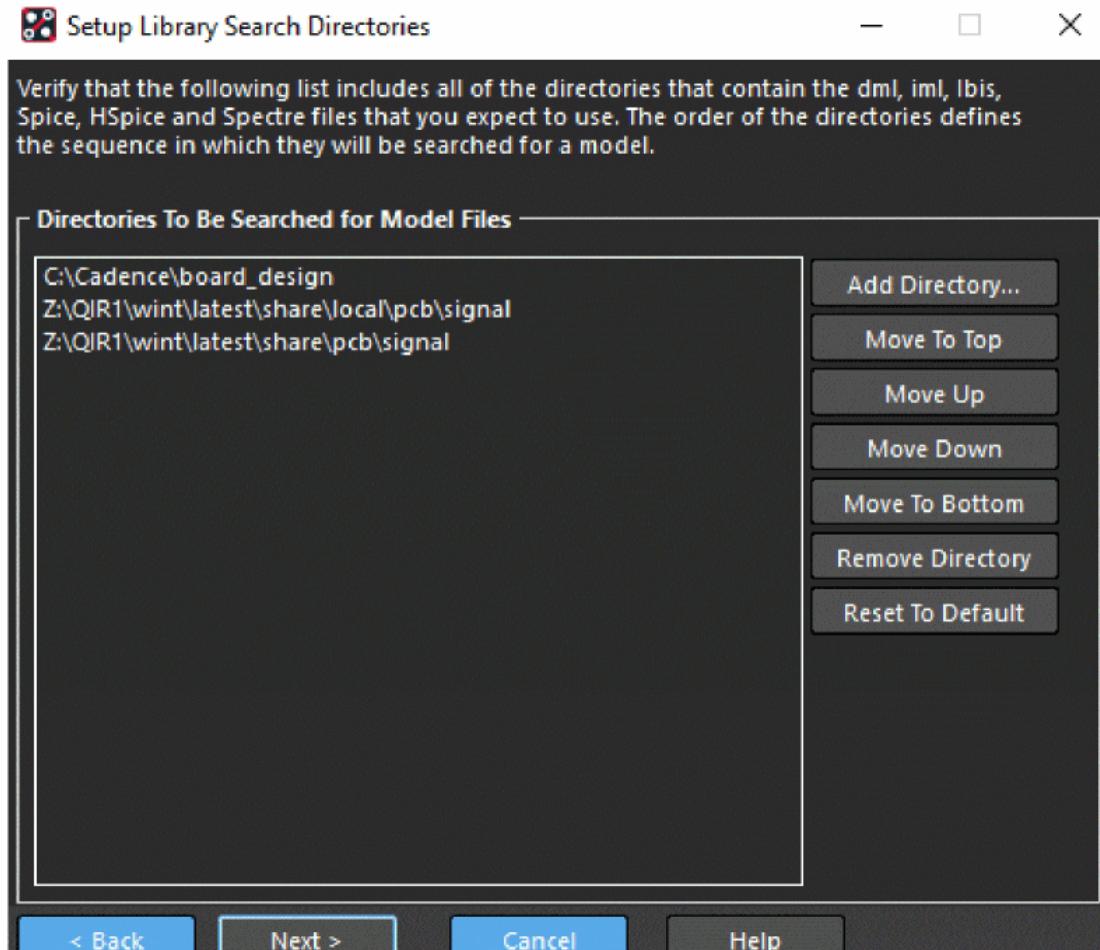
- Click **Next >** to run the setup operation on the specified Xnets and nets.

Setting Up Search Directories and File Extensions

The setup wizard further guides you through the steps to set up search directories, model library file extensions, and working libraries.

Library Search Directories

The Setup Library Search Directories dialog box prompts you to verify that all the directories containing the required model files are available for use. You can change the sequence in which the directories are searched to locate a model. You can also add a new directory to the list.

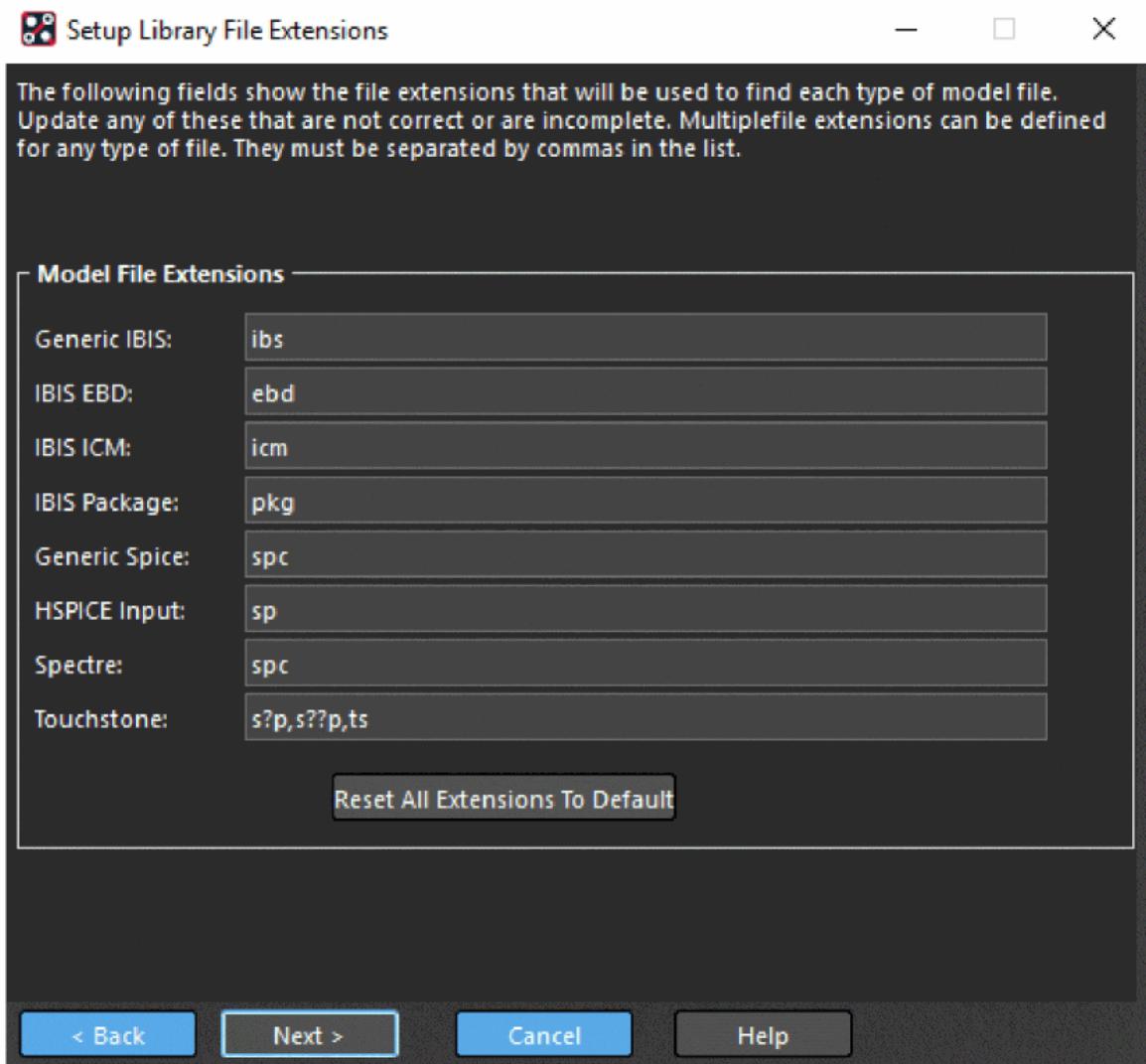


Library File Extensions

In the Setup Library File Extensions dialog box, you specify the file extension to be used for each type of model file. The default model file extensions are listed for each model type.

S Commands

S Commands--signal setup

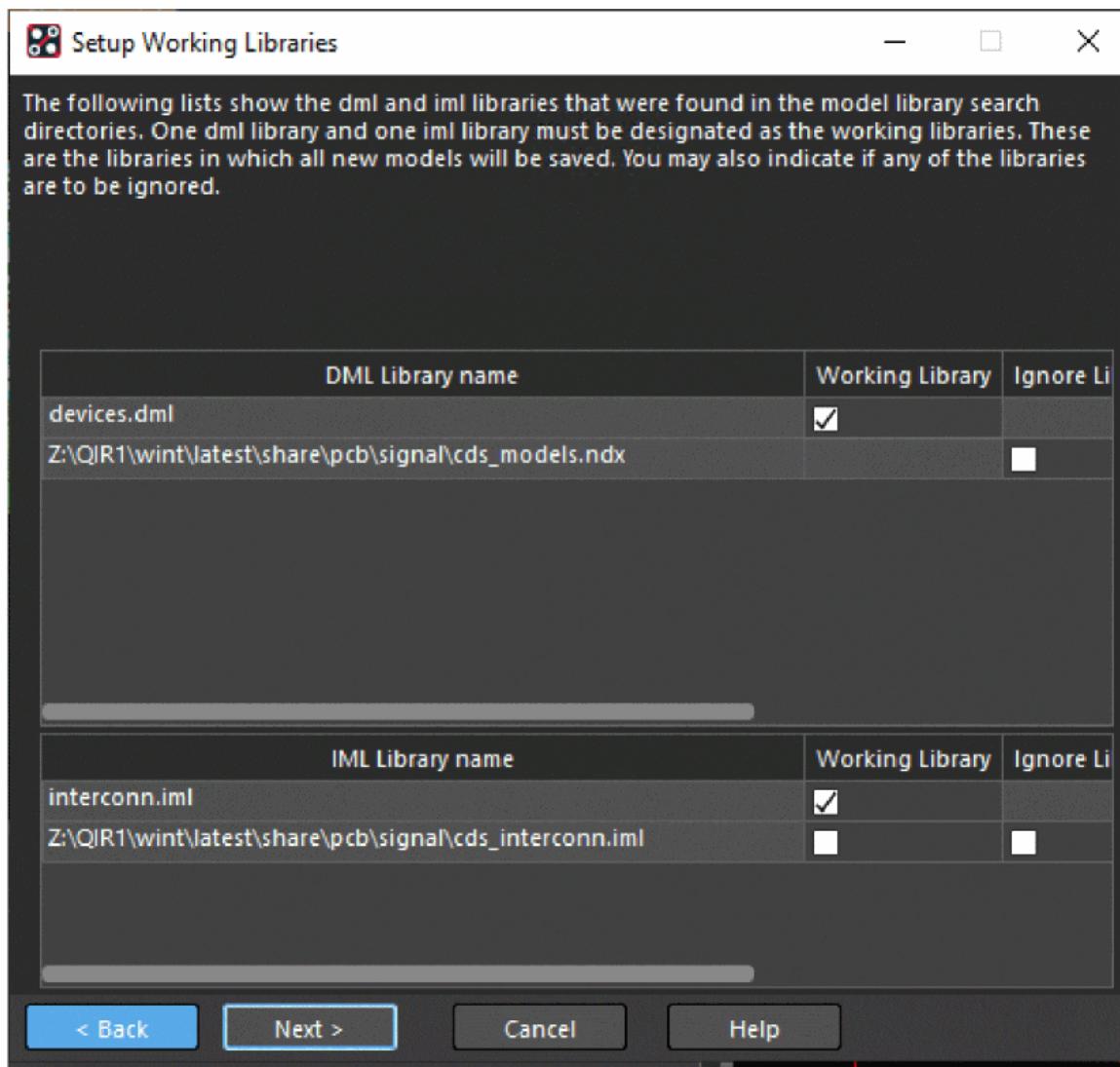


Working Libraries

The Setup Working Libraries dialog box displays the dml and iml libraries found in the specified search directories. Here you specify which libraries are to be used as working libraries where new models will be stored.

S Commands

S Commands--signal setup

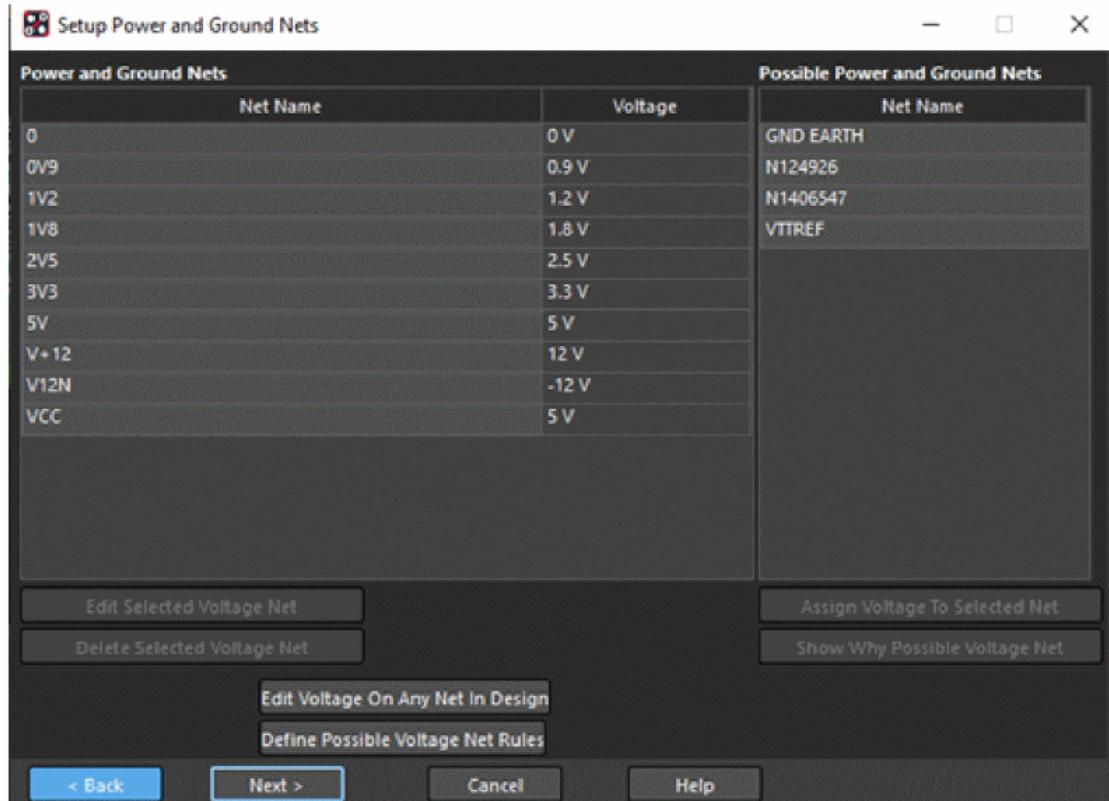


Performing Setup Operations on Selected Categories

For each setup category, one or more pages provide the options you need to specify to complete the setup operation.

Setting up Power and Ground Nets

The *Setup Power and Ground Nets* page is used for specifying setup settings for the Power and Ground Nets category.



This page of the wizard displays existing power and ground nets and also the possible power and ground nets that are not currently marked with the `VOLTAGE` property. You can assign a voltage to any of the possible DC nets or change the voltage that has been assigned to a DC net. You can see a net has been listed as a possible DC net.

On this page of SI Setup, you match nets to DC voltage levels. You can select the pins in the net as well as set the voltage source pins. You must identify one or more voltage source pins to perform EMI simulation. PCB SI needs source voltages for terminators and capacitors in order to build circuits that are electrically correct.

The signal models can contain data related to voltage tolerances. Simulations can be performed at these tolerance levels, but the simulator has no way of knowing what the terminator voltage value is. You must supply the DC voltage values.

Field	Description
Edit Selected Voltage Net	Lets you specify a new voltage for the selected net. Select a net from the <i>Power and Ground Nets</i> list and click this button. You can specify the new voltage value in the resultant text input box. Note that you can also specify a blank value for the voltage. ⚠ You can also click the <i>Voltage</i> field and directly change the value.
Delete Selected Voltage Net	Lets you delete an existing voltage net from the list.

S Commands

S Commands--signal setup

Assign Voltage to Selected Nets	Lets you assign voltage to selected nets from the <i>Possible Power and Ground Nets</i> list. The nets in the design which are potential candidates for power and ground nets as they fulfill one or more of criteria of net selection appear in the Possible Power and Ground Nets. You select a net from the list and click this button and specify a voltage value to the nets in the resultant text input box that appears. As you specify a voltage, the net name moves to the Power and Ground nets list.
Show Why possible Voltage Net	Shows which criteria the net fulfills to be a potential power/ground net. For example, if the rules say that if a net name contains the string SIG in its name, it qualifies to be a possible power/ground net candidate, the following message appears: <i>Net name contains the string "SIG".</i>
Edit Voltage On Any Net In Design	Opens the Identify DC Nets dialog to view and select nets to carry a DC voltage.
Define Possible Voltage Net Rules	Opens the <i>Possible Voltage Net</i> dialog where you can modify the default rules for net selection.

 The Design Engineer should apply the DC voltages in the schematic - the Signal Integrity Engineer should check them in the design using PCB SI.

The following default rules are used to select these nets:

1. A net that is part of a differential pair is not considered to be a voltage net.
2. A single pin net is not considered to be a voltage net.
3. If the name of the net contains any of the following strings, the net is considered as a possible voltage net: VCC, GND, VEE, VTT.
4. If the net contains any pins that have a POWER or GROUND pin use, the net is considered to be a possible voltage net.
5. If the net contains more than a specified number of pins, it is considered to be a possible voltage net. By default, this number is 25 but can be changed with the MAX_PINS_IN_NET environment variable.

You can exercise more control over these rules to find the possible power and ground nets using the *Possible Voltage Net Rules* dialog.

Using the various fields of this form you can set up your own list of strings, which will be matched against net names to find possible voltage nets.

Field	Description
Voltage Net Name Formats	Displays a list of formats for potential voltage net names. As you add more formats, the list expands.
Add New Voltage Net Name Format	Use this option to add new strings to define the format of possible net names.
Delete Selected Voltage Net Name Format	Use this option to delete existing strings that define possible net name formats.
Include nets that contain power and ground pins	This option sets the corresponding rule on.
Include nets that contain more than n pins	This option lets you specify the minimum number of pins a possible voltage net must contain.

The data from this form will be saved with the active drawing as an invisible property on the design so they will be reused each time the drawing is opened.

After you specify the setup options for a category, click the *Next* button. The audit tests associated with the category are run. If any errors are

S Commands

S Commands--signal setup

found, the Audit Errors dialog is displayed. When you have resolved or ignored all the errors reported for the category, the setup dialog for the next setup category appears. The number of categories depends on the categories you selected on an first page of the Setup wizard.

Assigning Models to Components

The *Assign Models to Components* page enables you to assign a signal model to each component that you simulate. You can create a new model, a default model, or assign an existing model to a component.

Field	Description
Create Default Models for all Discretes	Assigns default models to <i>all the discrete</i> components and displays the results of assignments.
Create Default Model	Assigns a default model to the selected discrete component (s).
Assign Existing Model	Launches the Signal Model Assignment dialog box where you can use the <i>Auto Setup</i> option for all 2-pin components with a VALUE property and no previous model assignment.
Create New Model	Opens the Create IBIS Device Model dialog where you create a new model for the selected components.
Load Assignments From a File	Prompts you to select a <i>SIGNAL_MODEL Assignment map (.dat)</i> file. After you select the file, the assignments defined by the file are loaded. The <i>Assign Models to Components</i> page of the wizard is updated to remove all the components that now have assigned models.

Setting up Design Cross-Section

The *Setup Design Cross-Section* provides you with the option to update the cross-section of the design. Here, you define the type and characteristics of the varied material layers in the layout. You can manually edit existing cross-section, load cross-section from a technology file or from another design.

In this setup module, you define the layout cross-section. This information includes the material for the layer, as well as the type, name, thickness, line width, and impedance information.

Layout Cross-Section: The layout cross-section defines the physical and electrical characteristics of the printed circuit board. When you receive your board file from the PCB designer, the cross-section should exist. As the signal integrity engineer, your job is to verify that the cross-section meets the electrical requirements for impedance.

The impedance of the traces depends on:

- the dielectric constant value of the insulating material.
- the thickness of the insulating material.
- the thickness and width of the traces.

Use the [Cross-Section Editor Dialog Box](#) to view and alter the characteristics of a selected board layer. You can view and edit the layout cross-section. The cross-section consists of the ordered layers of your board, including information about their type, thickness, spacing, electrical characteristics, and differential impedance.

Setting up Component Classes

The *Setup Component Classes* page enables you to classify components as one of the three component device classes, IC, Discrete, or IO. You can also update the class of a selected component, if required. Selecting *All Classes* will select all the visible classes and components.

 PCB SI uses the Device Class to determine part type. The IC class identifies an active component, such as a driver or receiver. The DISCRETE class identifies passive components such as resistors, inductors, and capacitors. The IO class identifies input and output devices, such as connectors.

Setting Up SI Simulations

The *Setup SI Simulations* pages lets you specify the simulations to be performed and the simulators to be used.

Completing the Setup

After all the selected setup categories are processed, the final page of the setup wizard appears. From this page, you can run an audit on the entire design. Choose the *Run SI Design Audit* button to run the audit tests for all the setup categories that you selected.

Setting Up Differential Pairs

The *Setup Diff Pairs* page displays a list of user-defined and model-defined differential pairs for the selected nets or Xnets. You can create a new differential pair or delete an existing one.

You can perform the following actions:

- Create default differential pairs from the selected nets or Xnets
- Create Diff pairs from User-Defined Rules
- Create a New Diff Pair
- Edit a Model-Defined Diff Pair
- Change Diff Pair to be Defined by a Model
- Delete selected differential pair

If existing differential pairs do not appear on the *Setup Diff Pairs* page, check whether they are set to *IGNORED* in SI Design Audit. In the [Audit Errors](#) form, do the following steps:

1. In the *Status Filter* field, choose *Ignored*.
2. Right-click the ignored differential pairs, choose *Reset to Unresolved* in the pop-up menu and click *OK*.
3. Save the design.
4. Choose *Setup – SI Design Setup*.

The differential pairs appear on the *Setup Diff Pairs* page.

Related Topics

- [Create Diff pairs from User-Defined Rules](#)
- [Create a New Diff Pair](#)
- [Edit a Model-Defined Diff Pair](#)
- [Change Diff Pair to be Defined by a Model](#)

Create Diff pairs from User-Defined Rules

This dialog helps you create differential pair from pairs of nets or Xnets which contain the same base name and a suffix specified in this dialog box. The prefix that you need to specify in this dialog box prefixed to the names of the newly created user-defined differential pairs.

Therefore, the name of each differential pair is the specified differential pair name prefix string followed by the base name of the Xnets. These differential pairs appear in the *Setup Diff Pairs* page of the setup wizard. You can then use the *Edit a Model-Defined Diff Pair* button on to convert these differential pairs from user-defined to model-defined.

Handling the Bus Bit Format

This functionality handles Xnet/net names that use the bus bit format (names ending with a `<#>`). For example, a name of `DATA<2>` indicates bit two in a bus named `DATA`. When this format is used, the differential pair suffixes that are specified in the above form will appear ahead of the bit number in the Xnet name. For example, there can be two Xnet names, `DATA_P<3>` and `DATA_N<3>`. If a differential pair Xnet prefix `DP_` and suffixes `_P` and `_N` are specified, a differential pair named `DP_DATA3` will be created from the two Xnets.

Related Topics

- [Edit a Model-Defined Diff Pair](#)

Create a New Diff Pair

Use this dialog box to create new model-defined differential pairs. You select the inverting and non-inverting nets/Xnets from the Select an Xnet dialog box. On the next page, you assign models to the differential pair pins and a model-defined differential pair is created.

Related Topics

- [Select an Xnet](#)

S Commands

S Commands--signal setup

Select an Xnet

Use this dialog box to select Xnets for inverting and non-inverting members.

Edit a Model-Defined Diff Pair

Use this dialog to modify the definition of a model-defined differential pair. You can perform the following operations on the selected differential pair:

- Swap pins in a model
- Swap nets assigned to pins
- Assign net to mate pin

Change Diff Pair to be Defined by a Model

The option to access this dialog box is available when you have the *Show User Defined Diff Pairs* options button selected on the main page. The option to convert a user-defined differential pair to a model-defined differential pair is enabled when you select a user-defined differential pair.

As a first step, you need to verify the polarity of the Xnet members. If required, you can swap the polarity of Xnet members by clicking the *Swap Polarity of Diff Pair Member Xnets* button.

On the next page you assign differential pair models to the selected pin by clicking the *Assign DiffPair Model To Selected Pin* button. When a diffpair model is assigned to a pin, the row representing it appears with a red highlight. Click *Finish* to go back to the main page of the wizard. A message box prompts to confirm the creation of the model-defined differential pair and it starts appearing under the *Show Model Defined Diff Pairs* list.

Setting up Design to Perform SI Simulations

Follow these steps to set up your design to perform SI simulations:

1. Run signal setup.
The SI Design Setup wizard is displayed.
2. Follow the instructions on each page of the wizard.

Related Topics

- [signal setup](#)
- [SI Design Setup Wizard](#)

setwindow

The `setwindow` command assigns the keyboard/script focus to the window you have identified. ("Focus" means the window receiving your keystrokes and mouse clicks.) If you execute the command without naming a window, `setwindow` displays the name of the active window. This command is typically used in scripts to direct commands to a window.

Syntax

```
setwindow [<window name>]
```

Example

```
setwindow browser
```

shadow

The `shadow` command (also called Shadow Mode) lets you control the visibility of individual design elements without affecting the visibility settings of that element's entire subclass. The `shadow` command is used in conjunction with the `hilight` and `dehilight` commands, as well as various interactive commands.

When you run `shadow` (or turn on Shadow Mode in the Color dialog box), the following conditions occur:

- The Brightness setting slide bar moves to its last applied percentage of brightness. The initial default percentage setting is 40%.
- The colors in the design dim to the chosen percentage of brightness in the slide bar when you click *Apply*, allowing you to preview how the colors in your design will be displayed.
- A *Dim active layer* check box lets you dim the active layer of your design. Dimming the active if it contains a large number of elements displayed normally (non-highlighted) can increase the effectiveness of Shadow Mode. You can dim the active layer by way of the check box in the Color dialog box or in the *Options* panel when shadow mode is turned on.
- The design elements of the current active drawing dim to the percentage of brightness set in the slide bar

With Shadow Mode active, elements in your design can be displayed in the following ways:

- Normal.
Objects on the active layer of your design remain unaffected by Shadow Mode unless you choose the *Dim active layer* control in the *Options* panel.
- Highlighted
Either permanently by way of the `hilight` command, or temporarily when you run an interactive command. In this state, elements are not affected by Shadow Mode. Objects affected or added by a current interactive command are temporarily highlighted while the command is active. For example, if you run `add connect` with Shadow Mode on, the elements highlighted would include:
 - Interconnecting pins
 - Existing etch being tied into
 - Connect lines, vias, and DRCs

When you complete the command, the added/affected elements are dimmed.

- Dim. The elements unaffected by the conditions described above. The degree of dimming depends on the percentage of brightness set in the Color dialog box.

Syntax

You can run `shadow` from the command window prompt, as well as from the Color dialog box.

The syntax for setting `shadow` at the command prompt is

```
shadow [on] [off] <+/-n>
```

Examples

shadow on	Turns Shadow Mode on
shadow 50	Turns Shadow Mode on and sets the brightness factor to 50%
shadow +10	Increases the current brightness factor by 10%
shadow off	Turns Shadow Mode off
shadow toggle	Toggles Shadow Mode off and on

You can set global Shadow Mode parameters through the use of keyboard commands entered at the command prompt, allowing you to assign function keys or toolbars to the dimming controls.

Example:

To toggle shadow mode on and off using the F3 key, you would enter the following at the command window prompt:

```
alias SF3 shadow toggle
```

Related Topics

- [Setting Shadow Mode from the Console Window Prompt](#)
- [hilight](#)
- [dehilight](#)

Color Dialog Box

Access Using

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

Shadow Mode controls are in the Display area of the Color dialog box:

<i>On/Off</i>	Controls the activity and settings of shadow mode
<i>Brightness setting slide bar</i>	Moves to its last applied percentage of brightness. The initial default percentage setting is 40%.
<i>Dim active layer</i>	Lets you dim the active layer of your design. Dimming the active layer if it contains a large number of elements displayed normally (non-highlighted) can increase the effectiveness of Shadow Mode. You can dim the active layer by way of the check box in the Color dialog box or in the <i>Options</i> panel when shadow mode is turned on.

Setting Shadow Mode from the Console Window Prompt

- Type `shadow` and the appropriate arguments and values, as described in the Syntax section.

To set shadow mode from the Color dialog box:

1. Run the `color192` command.
The Color dialog box appears.
2. Choose *Display*.
3. Click *Shadow Mode On*.
The slide bar moves to its last applied percentage of brightness. If Shadow Mode has never been used, the initial default percentage setting is 50%.
4. Set the brightness level to the desired percentage, if different.
The colors in the Color section dim to the chosen percentage of brightness in the slide bar. This allows you to preview how the colors in your design will be displayed.
5. Check *Dim Active Layer* if you want to dim the active layer of your design.
6. When satisfied with your settings, click *Apply* or *OK*.
The design elements of the current active drawing dim to the percentage of brightness set in the slide bar, and a *Dim Active Layer* check-box is displayed in the *Options* panel.

Related Topics

- [shadow](#)

shadow toggle

The `shadow toggle` command lets you turn off or on any settings you configured using Shadow Mode. See [shadow](#) for a complete description.

shape

An internal Cadence engineering command.

shape add

The `shape add` command adds a multi-sided enclosed polygon and creates a static, dynamic, unfilled, or cross-hatched shape, which may be used for a placebound, route keepout, or a board outline. (Dynamic shapes can only be added to ETCH/CONDUCTOR layers.)

The *Options* panel controls many physical options pertinent to a shape, including the electrical subclass layer on which it resides, which can be chosen prior to the first instantiated pick or at any time during shape creation. Color swatches appear in the subclass section in the *Options* panel that align with the ETCH/CONDUCTOR color on that particular subclass layer. Since shape grids tend to be more coarse than routing grids, a separate shape grid on the *Options* panel saves time toggling to *Setup – Grids* or right-clicking and choosing *Quick Utilities – Grids*.

When entering a polygon, an extra dynamic line displays from the last end point to the starting point of the polygon, maintaining a closed polygon image at all times. This dynamic line adheres to the current *Segment Type* set in the *Options* panel and appears in orange. Double or right-clicking and choosing *Done* from the pop-up menu completes the boundary and fills the shape to its respective parameter settings. If adding a dynamic shape, the boundary appears in the color specified as the boundary color class in the *Display – Color Visibility – Stackup* group, but the shape fill color overrides it when the shape is completely drawn. For example, a shape that has its fill color as blue and boundary as red appears as solid blue if the fill overlays the boundary. If a voided area overlays part of the boundary, it appears as the boundary subclass color (red).

 Connecting etch lines can run parallel to shape boundaries. Segments will be in violation as they exit a shape vicinity if they are within the line-shape same-net spacing distance and if they turn back on the shape, that is, the vertex farthest from the connection point is closer to the shape than the vertex nearer to the connection point.

For additional related information on working with shapes, see the *Preparing the Layout* user guide or *Best Practices: Working with Shapes* in your documentation set.

Related Topics

- [Adding a Dynamic Copper Fill Polygon Shape](#)
- [Adding a Static Polygon Shape](#)

Shape Add Command: Options Panel

Access Using

- Menu Path: *Shape – Polygon*



- Toolbar Icon:

Active Class and Subclass	Choose the proper etch layer upon which to draw the polygon. Color boxes in the subclass section align with the etch color on that particular subclass layer.
Shape fill	Specify a shape fill type. Changing fill type affects the shape you are currently adding immediately. If the shape boundary exists, the shape updates dynamically. If the active layer restricts shapes to unfilled type, <i>Unfilled</i> appears here, and the field is disabled.
Type	<i>Dynamic Copper</i> : Choose to create a positive shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary. You can only add a dynamic shape to the etch class. <i>Dynamic Crosshatch</i> : Choose to create a dynamic crosshatch-filled shape whose copper area and voids are dynamically filled or updated after you edit its elements or boundary. <i>Static Solid</i> : Choose to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary. A solid-fill shape is filled with a stencil pattern, which is transparent to allow drawing elements behind the shape to display. Use static positive shapes for handcrafting critical etch as shapes that you do not want modified automatically. <i>Static Crosshatch</i> : Choose to create a static crosshatch-filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary. <i>Unfilled</i> : Choose to create a static unfilled shape. You cannot add an unfilled shape on an etch layer.
Defer performing dynamic fill	Choose to prevent the shape you are currently adding from dynamically updating. Disabling this field causes the shape you are currently adding to be dynamically updated and filled if the shape boundary exists. This field is disabled for unfilled and nonetch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <i>Esc</i> key to stop the process.
Assign net name	Enter a net to assign to the shape. Choose a net from the dropdown list, or click ... to display the <i>Select Net</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using <i>Logic – Identify DC Nets</i> (<i>identify nets</i> command). Changing an assigned net dynamically fills and updates the shape. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current shape, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
Shape grid	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by choosing <i>Setup – Grids</i> or right-clicking and choosing <i>Quick Utilities – Grids</i> . If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None</i> : Choose to create shapes off grid in user units, specified on the <i>Design</i> tab of the <i>Design Parameter Editor</i> (<i>prmed</i> command). <i>Current Grid</i> : Choose to use the predefined system grid values for the active class/subclass. This is the default value.
Segment type	The following line segment types are available:
Type	<i>Line</i> : Choose to use any angle line. <i>Line 45</i> : Choose to miter lines to a 45 degree at vertex locations. <i>Line Orthogonal</i> : Choose to create lines at 90 degree angles at vertex locations. <i>Arc</i> : Choose to create an arc. Available only when adding polygons. Once you enter an arc, this field automatically defaults to the previous line segment type specified in the <i>Type</i> field. Cursor position as it moves toward the arc end point determines arc direction (clockwise or counter clockwise).
Angle	Available only if you specified <i>Arc</i> as the line segment type in the <i>Type</i> field as an alternative to selecting the end point of the arc. Enter a value to create an arc from the start point with the specified angle. The arc is tangent to the start and end point, which determines the arc's direction.
Arc radius	Available only if you specified <i>Arc</i> as the line segment type in the <i>Type</i> field. Enter the next arc segment with a given radius. A zero value creates a tangent arc.

Related Topics

- [Adding a Static Polygon Shape](#)
- [identify nets](#)
- [prmed](#)

Adding a Dynamic Copper Fill Polygon Shape

Follow these steps to add a dynamic copper fill polygon shape:

1. Choose *Setup – Constraints – Spacing* (`cmgr spac` command), then select *Shape* to specify spacing rules for shapes in Constraint Manager.
2. If required, assign element-level parameter properties using *Edit – Properties* (`property edit` command) to shapes, pins, vias, or clines to override parameters on the Global Dynamic Shape Parameters and Shape Instance Parameters dialog boxes.
3. Choose *Shape – Global Dynamic Params* (`shape global param` command) to display the *Shape Fill* tab of the *Global Dynamic Shape Parameters* dialog box. Choose *Smooth*, *Fast* or *Disabled* as the global value for the *Dynamic Fill* copper fill mode to be applied to all subsequent dynamic copper fill shapes you create.
4. On the *Void controls* tab, specify the global values for *Artwork Format* and *Minimum aperture for artwork fill* (depending on whether you chose a raster or vector artwork format) to be applied to all subsequent dynamic copper fill shapes you create.
5. Specify the global values for fields on the *Clearances* and the *Thermal relief connects* tabs to be applied to all subsequent dynamic copper fill shapes you create.
6. Choose *Shape – Polygon* (`shape add` command).
7. Verify the active class and subclass are correct.
8. Begin drawing the shape.
9. On the *Options* panel, specify the *Shape Fill Type* as *Dynamic Copper* to add a positive polygon shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary.

⚠️ You can defer the update of a dynamic copper fill shape by choosing the *Defer Performing Dynamic Fill* field on the *Options* panel. You can thereby edit the boundaries of a single shape without impacting performance and prevent other shapes from becoming out of date.

10. Right-click to display the pop-up menu and choose *Parameters* to display the *Shape Instance Parameters* dialog box, to specify a *solid* or *xhatch Fill Style* on the *Shape Fill* tab for the dynamic copper fill shape you are adding.
11. Attach the shape to a net by specifying a net name in the *Assign Net Name* field, choosing a net name from the dropdown list, right-clicking to display the pop-up menu and choosing *Assign Nets*, or clicking... to display the *Select Nets* dialog box from which you can choose a net. This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
12. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field (optional).
13. Left click at the vertices of the shape outline that you want to create. Complete the shape boundary by using the left mouse to click near the first pick, or by using the right to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up to the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.

Parameters	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.
-------------------	--

14. To interactively add or edit user-defined manual voids to the shape, use commands available from the *Shape – Manual Void/Cavity* menu ([shape void polygon](#), [shape void circle](#), [shape void rectangle](#), [shape void copy](#), [shape void move](#), or [shape void delete](#) commands). You must use *Shape – Select Shape or Void/Cavity* ([shape select](#) command) to choose the void before you can edit it.

Related Topics

- [shape add](#)

Adding a Static Polygon Shape

Follow these steps to add a static polygon shape:

1. Choose *Setup – Constraints – Spacing* (`cmgr spac` command), then select *Shape* to specify spacing rules for shapes in Constraint Manager.
2. If required, assign element-level parameter properties using *Edit – Properties* (`property edit` command) to shapes, pins, vias, or clines to override parameters you defined on the Static Shape Parameters dialog box.
3. Choose *Shape – Polygon* (`shape add` command).
4. Verify the active class and subclass are correct.
5. On the *Options* panel, specify the *Shape Fill Type*:
 - o Choose *Static Solid* to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
or
 - o Choose *Static Crosshatch* to create a static crosshatch filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
6. Attach the shape to a net by specifying a net name in the *Assign Net Name* field, choosing a net name from the dropdown list, right-clicking to display the pop-up menu and choosing *Assign Nets*, or clicking... to display the Select Nets dialog box from which you can choose a net. This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
7. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field (optional).
8. Left click at the vertices of the shape outline that you want to create. Complete the shape boundary by using the left mouse to click near the first pick, or by using the right to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

9. To interactively add or edit user-defined manual voids to the shape, use commands available from the *Shape – Manual – Void/Cavity* menu (`shape void polygon`, `shape void circle`, `shape void rectangle`, `shape void copy`, `shape void move`, or `shape void delete` commands). You must use *Shape – Select Shape or Void/Cavity* (`shape select` command) to choose the void before you can edit it.

Related Topics

- [shape add](#)
- [Shape Add Command: Options Panel](#)

shape add circle

The `shape add circle` command adds a circular shape. When you add a dynamic etch shape that crosses the route keepin, by default the layout editor clips the shape to the route keepin. To prevent the layout editor from clipping a dynamic shape that is completely outside the route keepin, enable the `shape_noclip_rki` board level environment variable in the *User Preferences* dialog box, available by running the `enved` command. DRCs then occur as a result.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Adding a Dynamic Copper Fill Circular Shape](#)
- [Adding a Static Circular Shape](#)

Shape Add Circle Command: Options Panel

Access Using

- Menu Path: *Shape – Circular*
- Toolbar Icon: 

Active Class and Subclass	Choose the proper etch layer upon which to draw the shape. Color boxes in the subclass section align with the etch color on that particular subclass layer.
Shape fill	Specify a shape fill type. Changing fill type affects the shape you are currently adding immediately. If the shape boundary exists, the shape updates dynamically. If the active layer restricts shapes to unfilled type, <i>Unfilled</i> appears here, and the field is disabled.
Type	<i>Dynamic Copper</i> : Choose to create a positive shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary. You can only add a dynamic shape to the etch class. <i>Dynamic Crosshatch</i> : Choose to create a dynamic crosshatch-filled shape whose copper area and voids are dynamically filled or updated after you edit its elements or boundary. <i>Static Solid</i> : Choose to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary. A solid-fill shape is filled with a stencil pattern, which is transparent to allow drawing elements behind the shape to display. Use static positive shapes for handcrafting critical etch as shapes that you do not want modified automatically. <i>Static Crosshatch</i> : Choose to create a static crosshatch-filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary. <i>Unfilled</i> : Choose to create a static unfilled shape. You cannot add an unfilled shape on an etch layer.
Defer performing dynamic fill	Choose to prevent the shape you are currently adding from dynamically updating. Disabling this field causes the shape you are currently adding to be dynamically updated and filled if the shape boundary exists. This field is disabled for unfilled and nonetch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <i>Esc</i> key to stop the process.
Assign net name	Enter a net to assign to the shape, choose a net from the dropdown list, or click ... to display the <i>Select Net</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using <i>Logic – Identify DC Nets</i> (Identify nets command). Changing an assigned net dynamically fills and updates the shape. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current shape, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
Shape grid	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by running the define grid command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None</i> : Choose to create shapes off grid in user units, specified on the <i>Drawing Parameters</i> dialog box (prmed command). <i>Current Subclass Grid</i> : Choose to use the predefined system grid values for the active class/subclass. This is the default value.
Circular Shape Creation	Choose to set circular shape creation mode. <i>Draw Circle</i> : Choose to draw circular shape. This option is selected by default. <i>Place Circle</i> : Choose to place the circular shape of known radius by setting the <i>Radius</i> value. <i>Center/Radius</i> : Choose to create the circular shape with known center and radius by setting the <i>Radius</i> and <i>Center</i> values in the field below. <i>Create</i> : Click to create the circular shape. <i>Radius</i> : Set radius of the circular shape. <i>Center</i> : Set the origin of the circular shape.

Related Topics

- [Adding a Static Circular Shape](#)

Adding a Dynamic Copper Fill Circular Shape

- [shape add circle](#)

Adding a Static Circular Shape

To add a static circular shape, perform these steps:

1. Choose *Setup – Constraints – Spacing* (`cmgr spac` command), then select *Shape* to specify spacing rules for shapes in Constraint Manager.
2. If required, assign element-level parameter properties using *Edit – Properties* (`property edit` command) to shapes, pins, vias, or clines to override parameters you defined on the *Static Shape Parameters* dialog box.
3. Choose *Shape – Circular* (`shape add circle` command).
4. Verify the active class and subclass are correct.
5. On the *Options* panel, specify the *Shape Fill Type*:
 - o Choose *Static Solid* to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
or
 - o Choose *Static Crosshatch* to create a static crosshatch filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
6. Attach the shape to a net by choosing a net name from the dropdown list, or clicking... to display the *Select Nets* dialog box from which you can choose a net.
This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
7. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field (optional).
8. Choose options in the *Circular Shape Creation* to create the circular shape.

Option	Description
<i>Draw Circle</i>	<ol style="list-style-type: none"> a. Specify the center of circular shape by moving the cursor to the position where you want to be the circle center, and left click. The coordinates of the center are updated in the <i>Options</i> panel. b. Specify the radius of circular shape by moving cursor to the position and left click. The value of the radius of the circular shape is updated in the <i>Options</i> panel.
<i>Place Circle</i>	<ol style="list-style-type: none"> a. Specify the radius of circular shape in the <i>Radius</i> field in the <i>Options</i> panel. b. The circular shape is attached to the cursor. c. Left click to place the circular shape. The coordinates of the center are updated in the <i>Options</i> panel.
<i>Center/Radius</i>	<ol style="list-style-type: none"> a. Specify the center of circular shape in the <i>Center</i> field in the <i>Options</i> panel. You can also specify the center by moving the cursor you want to be the circle center, and left click. The coordinates of the center are updated in the <i>Options</i> panel. b. Specify the radius of circular shape in the <i>Radius</i> field in the <i>Options</i> panel or move the cursor to the position, and left click. The value of the radius of the circular shape is updated in the <i>Options</i> panel. c. Choose <i>Create</i> to add the circular shape with specified radius.

Right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.

<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

⚠ To interactively add or edit user-defined manual voids to the shape, use commands available from the *Shape – Manual – Void/Cavity* menu ([shape void polygon](#), [shape void circle](#), [shape void rectangle](#), [shape void copy](#), [shape void move](#), or [shape void delete](#) commands). You must use *Shape – Select Shape or Void/Cavity* ([shape select](#) command) to choose the void before you can edit it.

Related Topics

- [shape add circle](#)
- [Shape Add Circle Command: Options Panel](#)

shape add rect

The `shape add rect` command Adds a rectangular shape. When you add a dynamic etch shape that crosses the route keepin, by default the layout editor clips the shape to the route keepin. To prevent the layout editor from clipping a dynamic shape that is completely outside the route keepin, enable the `shape_noclip_rki` board level environment variable in the *User Preferences* dialog box, available by running the `enved` command. DRCs then occur as a result.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Adding a Dynamic Copper Fill Rectangular Shape](#)
- [Adding a Static Rectangular Shape](#)

Shape Add Rect Command: Options Panel

Access Using

- Menu Path: *Shape – Rectangular*
- Toolbar Icon: 

<i>Active Class and Subclass</i>	Choose the proper etch layer upon which to draw the shape. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Shape fill</i>	Specify a shape fill type. Changing fill type affects the shape you are currently adding immediately. If the shape boundary exists, the shape updates dynamically. If the active layer restricts shapes to unfilled type, <i>Unfilled</i> appears here, and the field is disabled.
<i>Type</i>	<i>Dynamic Copper:</i> Choose to create a positive shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary. You can only add a dynamic shape to the etch class. <i>Dynamic Crosshatch:</i> Choose to create a dynamic crosshatch-filled shape whose copper area and voids are dynamically filled or updated after you edit its elements or boundary. <i>Static Solid:</i> Choose to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary. A solid-fill shape is filled with a stencil pattern, which is transparent to allow drawing elements behind the shape to display. Use static positive shapes for handcrafting critical etch as shapes that you do not want modified automatically. <i>Static Crosshatch:</i> Choose to create a static crosshatch-filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary. <i>Unfilled</i> Choose to create a static unfilled shape. You cannot add an unfilled shape on an etch layer.
<i>Defer performing dynamic fill</i>	Choose to prevent the shape you are currently adding from dynamically updating. Disabling this field causes the shape you are currently adding to be dynamically updated and filled if the shape boundary exists. This field is disabled for unfilled and non etch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <code>Esc</code> key to stop the process.
<i>Assign net name</i>	Enter a net to assign to the shape, choose a net from the dropdown list, or click ... to display the Select Net dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using <i>Logic – Identify DC Nets (identify nets)</i> command. Changing an assigned net dynamically fills and updates the shape. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current shape, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
<i>Shape grid</i>	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the Grids Display dialog box, available by running the <code>define grid</code> command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None:</i> Choose to create shapes off grid in user units, specified on the <i>Drawing Parameters</i> dialog box (see Creating Drawing Parameters). <i>Current Subclass Grid:</i> Choose to use the predefined system grid values for the active class/subclass. This is the default value.
<i>Shape Creation</i>	Choose to sets shape creation mode. <i>Draw Rectangle:</i> Choose to draw rectangular shape. This is the default value. <i>Place Rectangle:</i> Choose to place the rectangular shape of known size by setting the <i>Height</i> and <i>Width</i> fields.
<i>Corners</i>	Choose to sets type of corners for the shape. <i>Orthogonal:</i> Choose to set the corner type as orthogonal. This is the default value. <i>Chamfer:</i> Choose to set the corner type as chamfer. <i>Round:</i> Choose set the corner type as round. You can set the chamfer and round corner parameters in two ways: <i>Explicit Length:</i> Choose to control the corner length and radius. <i>% of Short Edge:</i> Choose to set the trim size as percent of short edge.

Related Topics

- [Adding a Static Rectangular Shape](#)

Adding a Dynamic Copper Fill Rectangular Shape

To add a rectangular shape with dynamic copper fill, perform these steps:

1. Choose *Setup – Constraints – Spacing* (`cmgr spac` command), then select *Shape* to specify spacing rules for shapes in Constraint Manager.
2. If required, assign element-level parameter properties using *Edit – Properties* (`property edit` command) to shapes, pins, vias, or clines to override parameters on the *Global Dynamic Shape Parameters* and *Shape Instance Parameters* dialog boxes.
3. Choose *Shape – Global Dynamic Params* (`shape global param` command) to display the *Shape Fill* tab of the *Global Dynamic Shape Parameters* dialog box. Choose *Smooth*, *Fast*, or *Disabled* as the global value for the *Dynamic Fill* copper fill mode to be applied to all subsequent dynamic copper fill shapes you create.
4. On the *Void Controls* tab, specify the global values for *Artwork Format* and *Minimum gap width or aperture for artwork fill (depending on whether you chose a raster or vector artwork format)* to be applied to all subsequent dynamic copper fill shapes you create.
5. Specify the global values for fields on the *Clearances* and the *Thermal Relief Connects* tabs to be applied to all subsequent dynamic copper fill shapes you create.
6. Choose *Shape – Rectangular* (`shape add rect` command).
7. Verify the active class and subclass are correct.
8. Begin drawing the shape.
9. On the *Options* panel, specify the *Shape Fill Type* as *Dynamic Copper* to add a positive rectangular shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary.

⚠️ You can defer the update of a dynamic copper fill shape by choosing the *Defer Performing Dynamic Fill* field on the *Options* panel. You can thereby edit the boundaries of a single shape without impacting performance and prevent other shapes from becoming out of date.

10. Right-click to display the pop-up menu and choose *Parameters* to display the *Shape Instance Parameters* dialog box, to specify a *solid* or *xhatch Fill Style* on the *Shape Fill* tab for the dynamic copper fill shape you are adding.
11. Attach the shape to a net by specifying a net name in the *Assign Net Name* field, choosing a net name from the dropdown list, right-clicking to display the pop-up menu and choosing *Assign Nets*, or clicking... to display the *Select Nets* dialog box from which you can choose a net. This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
12. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field (optional).
13. Select shape creation mode.
14. Select shape corner type from *Corners* field.
15. Left click and drag the cursor until the rectangle shape is the correct size; then right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up to the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.

Assign Region	To attach the shape to a region.
Arc	To set the rubber band mode to arc.
Snap pick to	To snap your next mouse pick to the closest design element you choose from the sub-menu.
Parameters	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

 If the created shape is placed out of drawing extents this operation is cancelled. A warning message is displayed in the command window.

W- (SPMHA2-54): Cannot place outside of the drawing extents.

16. To interactively add or edit user-defined manual voids to the shape, use commands available from the *Shape – Manual – Void/Cavity* menu ([shape void polygon](#), [shape void circle](#), [shape void rectangle](#), [shape void copy](#), [shape void move](#), or [shape void delete](#) commands). You must use *Shape – Select Shape or Void/Cavity* ([shape select](#) command) to choose the void before you can edit it.

Related Topics

- [shape add rect](#)

Adding a Static Rectangular Shape

To add a static rectangular shape, perform these steps:

1. Choose *Setup – Constraints – Spacing* (`cmgr spac` command), then select *Shape* to specify spacing rules for shapes in Constraint Manager.
2. If required, assign element-level parameter properties using *Edit – Properties* (`property edit` command) to shapes, pins, vias, or clines to override parameters you defined on the *Static Shape Parameters* dialog box.
3. Choose *Shape – Rectangular* (`shape add rect` command).
4. Verify the active class and subclass are correct.
5. On the *Options* panel, specify the *Shape Fill Type*:
 - o Choose *Static Solid* to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
or
 - o Choose *Static Crosshatch* to create a static crosshatch filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
6. Attach the shape to a net by specifying a net name in the *Assign Net Name* field, choosing a net name from the dropdown list, right-clicking to display the pop-up menu and choosing *Assign Nets*, or clicking... to display the *Select Nets* dialog box from which you can choose a net. This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
7. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field.(optional).
8. Select shape creation mode.
9. Select shape corner type from *Corners* field.
10. Drag the cursor until the rectangle shape is the correct size; then right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

⚠ If the created shape is placed out of drawing extents this operation is cancelled. A warning message is displayed in the command window.

W- (SPMHA2-54): Cannot place outside of the drawing extents.

11. To interactively add or edit user-defined manual voids to the shape, use commands available from the *Shape – Manual – Void/Cavity* menu (`shape void polygon`, `shape void circle`, `shape void rectangle`, `shape void copy`, `shape void move`, or `shape void delete` commands).

You must use *Shape – Select Shape or Void/Cavity* ([shape select](#) command) to choose the void before you can edit it.

Related Topics

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

shape_app add

The `shape_app add` command adds a multi-sided enclosed polygon and creates a static, dynamic, unfilled, or cross-hatched shape, which may be used for a placebound, route keepout, or a board outline. (Dynamic shapes can only be added to ETCH/CONDUCTOR layers.)

The *Options* panel controls many physical options pertinent to a shape, including the electrical subclass layer on which it resides, which can be chosen prior to the first instantiated pick or at any time during shape creation. Color swatches appear in the subclass section in the *Options* panel that align with the ETCH/CONDUCTOR color on that particular subclass layer. Since shape grids tend to be more coarse than routing grids, a separate shape grid on the *Options* panel saves time toggling to *Setup – Grids* or right-clicking and choosing *Quick Utilities – Grids*.

When entering a polygon, an extra dynamic line displays from the last end point to the starting point of the polygon, maintaining a closed polygon image at all times. This dynamic line adheres to the current *Segment Type* set in the *Options* panel and appears in orange. Double or right-clicking and choosing *Done* from the pop-up menu completes the boundary and fills the shape to its respective parameter settings. If adding a dynamic shape, the boundary appears in the color specified as the boundary color class in the *Display – Color Visibility – Stackup* group, but the shape fill color overrides it when the shape is completely drawn. For example, a shape that has its fill color as blue and boundary as red appears as solid blue if the fill overlays the boundary. If a voided area overlays part of the boundary, it appears as the boundary subclass color (red).

Related Topics

- [Adding a Dynamic Copper Fill Polygon Shape to the Design](#)
- [Adding a Static Polygon Shape to the Design](#)

Shape_app Add Command: Options Panel

Active Class and Subclass	Choose the proper etch layer upon which to draw the polygon. Color boxes in the subclass section align with the etch color on that particular subclass layer.
Shape fill	Specify a shape fill type. Changing fill type affects the shape you are currently adding immediately. If the shape boundary exists, the shape updates dynamically. If the active layer restricts shapes to unfilled type, <i>Unfilled</i> appears here, and the field is disabled.
Type	<p><i>Dynamic Copper:</i> Choose to create a positive shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary. You can only add a dynamic shape to the etch class <i>Static Solid</i>:</p> <p>Choose to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary. A solid-fill shape is filled with a stencil pattern, which is transparent to allow drawing elements behind the shape to display. Use static positive shapes for handcrafting critical etch as shapes that you do not want modified automatically.</p> <p><i>Static Crosshatch:</i> Choose to create a static crosshatch-filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary. <i>Unfilled</i> Choose to create a static unfilled shape. You cannot add an unfilled shape on an etch layer.</p>
Defer performing dynamic fill	Choose to prevent the shape you are currently adding from dynamically updating. Disabling this field causes the shape you are currently adding to be dynamically updated and filled if the shape boundary exists. This field is disabled for unfilled and nonetch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <i>Esc</i> key to stop the process.
Assign net name	Enter a net to assign to the shape. Choose a net from the dropdown list, or click ... to display the <i>Select Net</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using <i>Logic – Identify DC Nets</i> (identify nets command). Changing an assigned net dynamically fills and updates the shape. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current shape, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
Shape grid	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by choosing <i>Setup – Grids</i> or right-clicking and choosing <i>Quick Utilities – Grids</i> . If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None:</i> Choose to create shapes off grid in user units, specified on the <i>Design</i> tab of the <i>Design Parameter Editor</i> (prmed command). <i>Current Grid:</i> Choose to use the predefined system grid values for the active class/subclass. This is the default value.
Segment type	The following line segment types are available:
Type	<p><i>Line:</i> Choose to use any angle line. <i>Line 45:</i> Choose to miter lines to a 45 degree at vertex locations. <i>Line Orthogonal:</i> Choose to create lines at 90 degree angles at vertex locations. <i>Arc:</i> Choose to create an arc. Available only when adding polygons. Once you enter an arc, this field automatically defaults to the previous line segment type specified in the <i>Type</i> field. Cursor position as it moves toward the arc end point determines arc direction (clockwise or counter clockwise).</p>
Angle	Available only if you specified <i>Arc</i> as the line segment type in the <i>Type</i> field as an alternative to selecting the end point of the arc. Enter a value to create an arc from the start point with the specified angle. The arc is tangent to the start and end point, which determines the arc's direction.
Arc radius	Available only if you specified <i>Arc</i> as the line segment type in the <i>Type</i> field. Enter the next arc segment with a given radius. A zero value creates a tangent arc.

Related Topics

- [Adding a Static Polygon Shape to the Design](#)

Adding a Dynamic Copper Fill Polygon Shape to the Design

To add a polygon shape with dynamic copper fill, perform these steps:

1. Run `shape_app add` command.
2. Verify the active class and subclass are correct.
3. Begin drawing the shape.
4. On the *Options* panel, specify the *Shape Fill Type* as *Dynamic Copper* to add a positive polygon shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary.

⚠️ You can defer the update of a dynamic copper fill shape by choosing the *Defer Performing Dynamic Fill* field on the *Options* panel. You can thereby edit the boundaries of a single shape without impacting performance and prevent other shapes from becoming out of date.

5. Right-click to display the pop-up menu and choose *Parameters* to display the *Shape Instance Parameters* dialog box, to specify a *solid* or *xhatch Fill Style* on the *Shape Fill* tab for the dynamic copper fill shape you are adding.
6. Attach the shape to a net by specifying a net name in the *Assign Net Name* field, choosing a net name from the dropdown list, right-clicking to display the pop-up menu and choosing *Assign Nets*, or clicking... to display the *Select Nets* dialog box from which you can choose a net. This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
7. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field (optional).
8. Left click at the vertices of the shape outline that you want to create. Complete the shape boundary by using the left mouse to click near the first pick, or by using the right to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up to the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

Related Topics

- [shape_app add](#)

Adding a Static Polygon Shape to the Design

To add a static polygon shape, perform these steps:

1. Run `shape_app add` command.
2. Verify the active class and subclass are correct.
3. On the *Options* panel, specify the *Shape Fill Type*:
 - o Choose *Static Solid* to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
or
 - o Choose *Static Crosshatch* to create a static crosshatch filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
4. Attach the shape to a net by specifying a net name in the *Assign Net Name* field, choosing a net name from the dropdown list, right-clicking to display the pop-up menu and choosing *Assign Nets*, or clicking... to display the Select Nets dialog box from which you can choose a net. This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
5. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field (optional).
6. Left click at the vertices of the shape outline that you want to create. Complete the shape boundary by using the left mouse to click near the first pick, or by using the right to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

Related Topics

- [shape_app add](#)
- [Shape_app Add Command: Options Panel](#)

shape_app add circle

The `shape_app add circle` command adds a circular shape. When you add a dynamic etch shape that crosses the route keepin, by default the layout editor clips the shape to the route keepin.

Related Topics

- [Adding a Dynamic Copper Fill Circular Shape to the Design](#)
- [Adding a Static Circular Shape to the Design](#)

Shape_app Add Circle Command: Options Panel

Active Class and Subclass	Choose the proper etch layer upon which to draw the shape. Color boxes in the subclass section align with the etch color on that particular subclass layer.
Shape fill	Specify a shape fill type. Changing fill type affects the shape you are currently adding immediately. If the shape boundary exists, the shape updates dynamically. If the active layer restricts shapes to unfilled type, <i>Unfilled</i> appears here, and the field is disabled.
Type	<p>Dynamic Copper: Choose to create a positive shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary. You can only add a dynamic shape to the etch class.</p> <p>Static Solid: Choose to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary. A solid-fill shape is filled with a stencil pattern, which is transparent to allow drawing elements behind the shape to display. Use static positive shapes for handcrafting critical etch as shapes that you do not want modified automatically.</p> <p>Static Crosshatch: Choose to create a static crosshatch-filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.</p> <p>Unfilled: Choose to create a static unfilled shape. You cannot add an unfilled shape on an etch layer.</p>
Defer performing dynamic fill	Choose to prevent the shape you are currently adding from dynamically updating. Disabling this field causes the shape you are currently adding to be dynamically updated and filled if the shape boundary exists. This field is disabled for unfilled and non-etch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <code>Esc</code> key to stop the process.
Assign net name	Enter a net to assign to the shape, choose a net from the dropdown list, or click ... to display the <i>Select Net</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using <i>Logic – Identify DC Nets</i> (identify nets command). Changing an assigned net dynamically fills and updates the shape. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current shape, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
Shape grid	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by running the define grid command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings.
Circular Shape Creation	<p>Draw Circle: Choose to draw circular shape. This option is selected by default.</p> <p>Place Circle: Choose to place the circular shape of known radius by setting the <i>Radius</i> value.</p> <p>Center/Radius: Choose to create the circular shape with known center and radius by setting the <i>Radius</i> and <i>Center</i> values in the field below.</p> <p>Create: Click to create the circular shape.</p> <p>Radius: Set radius of the circular shape.</p> <p>Center: Set the origin of the circular shape.</p>

Related Topics

- [Adding a Static Circular Shape to the Design](#)

Adding a Dynamic Copper Fill Circular Shape to the Design

To add a circular shape with dynamic copper fill, perform these steps:

1. Run `shape_app add circle` command.
2. Verify the active class and subclass are correct.
3. Begin drawing the shape.
4. On the *Options* panel, specify the *Shape Fill Type* as *Dynamic Copper* to add a positive circular shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary.

⚠️ You can defer the update of a dynamic copper fill shape by choosing the *Defer Performing Dynamic Fill* field on the *left* tab. You can thereby edit the boundaries of a single shape without impacting performance and prevent other shapes from becoming out of date.

5. Right-click to display the pop-up menu and choose *Parameters* to display the *Shape Instance Parameters* dialog box, to specify a *solid* or *xhatch Fill Style* on the *Shape Fill* tab for the dynamic copper fill shape you are adding.
6. Attach the shape to a net by choosing a net name from the dropdown list, or clicking... to display the *Select Nets* dialog box from which you can choose a net.
This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
7. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field (optional).
8. Choose options in the *Circular Shape Creation* to create the circular shape.

Option	Description
<i>Draw Circle</i>	<ol style="list-style-type: none">a. Specify the center of circular shape by moving the cursor to the position where you want to be the circle center, and left click. The coordinates of the center are updated in the <i>Options</i> panel.b. Specify the radius of circular shape by moving cursor to the position and left click. The value of the radius of the circular shape is updated in the <i>Options</i> panel.
<i>Place Circle</i>	<ol style="list-style-type: none">a. Specify the radius of circular shape in the <i>Radius</i> field in the <i>Options</i> panel.b. The circular shape is attached to the cursor.c. Left click to place the circular shape. The coordinates of the center are updated in the <i>Options</i> panel.
<i>Center/Radius</i>	<ol style="list-style-type: none">a. Specify the center of circular shape in the <i>Center</i> field in the <i>Options</i> panel. You can also specify the center by moving the cursor you want to be the circle center, and left click. The coordinates of the center are updated in the <i>Options</i> panel.b. Specify the radius of circular shape in the <i>Radius</i> field in the <i>Options</i> panel or move the cursor to the position, and left click. The value of the radius of the circular shape is updated in the <i>Options</i> panel.c. Choose <i>Create</i> to add the circular shape with specified radius.

Right-click to choose any of the following from the pop-up menu:

<i>Done</i>	To exit the command.
<i>Oops</i>	To back up to the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.

<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	to set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

Related Topics

- [shape_app add circle](#)

Adding a Static Circular Shape to the Design

To add a static circular shape, perform these steps:

1. Run `shape_app add circle` command.
2. Verify the active class and subclass are correct.
3. On the *Options* panel, specify the *Shape Fill Type*:
 - o Choose *Static Solid* to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
or
 - o Choose *Static Crosshatch* to create a static crosshatch filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
4. Attach the shape to a net by choosing a net name from the dropdown list, or clicking... to display the Select Nets dialog box from which you can choose a net.
This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
5. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field (optional).
6. Choose options in the *Circular Shape Creation* to create the circular shape.

Option	Description
<i>Draw Circle</i>	<ol style="list-style-type: none"> a. Specify the center of circular shape by moving the cursor to the position where you want to be the circle center, and left click. The coordinates of the center are updated in the <i>Options</i> panel. b. Specify the radius of circular shape by moving cursor to the position and left click. The value of the radius of the circular shape is updated in the <i>Options</i> panel.
<i>Place Circle</i>	<ol style="list-style-type: none"> a. Specify the radius of circular shape in the <i>Radius</i> field in the <i>Options</i> panel. b. The circular shape is attached to the cursor. c. Left click to place the circular shape. The coordinates of the center are updated in the <i>Options</i> panel.
<i>Center/Radius</i>	<ol style="list-style-type: none"> a. Specify the center of circular shape in the <i>Center</i> field in the <i>Options</i> panel. You can also specify the center by moving the cursor you want to be the circle center, and left click. The coordinates of the center are updated in the <i>Options</i> panel. b. Specify the radius of circular shape in the <i>Radius</i> field in the <i>Options</i> panel or move the cursor to the position, and left click. The value of the radius of the circular shape is updated in the <i>Options</i> panel. c. Choose <i>Create</i> to add the circular shape with specified radius.

Right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.

<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	<i>To snap your next mouse pick to the closest design element you choose from the sub-menu.</i>
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

Related Topics

- [shape_app add circle](#)
- [Shape_app Add Circle Command: Options Panel](#)

shape_app add rect

The `shape_app add rect` command adds a rectangular shape. When you add a dynamic etch shape that crosses the route keepin, by default the layout editor clips the shape to the route keepin.

Related Topics

- [Adding a Dynamic Copper Fill Rectangular Shape to the Design](#)
- [Adding a Static Rectangular Shape to the Design](#)
- [Test](#)

Shape_app Add Rect Command: Options Panel

Active Class and Subclass	Choose the proper etch layer upon which to draw the shape. Color boxes in the subclass section align with the etch color on that particular subclass layer.
Shape fill	Specify a shape fill type. Changing fill type affects the shape you are currently adding immediately. If the shape boundary exists, the shape updates dynamically. If the active layer restricts shapes to unfilled type, <i>Unfilled</i> appears here, and the field is disabled.
Type	<p><i>Dynamic Copper:</i> Choose to create a positive shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary. You can only add a dynamic shape to the etch class.</p> <p><i>Static Solid:</i> Choose to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary. A solid-fill shape is filled with a stencil pattern, which is transparent to allow drawing elements behind the shape to display. Use static positive shapes for handcrafting critical etch as shapes that you do not want modified automatically.</p> <p><i>Static Crosshatch:</i> Choose to create a static crosshatch-filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.</p> <p><i>Unfilled:</i> Choose to create a static unfilled shape. You cannot add an unfilled shape on an etch layer.</p>
Defer performing dynamic fill	Choose to prevent the shape you are currently adding from dynamically updating. Disabling this field causes the shape you are currently adding to be dynamically updated and filled if the shape boundary exists. This field is disabled for unfilled and non etch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <i>Esc</i> key to stop the process.
Assign net name	Enter a net to assign to the shape, choose a net from the dropdown list, or click ... to display the Select Net dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using <i>Logic – Identify DC Nets</i> (identify nets command). Changing an assigned net dynamically fills and updates the shape. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current shape, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
Shape grid	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the Grids Display dialog box, available by running the define grid command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings.
None	<i>None:</i> Choose to create shapes off grid in user units, specified on the <i>Drawing Parameters</i> dialog box (prmed command).
Current Subclass Grid	<i>Current Subclass Grid:</i> Choose to use the predefined system grid values for the active class/subclass. This is the default value.
Shape Creation	Choose to sets shape creation mode.
Draw Rectangle	<i>Draw Rectangle:</i> Choose to draw rectangular shape. This is the default value.
Place Rectangle	<i>Place Rectangle:</i> Choose to place the rectangular shape of known size by setting the <i>Height</i> and <i>Width</i> fields.
Corners	Choose to sets type of corners for the shape.
Orthogonal	<i>Orthogonal:</i> Choose to set the corner type as orthogonal. This is the default value.
Chamfer	<i>Chamfer:</i> Choose to set the corner type as chamfer.
Round	<i>Round:</i> Choose to set the corner type as round. You can set the chamfer and round corner parameters in two ways:
Explicit Length	<i>Explicit Length:</i> Choose to control the corner length and radius.
% of Short Edge	<i>% of Short Edge:</i> Choose to set the trim size as percent of short edge.

Related Topics

- [Adding a Static Rectangular Shape to the Design](#)

Adding a Dynamic Copper Fill Rectangular Shape to the Design

To add a rectangular shape with dynamic copper fill, perform these steps:

1. Run `shape_app add rect` command.
2. Verify the active class and subclass are correct.
3. Begin drawing the shape.
4. On the *Options* panel, specify the *Shape Fill Type* as *Dynamic Copper* to add a positive rectangular shape whose copper area and voids are automatically filled and updated whenever you edit the shape's elements or its boundary.

 You can defer the update of a dynamic copper fill shape by choosing the *Defer Performing Dynamic Fill* field on the *Options* panel. You can thereby edit the boundaries of a single shape without impacting performance and prevent other shapes from becoming out of date.

5. Right-click to display the pop-up menu and choose *Parameters* to display the *Shape Instance Parameters* dialog box, to specify a *solid* or *xhatch* *Fill Style* on the *Shape Fill* tab for the dynamic copper fill shape you are adding.
6. Attach the shape to a net by specifying a net name in the *Assign Net Name* field, choosing a net name from the dropdown list, right-clicking to display the pop-up menu and choosing *Assign Nets*, or clicking... to display the *Select Nets* dialog box from which you can choose a net. This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
7. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field (optional).
8. Select shape creation mode.
9. Select shape corner type from *Corners* field.
10. Left click and drag the cursor until the rectangle shape is the correct size; then right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up to the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

Related Topics

- [shape_app add rect](#)

Adding a Static Rectangular Shape to the Design

To add a static rectangular shape, perform these steps:

1. Run `shape_app add rect` command.
2. Verify the active class and subclass are correct.
3. On the *Options* panel, specify the *Shape Fill Type*:
 - o Choose *Static Solid* to create a static solidly filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
or
 - o Choose *Static Crosshatch* to create a static crosshatch filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
4. Attach the shape to a net by specifying a net name in the *Assign Net Name* field, choosing a net name from the dropdown list, right-clicking to display the pop-up menu and choosing *Assign Nets*, or clicking... to display the *Select Nets* dialog box from which you can choose a net. This makes the shape part of the net you chose. Until you do this step, an etch shape is on a dummy net (which means no net). Non-etch shapes are never on a net.
5. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field.(optional).
6. Select shape creation mode.
7. Select shape corner type from *Corners* field.
8. Left click and drag the cursor until the rectangle shape is the correct size; then right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

Related Topics

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

shape_app change type to dynamic

The `shape_app change type to dynamic` command executes in Shape Edit application mode when you select a shape, right-click and choose *Change Shape Type to Dynamic* from the pop-up menu that displays. The command changes shape fill type from static solid into dynamic copper fill.

Access Using

In Shape Edit application mode, choose a shape. Click the right mouse button to display the pop-up menu. Choose *Change Shape Type to Dynamic*.

Changing Shape Fill From Static Solid To Dynamic Copper Fill

Perform the following steps to change shape fill from static solid to dynamic copper fill:

1. Select a shape. Right click and choose *Change Shape Type to Dynamic* from pop-up menu.
2. The layout editor displays following warning message:

Conversion will result in loss of voids within shapes.-- continue

3. Click Yes in the message dialog box.

The shape changes from static solid to dynamic copper fill.

shape_app change type to static

The `shape_app change type to static` command executes in Shape Edit application mode when you select a shape, right-click and choose *Change Shape Type to Static* from the pop-up menu that displays. The command changes shape fill type from dynamic copper fill into static solid.

Access Using

- Context menu: In the Shape Edit application mode, choose *Change Shape Type to Static*.

Changing Shape Fill From Dynamic Copper Fill To Static Solid

Perform the following steps to change shape fill from dynamic copper fill to static solid:

1. Select a shape. Right click and choose *Change Shape Type to Static* from pop-up menu.
2. The layout editor displays following warning message:

Conversion will result in loss of original shape boundary, parametr settings, and user defined voids.-- continue

3. Click *Yes* in the message dialog box.
The shape changes from dynamic copper fill to static solid.

shape_app check

The `shape_app check` command executes in Shape edit application mode when you choose a shape, right-click, and choose *Check* from the pop-up menu that displays.

The command performs checks to identify small or narrow areas that might cause problems during artwork generation. The layout editor identifies these problems with circular figures, called shape problems, which are the same size and color as DRC markers. These circular figures display on a subclass of the MANUFACTURING class called SHAPE PROBLEMS, which is created only after the layout editor detects a shape problem. The results are saved in a `shape_check.log` file. To check multiple shapes, select them by window.

When checking static solid fill shapes for vector-based artwork (Gerber 4x00 and Gerber 6x00 artwork formats), the layout editor is limited to checking apertures of 4 mils or greater as specified in the *Enter Smallest Aperture Available* dialog box. For dynamic shapes, same the value is used as specified in the *Minimum Aperture* field on the *Void Controls* tab of the *Global Dynamic Shape Parameters* dialog box.

Access Using

- Context menu: In the Shape Edit application mode, choose *Check*.

Checking For Narrow Parts In The Design

To check for narrow parts in the design, follow these steps:

1. In the *Find* filter, select *Shapes*.
2. Hover your cursor over a shape or draw a window to select multiple shapes.
The tool highlights the shape and a datatip identifies its name.
3. Right click and choose *Check* from pop-up menu.
4. Enter the aperture size of the smallest aperture in your aperture table in the *Enter Smallest Aperture Available* dialog box.
5. Click *OK*. The command window prompt displays the following message:

Checking for shape edges less than x.00 apart.

Clearing old narrow point markers ...

New markers indicate narrow parts in shape.

Shape checking completed ... x problem pts found.

Shape check results written to shape_check.log

If errors were found, change the *Active Class* and *Subclass* to MANUFACTURING class and SHAPE PROBLEMS subclass and review the errors.

shape assign net

The `shape assign net` command lets you assign a net to the selected shape.

- ✓ If the shapes are already placed in design and tied to a wrong net, and you would like the shapes to inherit net from pin, use *Derive Assignment* in APD. First, de-assign net of shapes to a dummy. Choose *Logic – Deassign Net*. Use [shapeedit](#) and select all the shapes you want to change the net name. Then, choose *Logic – Derive Assignment* and select all the shapes from first step. The shapes should then derive assignment from connected pins.

Related Topics

- [Assigning Nets To Selected Shapes](#)

Shape Assign Net Command: Options Panel

<i>Assign net name</i>	Enter a net to assign to the shape, choose a net from the dropdown list, or click ... to display the <i>Select a Net</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using <i>Logic – Identify DC Nets</i> (Identify nets command). Changing an assigned net dynamically fills and updates the shape. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current shape, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
------------------------	--

Assigning Nets To Selected Shapes

To assign nets to selected shapes:

1. Select a shape.
2. Run `shape assign net` command.
The *Select a Net* dialog box is displayed.
3. Select a net from the dropdown list.
4. Right-click and choose *Done* to exit the command.

Related Topics

- [shape assign net](#)

shape change type

The `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.

You may also change shape fill type from *Dynamic Copper* to *Static Solid* or vice versa. For example, you may change shape fill type from *Dynamic Copper* at the end of production to preserve its current state.

- ⓘ When you change shape fill type from *Static Solid* to *Dynamic Copper*, voids within the shape are lost. Similarly, when you change shape fill type from *Dynamic Copper* to *Static Solid*, the following important shape information is lost:
- original shape boundary
 - dynamic shape parameter settings
 - user-defined (manual) void information

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Changing a Static Shape Fill Type To Dynamic Copper](#)
- [Changing a Dynamic Copper Shape Fill Type to Static](#)

Shape Change Type Command: Options Panel

Access Using

- Menu Path: *Shape – Change Shape Type*

<i>Active Class and Subclass</i>	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Shape Fill</i>	
<i>Type</i>	
<i>To Dynamic Copper</i>	Choose to convert the currently chosen shapes from static solid to dynamic copper fill.
<i>To Static Solid</i>	Choose to convert the currently chosen shapes from dynamic copper fill to static solid.
<i>Defer performing dynamic fill</i>	Choose to prevent the currently chosen shape from dynamically updating. Disabling this field causes the shape you are currently adding to be dynamically updated and filled if the shape boundary exists. This field is disabled for unfilled and nonetch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <code>Esc</code> key to stop the process.
<i>Assign net name</i>	Disabled for this command.
<i>Shape grid</i>	Disabled for this command.
<i>Segment type</i>	Disabled for this command.
<i>Type</i>	Disabled for this command.
<i>Angle</i>	Disabled for this command.
<i>Arc radius</i>	Disabled for this command.

Related Topics

- [Changing a Dynamic Copper Shape Fill Type to Static](#)

Changing a Static Shape Fill Type To Dynamic Copper

Perform these steps to change a static shape fill type to a dynamic copper:

1. Run `shape change type`. The command window prompt displays the following message:

Pick static shapes to be converted to dynamic

2. Choose *To dynamic copper* in the *Shape Fill* field.
3. Choose a shape by clicking on it. The command window prompt displays the following message:

Converting static shapes to dynamic copper fill

4. Right-click to display the pop-up menu and select:
Done to exit the command.
Cancel to delete any modifications made during this session.
Oops to back up to the last pick.

Related Topics

- [shape change type](#)

Changing a Dynamic Copper Shape Fill Type to Static

Perform these steps to change a dynamic copper shape fill type to static:

1. Run `shape change type`. The command window prompt displays the following message:

Pick dynamic shapes to be converted to static solid

2. Choose *To static solid* in the *Shape Fill* field.

3. Choose a shape by clicking on it. A confirm dialog box displays:

Warning - This conversion will result in loss of original shape boundary, parameter settings, and user-defined voids- Continue?

4. Click *Yes* to proceed with the conversion. The command window prompt displays the following message:

Converting dynamic copper fill shapes to static solid

5. Right-click to display the pop-up menu and choose:

Done to exit the command.

Cancel to delete any modifications made during this session.

Oops to back up to the last pick

Related Topics

- [shape change type](#)
- [Shape Change Type Command: Options Panel](#)

shape check

The `shape check` command identifies small or narrow areas that might cause problems during artwork generation. The layout editor identifies these problems with circular figures, called shape problems, which are the same size and color as DRC markers. These circular figures display on a subclass of the MANUFACTURING class called SHAPE PROBLEMS, which is created only after the layout editor detects a shape problem.

When checking static solid fill shapes for vector-based artwork (Gerber 4x00 and Gerber 6x00 artwork formats), the layout editor is limited to checking apertures of 4 mils or greater as specified in the *Enter Smallest Aperture Available* dialog box. For dynamic shapes, same the value is used as specified in the *Minimum Aperture* field on the *Void Controls* tab of the *Global Dynamic Shape Parameters* dialog box.

Related Topics

- [Checking Solid Fill Etch/Conductor Shapes for Vector-based Artwork](#)
- [Checking Dynamic Copper Fill Etch/Conductor Shapes for Vector-based Artwork](#)

Shape Check Command: Options Panel

Access Using

- Menu Path: *Shape – Check*

<i>Active Class and Subclass</i>	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Shape fill</i>	Disabled for this command.
<i>Type</i>	Disabled for this command.
<i>Defer performing dynamic fill</i>	Disabled for this command.
<i>Assign net name</i>	Enter a net to assign to the shape, choose a net from the dropdown list, or click ... to display the <i>Select Nets</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using <i>Logic – Identify DC Nets</i> (Identify nets command). Changing an assigned net dynamically fills and updates the shape. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current shape, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
<i>Shape grid</i>	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by running the define grid command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None</i> : Choose to create shapes off grid in user units, specified on the Drawing Parameters dialog box (see Creating Drawing Parameters). <i>Current Subclass Grid</i> : Choose to use the predefined system grid values for the active class/subclass. This is the default value.
<i>Segment type</i>	Disabled for this command.
<i>Type</i>	Disabled for this command.
<i>Angle</i>	Disabled for this command.
<i>Arc radius</i>	Disabled for this command.

Related Topics

- [Checking Dynamic Copper Fill Etch/Conductor Shapes for Vector-based Artwork](#)

Checking Solid Fill Etch/Conductor Shapes for Vector-based Artwork

To check solid fill etch/conductor shapes for vector based artwork, follow these steps:

1. Choose *Shape – Check* (`shape check` command). The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose a shape.
3. Enter the aperture size of the smallest aperture in your aperture table in the *Enter Smallest Aperture Available* dialog box.
4. Click OK. The command window prompt displays the following message:

Shape checking completed...x problem pts found

If errors were found, change the *Active Class* and *Subclass* to MANUFACTURING class and SHAPE PROBLEMS subclass and review the errors.

Related Topics

- [shape check](#)

Checking Dynamic Copper Fill Etch/Conductor Shapes for Vector-based Artwork

To check dynamic copper fill etch/conductor shapes for vector based artwork, follow these steps:

1. Choose *Shape – Check* (`shape check` command). The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose a shape. The command window prompt displays the following message:

Shape checking completed...x problem pts found

If errors occur, change the *Active Class* and *Subclass* to MANUFACTURING class and SHAPE PROBLEMS subclass and review the errors.

Related Topics

- [shape check](#)
- [Shape Check Command: Options Panel](#)

shape copy layers

The `shape copy layers` command executes in Shape Edit application mode when you choose a shape, right-click and choose and *Copy to Layers* from the pop-up menu that displays. The command replicates a shape on the chosen subclass layers.

Related Topics

- [Copying Shapes to Selected Subclass Layers](#)

Shape Copy to Layers Dialog Box

Access Using

In Shape Edit application mode, choose a shape. Click the right mouse button to display the pop-up menu. Choose *Copy to Layers*.

Select subclasses to copy to	Choose the subclass layers to replicate the shape.
<i>Copy</i>	Replicates the shape on the chosen subclass layers.
<i>Undo last copy</i>	Choose to convert the currently chosen shapes from dynamic copper fill to static solid.
<i>OK</i>	Saves all copies and closes the dialog box.
<i>Cancel</i>	Cancels all copies and closes the dialog box.

Copying Shapes to Selected Subclass Layers

To copy shapes to selected subclass layers:

1. Select a shape. Right click and choose *Copy to Layers* from pop-up menu.
The *Shape copy to layers* dialog box is displayed.
2. Enable checkboxes for subclasses.
3. Click *OK* to close and apply the command.

Related Topics

- [shape copy layers](#)

shape defer fill

An internal Cadence engineering command.

shape edit boundary

The `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 is closer, or to the boundary edge. Snapping to the edge occurs on the intersection of a normal vector to the edge.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Changing a Shape or Void Outline](#)

Shape Edit Boundary Command: Options Panel

Access Using

- Menu Path: *Shape – Edit Boundary*



- Toolbar Icon:

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
Shape fill	Disabled for this command.
Type	Disabled for this command.
Defer performing dynamic fill	Disabled for this command.
Shape grid	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by running the define grid command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None</i> : Choose to create shapes off grid in user units, specified on the <i>Drawing Parameters</i> dialog box (see Creating Drawing Parameters). <i>Current Subclass Grid</i> : Choose to use the grid values defined for the active class/subclass. This is the default value.
Segment type	The following line segment types are available:
Type	<i>Line</i> : Choose to use any angle line. <i>Line 45</i> : Choose to miter lines to a 45 degree at vertex locations. <i>Line Orthogonal</i> : Choose to create lines at 90 degree angles at vertex locations. <i>Arc</i> : Choose to create an arc. Available only when adding polygons. Once you enter an arc, this field automatically defaults to the previous line segment type specified in the <i>Type</i> field. Cursor position as it moves toward the arc end point determines arc direction (clockwise or counter clockwise).
Angle	Available only if you specified <i>Arc</i> as the line segment type in the <i>Type</i> field as an alternative to selecting the end point of the arc. Enter a value to create an arc from the start point with the specified angle. The arc is tangent to the start and end point, which determines the arc's direction.
Arc radius	Available only if you specified <i>Arc</i> as the line segment type in the <i>Type</i> field. Enter the next arc segment with a given radius. A zero value creates a tangent arc.

Changing a Shape or Void Outline

To change shapes or void outlines, follow these steps:

1. Run the `shape edit boundary` command.

The command window prompt displays:

`Pick shape to edit.`

2. 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 or void boundary.`

3. Choose the next point of the new boundary, and continue choosing points until the edit is complete.

4. 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.

5. Right-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.

Related Topics

- [shape edit boundary](#)

shape global param

The `shape global param` command displays the *Global Dynamic Shape Parameters* dialog box from which you can apply shape outline parameters to all dynamic copper fill shapes.

- ⚠ The dialog box title bar displays *Global Dynamic Shape Parameters* when you run the `shape global param` command and *Shape Instance Parameters dialog box* when you run the `shape param` command with a particular dynamic copper fill shape chosen. The *Global Dynamic Shape Parameters* dialog box defines parameters for all dynamic shapes whereas the *Shape Instance Parameters* dialog box defines information for a single dynamic shape instance.

Parameters include those governing the type of shape fill, thermal relief connect lines, and void clearances. You can change these global default parameter settings, and all modifications then propagate to all existing dynamic shapes. If custom setting are required, you can override Global Dynamic Shape Parameters on individual shapes using the `shape param` command or on elements such as pins or vias using properties available using the `property edit` command. For additional related information on shape-related properties, see the [Allegro Platform Properties Reference](#).

The custom settings always override the global default settings. After modifying these settings, you may also choose to revert to the global parameter setting values.

The following conditions will make shapes go to disabled:

- Uprev involving automatic route keepout padstacks.
- Techfile import if not doing DRC update
- Glossing
- `axlPadstackEdit` (SKILL padstack editing function).

The following require shape up to date:

- ICP2581 output
- Stream out
- Artwork out
- Delete islands (not automatic island deletion).
- Batch DRC

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Deferring Dynamic Copper Fill for All Shapes In a Board Design](#)
- [Specifying Global Parameters for Dynamic Shapes](#)

Global Dynamic Shape Parameters Dialog Box

Access Using

- Menu Path: *Shape – Global Dynamic Params*

- ✓ The Design Parameter Editor is also available for setting the parameters you define in the Global Dynamic Shape Parameters dialog box. Choose *Setup – Design Parameters* (`prmed` command), click the *Shapes* tab and click *Edit global dynamic shape parameters*.

Shape Fill Tab

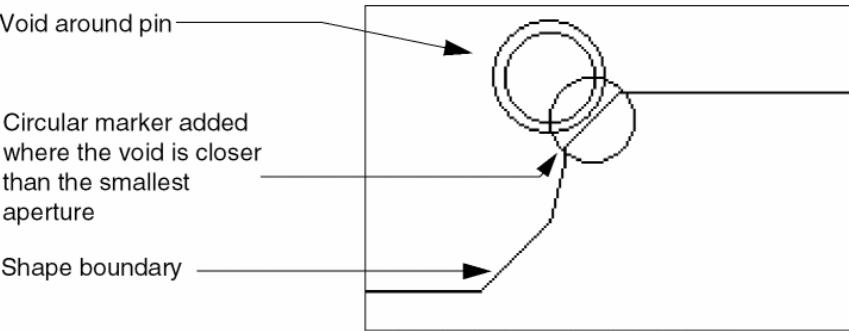
<i>Out of date shapes</i>	<p>Lists the number of dynamic shapes for which the <i>Dynamic Copper Fill</i> mode has been set to <i>Fast</i> or <i>Disabled</i>.</p>												
<i>Update to Smooth</i>	<p>Click to automatically void and run DRC on all dynamically filled shapes, making all dynamic shapes up-to-date (<i>Dynamic Copper Fill</i> mode set to <i>Smooth</i>) and produce artwork-quality output (regardless of whether you chose <i>Fast</i> or <i>Disabled</i> in the <i>Fill Mode</i> field). Changes the current <i>Dynamic Copper Fill</i> mode to <i>Smooth</i>. To cancel dynamic filling of complex shapes for a large design, you can use the <code>Esc</code> key to stop the process, which leaves the shapes out of date. If several shapes are in the midst of dynamically filling when you invoke the Esc key:</p> <ul style="list-style-type: none"> • Shapes already dynamically filled remain completed. • Shapes in the process of dynamically filling remain unfilled and marked out of date. • Shapes whose dynamic fill is yet to be updated remain filled but marked out of date. 												
<i>Dynamic Fill</i>	<p>Choose one of these copper fill options that controls the update of all dynamically filled shapes globally.</p>												
<i>Smooth</i>	<p>Choose to automatically fill, void, and run DRC on all dynamic shapes and produce artwork quality output. If this field is disabled, subsequently choosing it automatically updates existing out-of-date shapes.</p>												
<i>Fast</i>	<p>Choose to see connectivity without full edge smoothing and thermal hookups in a fast fill mode to obtain true clearances around elements and resolve intersections with other voids. Shape voiding results are similar to smooth shapes, preserving connectivity and overall appearance.</p> <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p>⚠ Update to <i>Smooth</i> before releasing artwork.</p> </div> <p>The following operations are not performed when in the <i>Fast</i> mode:</p> <ul style="list-style-type: none"> • Automatic deletion of antenna shapes • Suppression of shape less than a specified value • Display of shape related DRCs 												
<i>Disabled</i>	<p>Choose to globally defer dynamically filling all dynamic shapes you subsequently create or modify. Use this option to edit etch for medium to large ECOs, manual ECOs or to run batch programs such as netin, gloss, testprep add/replace vias, for example, for faster performance. Shapes created under this global setting are "out of date" because they are not filled or voided, and DRC does not run, and as a result artwork cannot be produced. If this field is disabled, subsequently choosing <i>Smooth</i> or <i>Fast</i> automatically updates existing out-of-date shapes.</p>												
<i>Xhatch style</i>	<p>Click the drop-down list and choose one of the listed styles, as shown in the following image. The configuration of the displayed field options depends on the fill style you choose.</p> <table style="width: 100%; border-collapse: collapse;"> <tbody> <tr> <td style="text-align: center; width: 30px;"> </td> <td style="width: 100px; vertical-align: top;"> Vertical Vertical lines only </td> </tr> <tr> <td style="text-align: center;"> </td> <td style="vertical-align: top;"> Horizontal Horizontal lines only </td> </tr> <tr> <td style="text-align: center;"> </td> <td style="vertical-align: top;"> Diag_Pos Diagonal +45 lines only </td> </tr> <tr> <td style="text-align: center;"> </td> <td style="vertical-align: top;"> Diag_Neg Diagonal -45 lines only </td> </tr> <tr> <td style="text-align: center;"> </td> <td style="vertical-align: top;"> Diag_Both Diagonal +45 And -45 Lines </td> </tr> <tr> <td style="text-align: center;"> </td> <td style="vertical-align: top;"> Hori_Vert Vertical and horizontal lines </td> </tr> </tbody> </table>		Vertical Vertical lines only		Horizontal Horizontal lines only		Diag_Pos Diagonal +45 lines only		Diag_Neg Diagonal -45 lines only		Diag_Both Diagonal +45 And -45 Lines		Hori_Vert Vertical and horizontal lines
	Vertical Vertical lines only												
	Horizontal Horizontal lines only												
	Diag_Pos Diagonal +45 lines only												
	Diag_Neg Diagonal -45 lines only												
	Diag_Both Diagonal +45 And -45 Lines												
	Hori_Vert Vertical and horizontal lines												
<i>Hatch Set</i>	<p>A hatch set is the set of parallel lines that the layout editor creates to fill a crosshatched shape. It creates one or two sets according to your choice of style, and if two sets, usually at 90 degrees to each other. You can set the width, spacing, and angle for each set independently. If you change any setting after choosing one of the styles above, the layout editor changes the choice in the <i>Xhatch Style</i> field to <i>Custom</i>.</p>												
<i>Linewidth</i>	<p>Sets the width in user units of the crosshatch lines for each set. The line width must be less than or equal to the Border Width specified.</p>												

<i>Spacing</i>	Sets the center-to-center spacing in user units of the crosshatch lines for each set.
<i>Angle</i>	Sets the angle in degrees of the crosshatch lines for each set.
<i>Origin X and Origin Y</i>	Sets the x,y coordinates of the crosshatch lines.
<i>Border Width</i>	Specifies the width of the shape boundary line. The border width cannot be smaller than the line width.
<i>Automatically delete isolated shapes</i>	<p>Deletes all islands dynamically from the shapes or when shapes are updated in <i>Smooth</i> mode.</p> <p>⚠ When a shape etch is updated, an etch segment that has no connected elements is removed from the database. If you add an object in an area that the removed etch segment would connect to, the segment will be kept in the design.</p> <p>✓ It is suggested that before setting this option, you either delete all the boundary shape voids by adding the <code>DYN_ISLAND_DELETE</code> property, or delete and repour the shapes themselves.</p>
<i>Automatically delete antenna shapes</i>	Deletes all dynamic etch shapes, which have connected to single via only.

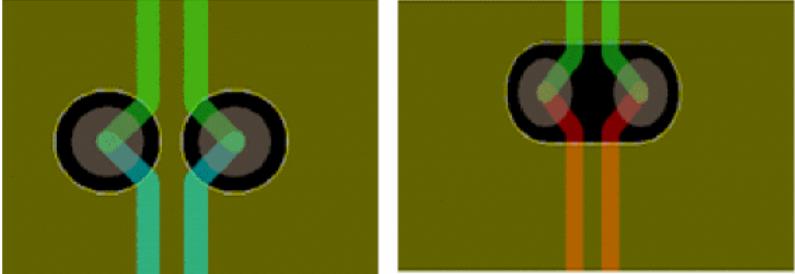
Void Controls Tab

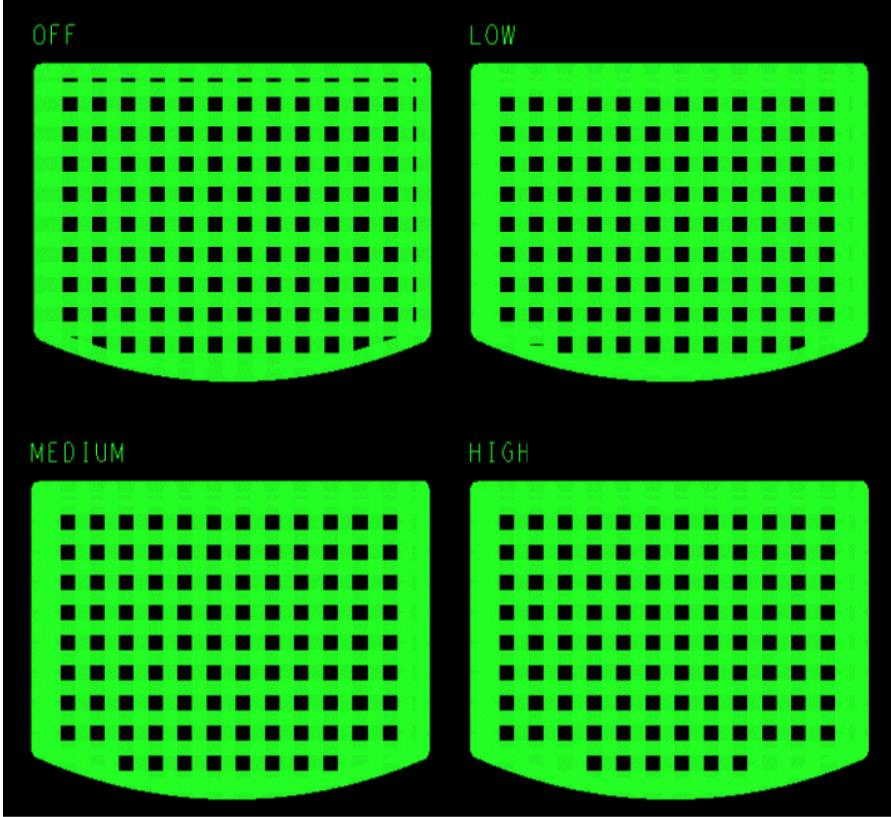
Controls the artcheck routine, which verifies that the copper area can be filled properly when creating the artwork (Gerber) files for a layer. You can specify how small copper islands are treated, and affect whether a copper area is contiguous or split. To simplify clearance areas, the settings on this tab evaluate pin patterns and control the merging of multiple polygons into one copper area (if on the same net).

<i>Artwork format</i>	Specifies an expected artwork format (raster/vector) and determines the next displayed field option. If you choose Gerber 4x00 or 6x00, Minimum aperture for artwork fill becomes the next displayed field option; otherwise Minimum aperture for gap width displays. It cannot be overridden at the shape-instance parameter level.
-----------------------	--

<p><i>Minimum aperture for artwork fill</i></p>	<p>Specifies the width in user units of the smallest plotting aperture to the <code>artwork</code> command. For solid fill style only. This is a shape-instance value.</p> <p>⚠ When you prepare artwork, the voids do not fill when a void and the shape boundary are closer than the smallest aperture. All points like this are marked with a circular figure on class MANUFACTURING, subclass SHAPE PROBLEMS. However, the voids are added as shown in the following figure. When you fill the shape, all the MANUFACTURING/SHAPE PROBLEMS markers are deleted.</p> <p>Figure -1 Added Voids</p> 
<p><i>Minimum aperture for gap width</i></p>	<p>Specifies in user units that distance between two voids. The voids are merged and treated as a single void, if void edges are less than this value.</p>
<p><i>Suppress Shapes less than</i></p>	<p>Specifies in user units that any shape areas smaller than the area value specified here be suppressed. Dynamic voiding can split a shape into multiple shapes. Any shapes smaller than the surface area you specify in this field are ignored.</p>
<p><i>Create pin voids</i></p>	<p>Generates voids around a series of pads, mainly DIP patterns, either <i>In-Line</i> or <i>Individually</i>. <i>In Line</i> correlates to drawing one void around the entire group of pads. <i>Individually</i> correlates to drawing a void around each pad separately.</p>
<p><i>Acute angle trim control</i></p>	<p>Specifies solid outline corner style (round or chamfered) for solid shapes for raster artwork formats. The minimum gap width is used for the corner radius (round) or length (chamfered).</p> <p>⚠ Full Round mode rounds all 90 degrees corners; route keepouts with square corners will have their voids rounded.</p>
<p><i>Rectangle pad void corner style</i></p>	<p>Specifies dynamic shapes voiding style at the corners of the rectangular pads that are defined as rectangle or square in the padstack. You can choose between <i>Round</i> or <i>Square</i> corners.</p>
<p><i>Void to suppressed pads (ignore dynamic suppression)</i></p>	<p>Set for static copper shapes to retain the pad to shape size even if pads are suppressed. Pads might be suppressed for some static copper shapes to enable legacy boards that used Gerber style pad suppression to maintain shape voiding when updated.</p>

S Commands
S Commands--shape global param

<i>DiffPair combined void for vias added with Return Path Option</i>	Enables creating a single oblong void for Diff Pair vias when the vias are part of a Diff Pair return path via group.
	 <p style="text-align: center;">Default Single Oblong Void</p>
<i>Snap Voids to Hatch Grid</i>	Attaches the created voids to the hatch grid. For crosshatched shapes only. The following example on the right shows how the void snaps to the hatch grid if this field is enabled; on the left, if this field is disabled.

<p>Fill Xhatch cells</p> <p>Fill xhatch cells from low to high. The choices are <i>Off</i>, <i>Low</i>, <i>Medium</i>, and <i>High</i>. The <i>Off</i> does no filling whereas <i>High</i> completely fills a cell that has an intersection with a void or shape boundary. The following example shows results with different options.</p>	
<p>⚠ <i>Fill Xhatch cells</i> does not work with <i>Snap Voids to Hatch Grid</i> option.</p>	

Clearances Tab

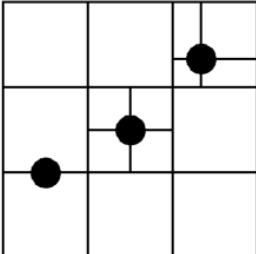
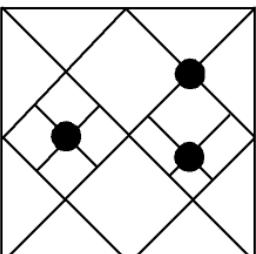
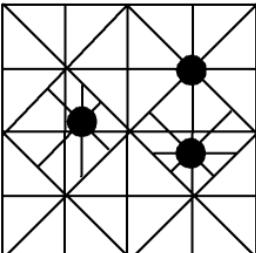
Specifies how far the copper is recessed from any conductive features contained within the copper area to prevent shorting. These include thru and SMD pins, vias, lines and clines, shapes/rectangles, and text. The choices are:

Thermal/Anti	Makes a void the size of thermal relief and antipad as defined in the padstack of a pin or via. Applies only to the pin clearances. If antipad clearance is smaller than the DRC values, voiding increases the clearance to the DRC value.
DRC	Makes a void sized using the DRC distance as clearance around the pad.
Oversize Value	Increases the clearance beyond the specified DRC or thermal/antipad value for elements requiring voiding that are inside a shape boundary. Use this value as an alternative to setting up spacing constraints or if the specified DRC value is especially small. This value is not meant for use with shape keepouts.

Thermal Relief Connects Tab

Specifies how pins and vias with the same net name as the shape should be connected to the shape.

Thru/SMD Pins/Vias	Indicates how clines are to be generated. <ul style="list-style-type: none"> • <i>Orthogonal</i>: Connects straight up-down or left-right as shown in the following figure. The pin connects directly to the void outline or hatch lines.
---------------------------	--

	<p>Figure -2 Orthogonal Clines</p> 
	<ul style="list-style-type: none"> • <i>Diagonal</i>: Connects diagonally upper left to lower right and lower left to upper right as shown in the following figure.
	<p>Figure -3 Diagonal Clines</p> 
	<ul style="list-style-type: none"> • <i>8-Way</i>: Connects lines from the thermal relief to the pin/via both diagonally and orthogonally as shown in the following figure.
	<p>Figure -4 8-Way Clines</p> 
	<ul style="list-style-type: none"> • <i>Full Contact</i>: Creates no voids. For crosshatched shapes, the hatch lines provide the connections, or the layout editor adds short connect lines.
	<ul style="list-style-type: none"> • <i>None</i>: No thermal connections are created. <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p>⚠ If pad is also suppressed for the layer, voiding via with <i>None</i> will void to drill hole.</p> </div>
	<div style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> <p>⚠ Thermals are generated from pins to a static shape only if the static shape overlaps a dynamic shape and has the same thermal settings as the dynamic shape.</p> </div>
Best Contact	<p>Choose to rotate the thermal relief lines in 15 degree increments and override the chosen thermal connect style if it doesn't provide sufficient thermal connects within the specified minimum and maximum number of thermals. Use this field when the number of connected pins takes precedence over the placement (angle between thermals).</p>

Use Fixed Thermal Width of:	Controls thermal line width independently of physical constraint set mappings. Defaults to 10 mils. Value cannot be smaller than 1 mils.
Use Thermal Width Oversize of:	<p>Adds the value you specify to the default thermal connect line width, which originates in the layout editor's <i>Physical Constraint Set</i>. For example, if for a net \min line width is set to 5 mils and thermal width oversize is set to 5, the resultant line width used to generate the thermal width will be 10 mils.</p> <div style="border: 1px solid #ccc; padding: 5px;"> <p> To set different thermal width oversize values for pins and vias, use find_by_query command to search all the pins/vias in the design and apply DYN_OVERSIZE_THERM_WIDTH_ARRAY property. For more information, see Finding Objects by Query in <i>Allegro User Guide: Getting started with Physical Design</i>.</p> </div>
Use xhatch Thermal Width:	Allows cross-hatched dynamic shapes to generate thermal clines widths based upon the shape's cross hatch width.
Minimum Connects	Specifies number of connections for thru/SMD pins and vias (choosing <i>Full Contact</i> disables this field) in conjunction with the <i>Maximum Connects</i> field to control the number of thermal connections. Ignores existing clines that connect to the shape.
Maximum Connects	Specifies how many connect lines are created for the thermal relief. Up to four are allowed on orthogonal and diagonal; up to eight on the 8-way option.
Ok	Saves the settings and closes the dialog box.
Cancel	Closes the dialog box.
Apply	Saves the settings and leaves the dialog box open. If you set the <i>Dynamic Copper Fill</i> mode to <i>Smooth</i> , automatically voids and runs DRC on all dynamic shapes, making all dynamic shapes up-to-date and producing artwork-quality output.
Reset	Reverts to the original parameters that appeared when you initially opened the dialog box. You can use <i>Apply</i> and <i>Reset</i> to toggle between previous and current settings to assess before/after effects of parameter changes.

Related Topics

- [Specifying Global Parameters for Dynamic Shapes](#)

Deferring Dynamic Copper Fill for All Shapes In a Board Design

To cancel dynamic filling of complex shapes for a large design, you can use the `Esc` key to stop the process. You can defer the automatic dynamic fill and voiding of all shapes or a particular shape by following these steps:

1. Run `shape global param` to display the *Global Dynamic Shape Parameters* dialog box or run `status` to display the *Status* dialog box to specify the global dynamic copper fill mode.
 - a. In the *Global Dynamic Shape Parameters* dialog box, choose *Disabled* as the global value for the *Dynamic Fill* copper fill mode.
OR
 - b. In the *Status* tab, choose *Disabled* as the *Dynamic Fill* copper fill mode in the *Shapes (Dynamic Copper Pour)* field.

 Choosing *Disabled* allows you to edit etch for large ECOs without impacting performance; however, shapes you create do not get DRCs or automatic voiding. Once modifications are complete, you can reset this field to *Smooth* to automatically update existing out of date shapes.

Related Topics

- [shape global param](#)

Specifying Global Parameters for Dynamic Shapes

To specify global parameters for dynamic shapes:

1. On the *Shape Fill* tab, specify the dynamic copper fill mode to *Smooth* or *Fast*, or *Disable* to prevent dynamic copper fill.
2. Specify the *Xhatch Style*.
3. Specify the width, spacing, and angle for each hatch set.
4. On the *Void Control* tab, specify raster or vector artwork format; if raster specify a solid outlines corner style in the *Acute Angle Trim Control* field.
5. In the *Suppress Shapes Less Than* field, specify whether to suppress shape areas smaller than the area value you enter.
6. Choose to generate voids in line or individually.
7. For crosshatch shapes, choose the *Snap Voids to Hatch Grid* field to attach created voids to the hatch grid.
8. On the *Clearances* tab, specify how far away copper should be kept from pins, vias, lines, shapes, and text.
9. Click *Apply* to implement the parameters. If you set the *Dynamic Copper Fill* mode to *Smooth*, automatically voids and runs DRC on all dynamic shapes, making all dynamic shapes up-to-date and producing artwork-quality output.

Related Topics

- [shape global param](#)
- [Global Dynamic Shape Parameters Dialog Box](#)

shape layer param

The `shape layer param` command displays the Global Shape Layer Parameters dialog box from which you can apply shape outline parameters to all dynamic copper fill shapes in a specified layer.

See [shape global param](#) which works similarly but applies to all the layers.

The fields in the Global Shape Layer Parameters dialog box are the same as in Global Dynamic Shape Parameters except for the *Layer* field that lets you select a specific layer for which the shape parameters are being set.

shape lower priority

In Shape Edit application mode, the `shape lower priority` command assigns a lower 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 *Dynamic Shapes Report* (Select a shape and right-click to choose *Report* from pop-up menu or by using *Display – Element* ([show element](#) command) on the chosen shape).

Access Using

In Shape Edit application mode, select the shape, right-click, and choose *Lower Shape Priority* from the pop-up menu.

Lowering Shape Priority

To lower shape priority:

1. Select the dynamic shape for which you want to increase the priority.
2. Right-click and choose *Lower Shape Priority* from the pop-up menu that appears.
3. Click on the overlapping dynamic shape to which to assign a lower priority.
4. The shape's priority is immediately updated.

shape merge shapes

The `shape merge shapes` command merges overlapped shapes, as well as filled rectangles. Shapes to be merged inherit the properties of the primary shape into which other shapes are merged. Shapes must be assigned to the same net to be merged. Merging a shape over a user-defined (manual) void trims the void.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Merging Overlapping Shapes Connected to the Same Net in the Design](#)

Shape Merge Shapes Command: Options Panel

Access Using

- Menu Path: *Shape – Merge Shapes*

<i>Active Class and Subclass</i>	Choose the proper etch layer upon which to draw the polygon. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Shape fill</i>	Disabled for this command.
<i>Type</i>	Disabled for this command.
<i>Defer performing dynamic fill</i>	Disabled for this command.
<i>Assign net name</i>	Disabled for this command.
<i>Shape grid</i>	Disabled for this command.
<i>Shape Creation</i>	Disabled for this command.
<i>Corners</i>	Disabled for this command.

Merging Overlapping Shapes Connected to the Same Net in the Design

Perform the following steps to merge overlapping shapes connected to the same net in a design:

1. Run the `shape merge shapes` command. The command window prompt displays the following message:

Pick primary shape to merge to, or select all shapes to merge all possible combinations.

2. Choose a primary shape. The command window prompt displays the following message:

Pick shapes to be merged

3. Alternatively, window select all the shapes.

4. Right-click and choose *Done* from the pop-up menu to merge the shapes.

Related Topics

- [shape merge shapes](#)

shape operations or

The `shape operations or` command merges two or more shapes that are overlapping each other. This command performs logical OR operation on selected shapes and objects (shape or cline). The resultant shape is the combination of the selected shapes or clines.

Using this command, you can combine multiple shapes (or clines) that are on

- the same class/subclass
- the same logical net
- the same shape type

The resultant shape inherits common properties from the base shape. The non-common properties are inherited from the non-base objects.

This command works in both pre-select and post-select mode.

Access Using

- Menu Path: *Shape – Shape Operations – OR*

Merging Multiple Overlapping Shapes to get the Combined Shape

Follow these steps to merge overlapping shapes to get the combined shape:

1. Run the `shape operations` or command. The command window prompt displays the following message:

Pick a shape as base one.

2. Choose a base shape. The command window prompt displays the following message:

Pick another shape to operate

3. Choose another shape.

4. To select multiple shapes use the `Ctrl` key.

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

shape operations and

The `shape operations and` command merges two or more shapes that are overlapping each other. This command performs logical AND operation on selected shapes and objects (shape or cline). The resultant shape is the intersection of the selected shapes or clines that was common (overlapping) between the two shapes.

Using this command you can combine multiple shapes (or clines) that are on

- the same class/subclass
- the same logical net
- the same shape type

The resultant shape inherits common properties from the base shape. The non-common properties are inherited from the non-base objects

This command works in both pre-select and post-select mode.

Access Using

- Menu Path: *Shape – Shape Operations – AND*

Merging Multiple Overlapping Shapes to get the Intersection Shape

Follow these steps to merge overlapping shapes to get the shape of the intersection of the shapes:

1. Run the `shape operations and` command. The command window prompt displays the following message:

Pick a shape as base one.

2. Choose a base shape. The command window prompt displays the following message:

Pick another shape to operate

3. Choose another shape.

Use the `Ctrl` key to select multiple shapes .

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

shape operations andnot

The `shape operations andnot` command performs logical ANDNOT operation on overlapping shapes. The resultant shape is a void of any area that overlaps with the other shapes.

The resultant shape inherits common properties from the base shape. The non-common properties are inherited from the non-base objects.

This command works in both pre-select and post-select mode.

Access Using

- Menu Path: *Shape – Shape Operations – ANDNOT*

Merging Overlapping Shapes with ANDNOT Operation

To merge overlapping shapes such that the resultant shape is a void of any area that overlaps with the other shapes, follow these steps:

1. Run the `shape operations andnot` command. The command window prompt displays the following message:

Pick a shape as base one.

2. Choose a base shape. The command window prompt displays the following message:

Pick another shape to operate

3. Choose another shape.

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

shape operations xor

The `shape operations xor` command performs logical XOR operation on overlapping shapes. The resultant shape has a void of overlapping areas.

The resultant shape inherits common properties from the base shape. The non-common properties are inherited from the non-base objects.

This command works in both pre-select and post-select mode.

Access Using

- Menu Path: *Shape – Shape Operations – XOR*

Merging Overlapping Shapes with XOR Operation

To merge overlapping shapes such that the resultant shape is a void of overlapping areas, follow these steps:

1. Run the `shape operations xor` command. The command window prompt displays the following message:

Pick a shape as base one.

2. Choose a base shape. The command window prompt displays the following message:

Pick another shape to operate

3. Choose another shape.

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

shape param

The `shape param` command displays the *Static Shape Parameters* dialog box if a shape whose *Shape Fill Type* is defined as *Static Solid* or *Static Crosshatch* is chosen, or the *Dynamic Shape Instance Parameters* dialog box if a shape whose *Shape Fill Type* is defined as *Dynamic Copper* is chosen.

The dialog box and its contents are specific to the shape type that you choose.

- Use the *Static Shape Parameters* dialog box to define or modify shape-outline parameters that apply to a particular static shape.
- Use the *Shape Instance Parameters* dialog box to modify parameters that apply to a particular dynamic copper fill shape and thereby override those defined on the *Global Dynamic Shape Parameters* dialog box.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

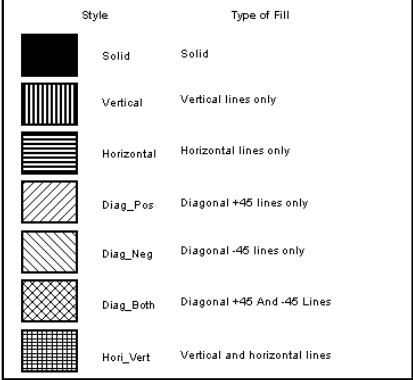
Related Topics

- [Dynamic Shape Instance Parameters Dialog Box](#)
- [Changing Parameters for a Shape Instance](#)
- [Restoring a Shape-instance Override Value to a Global Parameter Value](#)

Static Shape Parameters Dialog Box

The *Shapes* tab of the Design Parameter Editor is also available for setting the parameters you define in the Static Shape Parameters dialog box. Choose *Setup – Design Parameters* (`prmed` command) to access the Design Parameter Editor.

Shape Fill Tab

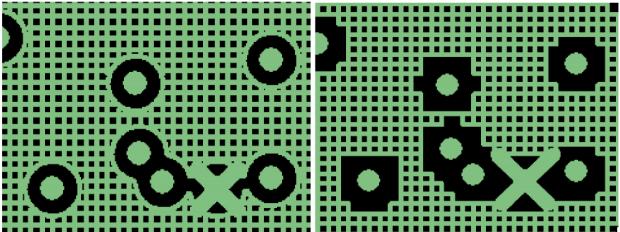
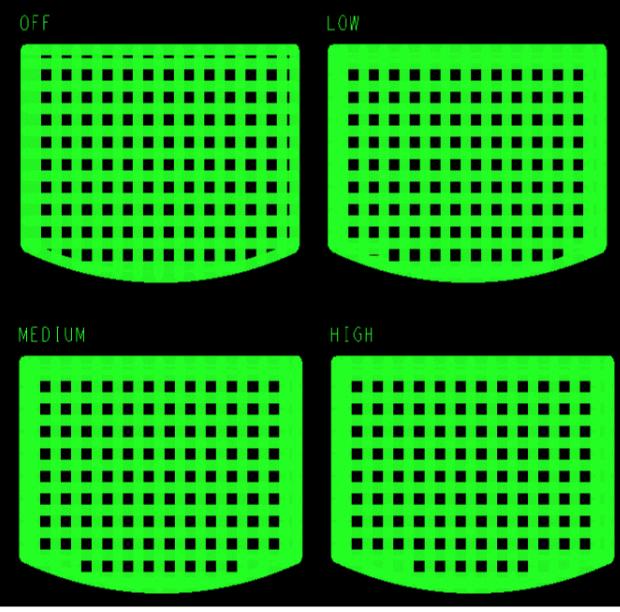
Fill style	<p>Click the drop-down list to choose a fill style. Choose <i>Custom</i> to make non-symmetrical changes, such as altering the <i>Angle</i> field. If you change any setting after choosing one of the styles above, the layout editor changes the choice in the <i>Fill Style</i> field to <i>Custom</i>.</p>  <p>If you choose any fill type except <i>Solid</i> as a fill style, the following field options display.</p>
Hatch Set	A hatch set is the set of parallel lines that the layout editor creates to fill a crosshatched shape. It creates one or two sets according to your choice of style, and if two sets, usually at 90 degrees to each other. You can set the width, spacing and angle for each set independently.
Linewidth	Sets the width in user units of the crosshatch lines for each set. The line width must be less than or equal to the <i>Border Width</i> specified.
Spacing	Sets the center-to-center spacing in user units of the crosshatch lines for each set.
Angle	Sets the angle in degrees of the crosshatch lines for each set.
Origin X and Origin Y	Sets the x, y coordinates of the crosshatch lines.
Border Width	Specifies the width of the shape boundary. The border width cannot be smaller than the line width.

Void Controls Tab

Aperture for artwork chk	Specifies in user units that distance between two voids. The voids are merged and treated as a single void, if void edges are less than this value.
Suppress Shapes Less Than	Specifies in user units that any shape areas smaller than the area value specified here be suppressed during automatic dynamic voiding, which can cause a shape to split into multiple shapes. Any shapes smaller than the surface area you specify in this field are ignored.

S Commands

S Commands--shape param

Allow Shapes to Split	Controls void creation, if the voids to be added cause the shape to be cut into disjoint parts. The options are: <i>Yes</i> : Causes the shape to be split into disjoint parts, the shape is split into two or more shapes and the process completed. <i>No</i> : Causes the shape to stay intact. In this autovoiding is canceled and the following message is seen: Shape has been broken, autovoid canceled. <i>Confirm</i> : Uses a message to display telling you the shape had been broken into multiple shapes. You are asked to Cancel the command without adding voids or to Continue the operation.
Create Pin Voids	Generates voids around a series of pads, mainly DIP patterns, either <i>In-Line</i> or <i>Individually</i> . <i>In Line</i> correlates to drawing one void around the entire group of pads. <i>Individually</i> correlates to drawing a void around each pad separately.
Trim Control	<i>Specifies solid outline corner style (round or chamfered) for solid shapes for raster artwork formats. The minimum gap width is used for the corner radius (round) or length (chamfered). The full round option trims right angles and most exterior obtuse angles. This option is used for standalone shapes which do not intersect with other objects.</i>
Snap Voids to Hatch Grid	Attaches the created voids to the hatch grid. For crosshatched shapes only. The following example on the right shows how the void snaps to the hatch grid if this field is enabled; on the left, if this field is disabled. 
Fill Xhatch cells	Fill xhatch cells from low to high. The choices are <i>Off</i> , <i>Low</i> , <i>Medium</i> , and <i>High</i> . The <i>Off</i> does no filling whereas <i>High</i> completely fills a cell that has an intersection with a void or shape boundary. The following example shows results with different options.  ⚠ Fill Xhatch cells does not work with Snap Voids to Hatch Grid option.

Clearances Tab

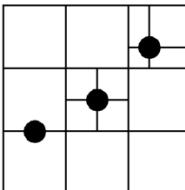
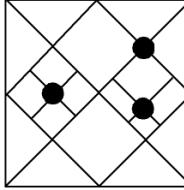
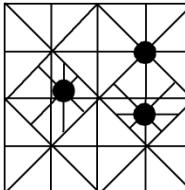
Specifies how far away the copper should be kept from thru and SMD pins, vias, lines and connect lines (clines), shapes, rectangles, and text. The choices are:

DRC Value	Uses the DRC distance as clearance around the pad.
-----------	--

Default	Uses the value shown in the Default column to the right of the field as clearance.
None	Does not create a void for this type of element

Thermal Relief Connects Tab

Specifies how pins and vias with the same net name as the shape should be connected to the shape.

Pins and Vias	Indicates how clines are to be generated. <i>Orthogonal</i> : Connects straight up-down or left-right as shown in the following figure. The pin connects directly to the void outline or hatch lines.
	Orthogonal Clines 
	<i>Diagonal</i> : Connects diagonally upper left to lower right and lower left to upper right as shown in the following figure.
	Diagonal Clines 
Full Contact	Creates no voids. For solid shapes, the shape completely fills around the pin. For crosshatched shapes, the hatch lines provide the connections or the layout editor adds short connect lines.
8-Way	Connects lines from the thermal relief to the pin/via both diagonally and orthogonally. 
Width	Specifies the width of the connect lines added as thermal relief using the DRC Value or the Default you specify. The width of the reliefs should be less than or equal to the width of the hatch line to which they connect. <i>DRC Value</i> : Uses the applicable physical constraint line width that applies to that pin or via. <i>Default</i> : Uses the value you enter in the field to the right of the button.
Maximum Connects	Specifies how many connect lines are created for the thermal relief. Up to four are allowed on orthogonal and diagonal; up to eight on the 8-way option.
Ok	Saves the settings and closes the dialog box.
Cancel	Closes the dialog box.
Apply	Saves the settings and leaves the dialog box open.
Reset	Reverts to the original parameters that appeared when you initially opened the dialog box. You can use the <i>Apply</i> and <i>Reset</i> buttons to toggle between previous and current settings to assess before/after effects of parameter changes.

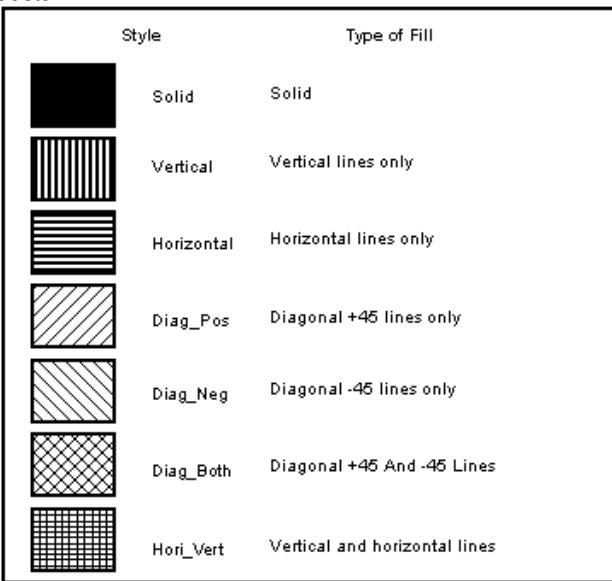
Related Topics

- [Changing Parameters for a Shape Instance](#)
- [Restoring a Shape-instance Override Value to a Global Parameter Value](#)
- [prmed](#)

Dynamic Shape Instance Parameters Dialog Box

Shape Fill Tab

Field values that appear in blue default from values set in the *Global Dynamic Shape Parameters* dialog box.

<i>Dynamic Fill</i>	You cannot change dynamic copper fill options here when you edit a single dynamic shape. The value shown defaults from the <i>Global Dynamic Shape Parameters</i> dialog box.
<i>Fill style</i>	<p>Click the drop-down list to choose a fill style. Choose <i>Custom</i> to make non-symmetrical changes, such as altering the <i>Angle</i> field. If you change any setting after choosing one of the styles above, the layout editor changes the choice in the <i>Fill Style</i> field to <i>Custom</i>.</p>  <p>If you choose any fill type except <i>Solid</i> as a fill style, the following field options display:</p>
<i>Hatch Set</i>	A hatch set is the set of parallel lines that the layout editor creates to fill a crosshatched shape. It creates one or two sets according to your choice of style, and if two sets, usually at 90 degrees to each other. You can set the width, spacing and angle for each set independently. If you change any setting after choosing one of the styles above, the layout editor changes the choice in the <i>Xhatch Style</i> field to <i>Custom</i> .
<i>Linewidth</i>	Sets the width in user units of the crosshatch lines for each set. The line width must be less than or equal to the <i>Border Width</i> specified.
<i>Spacing</i>	Sets the center-to-center spacing in user units of the crosshatch lines for each set.
<i>Angle</i>	Sets the angle in degrees of the crosshatch lines for each set.
<i>Origin X and Origin Y</i>	Sets the x,y coordinates of the crosshatch lines.
<i>Border Width</i>	Specifies the width of the shape boundary line. The border width cannot be smaller than the line width.
<i>Net Name</i>	Displays the name of the net to which the shape is attached.
<i>Layer</i>	Displays the layer upon which the shape exists.

Void Controls Tab

Controls the artcheck routine, which verifies that the copper area can be filled properly when creating the artwork (Gerber) files for a layer. You can specify how small copper islands are treated, and affect whether a copper area is contiguous or split. To simplify clearance areas, the settings on this tab evaluate pin patterns and control the merging of multiple polygons into one copper area (if on the same net).

<i>Artwork format</i>	The value shown defaults from the <i>Global Dynamic Shape Parameters</i> dialog box. <i>It cannot be overridden</i> when you edit a single dynamic shape.
<i>Minimum aperture for gap width</i>	Specifies in user units that distance between two voids. The voids are merged and treated as a single void, if void edges are less than this value.
<i>Suppress Shapes Less Than</i>	Specifies in user units that any shape areas smaller than the area value specified here be suppressed. Dynamic voiding can split a shape into multiple shapes. Any shapes smaller than the surface area you specify in this field are ignored.
<i>Create Pin Voids</i>	Generates voids around a series of pads, mainly DIP patterns, either <i>In-Line</i> or <i>Individually</i> . <i>In Line</i> correlates to drawing one void around the entire group of pads. <i>Individually</i> correlates to drawing a void around each pad separately.
<i>Acute Angle Trim Control</i>	<i>Specifies solid outline corner style (round or chamfered) for solid shapes for raster artwork formats. The minimum gap width is used for the corner radius (round) or length (chamfered).</i>
<i>Snap Voids to Hatch Grid</i>	Attaches the created voids to the hatch grid. For crosshatched shapes only. The example on the right shows how the void snaps to the hatch grid if this field is enabled; on the left, if this field is disabled.

Clearances Tab

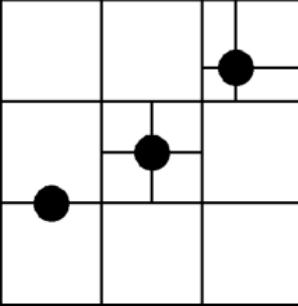
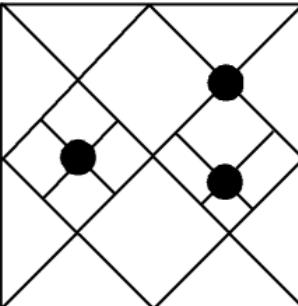
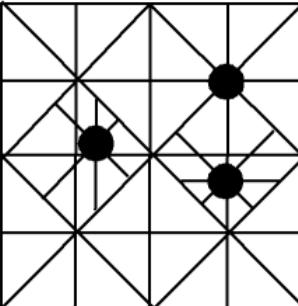
Specifies how far the copper is recessed from any conductive features contained within the copper area to prevent shorting. These include thru and SMD pins, vias, lines and clines, shapes/rectangles, and text. The choices are:

<i>Thermal/Anti</i>	Makes a void the size of thermal relief and antipad as defined in the padstack of a pin or via. Applies only to the pin clearances.
<i>DRC</i>	Makes a void sized using the DRC distance as clearance around the pad.
<i>Oversize Value</i>	Increases the clearance beyond the specified DRC or thermal/ antipad value for elements requiring voiding that are inside a shape boundary. Use this value as an alternative to setting up spacing constraints or if the specified DRC value is especially small. This value is not meant for use with shape keepouts.

Thermal Relief Connects Tab

Specifies how pins and vias with the same net name as the shape should be connected to the shape.

<i>Thru/SMD Pins and Vias</i>	Indicates how clines are to be generated.
-------------------------------	---

<i>Orthogonal</i>	<p><i>Orthogonal:</i> Connects straight up-down or left-right as shown in the following figure. The pin connects directly to the void outline or hatch lines.</p> <p>Orthogonal Clines</p> 
<i>Diagonal</i>	<p><i>Diagonal:</i> Connects diagonally upper left to lower right and lower left to upper right as shown in the following figure.</p> <p>Diagonal Clines</p> 
<i>8-Way</i>	<p>Connects lines from the thermal relief to the pin/via both diagonally and orthogonally as shown in the following figure.</p> <p>8-Way Clines</p> 
<i>Full Contact</i>	<p>Creates no voids. For crosshatched shapes, the hatch lines provide the connections, or the layout editor adds short connect lines.</p>
<i>Best Contact</i>	<p>Choose to rotate the thermal relief lines in 15 degree increments and override the chosen thermal connect style if it doesn't provide sufficient thermal connects within the specified minimum and maximum number of thermals. Use this field when the number of connected pins takes precedence over the placement (angle between thermals).</p>
<i>Use Fixed Thermal Width of:</i>	<p>Controls thermal line width independently of physical constraint set mappings. Prior to 15.2, thermal line width derived only from the physical constraint set, hampering control of power/ground routing and thermal line width using a single set of constraint values. For example, for GND routing of 25 mils, but spoke width of 10 mils, specify <code>min line width</code> of 25 mils in the physical constraint set for GND routing and thermal width of 10 mils. Defaults to 0 mils to avoid <code>uprev</code> problems.</p>

<i>Use Thermal Width Oversize of:</i>	Adds the value you specify to the default thermal connect line width, which originates in the layout editor's <i>Physical Constraint Set</i> . For example, if for a net <code>min line width</code> is set to 5 mils and thermal width oversize is set to 5, the resultant line width used to generate the thermal width will be 10 mils.
<i>Minimum Connects</i>	Specifies number of connections for thru/SMD pins and vias (choosing <i>Full Contact</i> disables this field) in conjunction with the <i>Maximum Connects</i> field to control the number of thermal connections. Ignores existing clines that connect to the shape.
<i>Maximum Connects</i>	Specifies how many connect lines are created for the thermal relief. Up to four are allowed on orthogonal and diagonal; up to eight on the 8-way option.
<i>Ok</i>	Saves the settings and closes the dialog box.
<i>Cancel</i>	Closes the dialog box.
<i>Apply</i>	Saves the settings and leaves the dialog box open. If you set the <i>Dynamic Copper Fill</i> mode to <i>Smooth</i> , automatically voids and runs DRC on all dynamic shapes, making all dynamic shapes up-to-date and producing artwork-quality output.
<i>Reset</i>	Reverts to the original parameters that appeared when you initially opened the dialog box. You can use the <i>Apply</i> and <i>Reset</i> buttons to toggle between previous and current settings to assess before/after effects of parameter changes.
<i>Clear Override</i>	Restores an override value (shown in blue) to the value (shown in black) set on the Global Dynamic Shape Parameters dialog box.

Related Topics

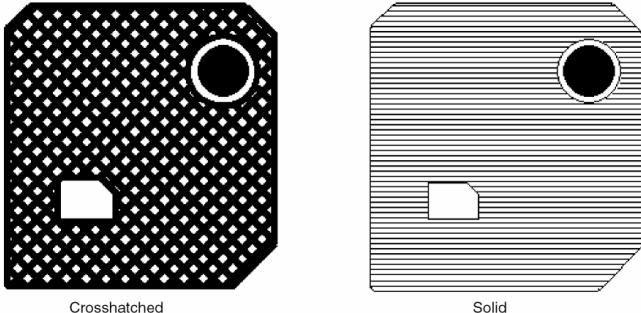
- [shape param](#)
- [Restoring a Shape-instance Override Value to a Global Parameter Value](#)

Changing Parameters for a Shape Instance

To change the parameters for a shape instance, follow these steps:

1. Choose *Shape – Select Shape or Void\Cavity* ([shape select](#) command). The command window prompt displays the following message:
Pick a shape or void to edit
2. Choose the shape whose parameters you want to change.
3. Run the `shape param` command. The dialog box that appears and its contents are specific to the shape type that you choose:
 - the *Static Shape Parameters* dialog box if a shape whose *Shape Fill Type* is defined as *Static Solid* or *Static Crosshatch* is chosen, or
 - the *Dynamic Shape Instance Parameters* dialog box if a shape whose *Shape Fill Type* is defined as *Dynamic Copper* is chosen.
4. On the *Shape Fill* tab, specify *Solid* or *Xhatch*.
The configuration of the displayed field options depends on the fill style you choose.
5. Specify the width, spacing, and angle for each hatch set if you chose a *Fill Style* of *Xhatch*.
6. On the *Void Controls* tab, in the *Suppress Shapes Less Than* field, specify whether to suppress shape areas smaller than the area value you enter.
7. Choose to generate voids in line or individually.
8. On the *Clearances* tab, specify how far away copper should be kept from pins, vias, lines, shapes, and text.
9. On the *Thermal relief connects* tab, specify how far away copper should be kept from pins, vias, lines, shapes, and text.
10. Click *Apply* to implement the parameters.

 If you change a crosshatched shape to solid fill, expands the shape boundary by one-half the boundary width, and shrinks void boundaries by one-half the boundary width, as the following example shows.



Related Topics

- [shape param](#)
- [Static Shape Parameters Dialog Box](#)

Restoring a Shape-instance Override Value to a Global Parameter Value

To restore a shape-instance override value to a global parameter value, follow these steps:

1. Choose *Shape – Select Shape or Void/Cavity* ([shape select](#) command). The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose the dynamic shape whose parameters you want to restore.

3. Run the `shape param` command.

The *Shape Instance Parameters* dialog box displays.

4. Click *Clear Override*.

The cursor changes to a cross. The status line displays the following:

Select an overridden field (in blue) to restore global value.

5. Choose the field by moving the cross-shaped cursor over it and clicking.

The field highlights, the global value is restored, and the field text color reverts to black. The status line displays:

Override cleared

 For the *Minimum* and *Maximum Connects* fields, clearing an override from one clears the other as well.

6. To cancel, pick anywhere but a blue field. The status line displays:

Override mode cancelled.

7. Repeat these steps for each override you want to clear.

Related Topics

- [shape param](#)
- [Static Shape Parameters Dialog Box](#)
- [Dynamic Shape Instance Parameters Dialog Box](#)

shape raise priority

In Shape Edit application mode, the `shape raise priority` command assigns a 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 *Dynamic Shapes Report* (Select a shape and right-click to choose *Report* from pop-up menu or by using *Display – Element* ([show element](#) command) on the chosen shape).

Access Using

- Context Menu: In the Shape Edit application mode, choose *Lower Shape Priority*.

Raising Priority of Dynamic Shapes

To raise the priority of dynamic shapes:

1. Select the dynamic shape for which you want to increase the priority.
2. Right-click and choose *Raise Shape Priority* from the pop-up menu that appears.
3. Click on the overlapping dynamic shape to which to assign a higher priority.
4. The shape's priority is immediately updated.

shape report

The `shape report` command produces the *Dynamic Shapes* report. In Shape Edit application mode, when you choose a shape, right-click and choose *Report* the command produces the *Dynamic Shapes* report as well. This report 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.

Access Using

- Context Menu: In the Shape Edit application mode, choose *Report*.

Generating a Dynamic Shapes Report

To generate a dynamic shapes report:

1. Select the shape for which you want to generate the report.
2. Right-click and choose *Report* from the pop-up menu that appears.
The report is launched in a separate window.

shape select

This command 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.

 To highlight the net associated with a shape when you choose it, enable the `highlight_shape_net` board level environment variable in the *User Preferences* dialog box, available by running the `enved` command.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Shape Select Command: Options Panel

Access Using

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



- Toolbar Icon:

<i>Active Class and Subclass</i>	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Shape fill</i>	Disabled for this command.
<i>Type</i>	Disabled for this command.
<i>Defer performing dynamic fill</i>	Choose to prevent the currently chosen shape from dynamically updating. Disabling this field causes the shape you are currently adding to be dynamically updated and filled if the shape boundary exists. This field is disabled for unfilled and non etch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <i>Esc</i> key to stop the process.
<i>Assign net name</i>	Enter a net to assign to the shape, choose a net from the dropdown list, or click ... to display the Select Net dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using identify nets (Logic – <i>Identify DC Nets</i>). Changing an assigned net dynamically fills and updates the shape. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current shape, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
<i>Shape grid</i>	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the Grids Display dialog box, available by running the define grid command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None</i> : Choose to create shapes off grid in user units, specified on the <i>Drawing Parameters</i> dialog box (see Creating Drawing Parameters). <i>Current Subclass Grid</i> : Choose to use the grid values defined for the active class/subclass. This is the default value.
<i>Segment type</i>	Disabled for this command.
<i>Type</i>	Disabled for this command.
<i>Angle</i>	Disabled for this command.
<i>Arc Radius</i>	Disabled for this command.

Related Topics

- [Shape Select Command Tasks](#)

Shape Select Command Tasks

You can perform the following tasks using the `shape select` command:

- [Deferring Dynamic Copper Fill for a Single Dynamic Shape](#)
- [Canceling Dynamic Fill](#)
- [Deleting a Vertex](#)
- [Editing User-defined Manual Voids](#)
- [Changing the Net Assigned to the Shape](#)
- [Changing a Filled Rectangle to a Shape](#)
- [Reviewing Shape Instance Parameters](#)
- [Moving an existing shape or void](#)
- [Deleting a Shape](#)
- [Moving a Segment](#)
- [Running a Dynamic Shape Report](#)
- [Identifying Out-of-date Dynamic Shapes](#)
- [Assigning a Higher Voiding Priority to a Dynamic Shape](#)

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)

Deferring Dynamic Copper Fill for a Single Dynamic Shape

Follow these steps to defer dynamic copper fill for a single dynamic shape:

1. Choose *Shape – Select Shape or Void/Cavity* ([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. Choose one of the following:
 - o Choose *Defer Performing Dynamic Fill* in the *Options* panel. The command window prompt displays the following message:

Unfilling shape in progress

The shape immediately unfills.

- o Right-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.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Canceling Dynamic Fill

To cancel dynamic filling of complex shapes for a large design:

- Use the `Esc` key to stop the process. The command window prompt displays the following message:

Autovoid cancel. Disabling dynamic fill.

Cancel Received, Finishing up connecting shape

Shape DRC set out of date

If several shapes are in the midst of dynamically filling when you invoke the `Esc` key:

- Shapes already dynamically filled remain completed.
- Shape in the process of dynamically filling remain unfilled and marked out of date.
- Shapes whose dynamic fill is yet to be updated remain filled but marked out of date.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Deleting a Vertex

To delete a vertex:

1. Choose *Shape – Select Shape or Void/Cavity* ([shape select](#) command). The command window prompt displays the following message:

Pick a shape or void to edit

2. 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

3. Left mouse click to choose the vertex to delete. The command window prompt displays the following message:

Pick destination

4. Move the chosen vertex as required to delete it; the cursor shape changes as it moves over the shape with edit handles. Deleting a corner of a rectangle converts it into a polygon.

5. Right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To insert the original vertex, breaking the line segment into two.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To delete another vertex.
<i>Reject</i>	To reject or undo the edits you made.
<i>Delete Vertex</i>	To delete the selected vertex.
<i>Move</i>	To reposition the entire shape you chose. Right-click again to rotate the chosen shape by choosing <i>Rotate</i> from the pop-up menu that displays.
<i>Copy</i>	To duplicate the entire shape you chose.
<i>Copy to Layers</i>	To replicate a shape on the chosen subclass layers.
<i>Edit Boundary</i>	To redefine the boundary of the copper area shape or its voids.
<i>Defer Dynamic Fill</i>	To prevent the currently chosen shape from dynamically updating. Disabling this field dynamically updates the shape you are currently working with and filled if the shape boundary exists. This field is disabled for unfilled and non etch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later.
<i>Change Shape Type</i>	To modify a shape fill type from <i>Static Solid</i> to <i>Dynamic Copper</i> or visa versa.
<i>Assign Net</i>	To attach the shape to a net by specifying a net name.
<i>Assign Region</i>	To attach the shape to a region.
<i>Raise Priority</i>	To assign 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.
<i>Parameters</i>	To specify parameter settings for the currently chosen shape.
<i>Report</i>	To generate a Dynamic Shape report.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Editing User-defined Manual Voids

To edit user-defined manual voids:

1. Choose *Shape – Select Shape or Void/Cavity* ([shape select](#) command). The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose a void to edit by clicking on it. Handles appear on the chosen void. The command window prompt displays the following message:

Pick vertex or segment to edit

3. Left mouse click to choose the vertex or segment to edit.

4. To complete the void boundary, left, double, or right-click to choose any of the following from the pop-up menu:

<i>Done</i>	To exit the command.
<i>Oops</i>	To back up to the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To edit another void.
<i>Reject</i>	To reject or undo the edits you made.
<i>Delete Vertex</i>	To delete the selected vertex.
<i>Move</i>	To reposition the entire shape you chose. Right-click again to rotate the chosen shape by choosing <i>Rotate</i> from the pop-up menu that displays.
<i>Copy</i>	To duplicate the entire shape you chose.
<i>Copy to Layers</i>	To replicate a shape on the chosen subclass layers.
<i>Edit Boundary</i>	To redefine the boundary of the copper area shape or its voids.
<i>Defer Dynamic Fill</i>	To prevent the currently chosen shape from dynamically updating. Disabling this field dynamically updates the shape you are currently working with and filled if the shape boundary exists. This field is disabled for unfilled and non etch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later.
<i>Change Shape Type</i>	To modify a shape fill type from <i>Static Solid</i> to <i>Dynamic Copper</i> or visa versa.
<i>Assign Net</i>	To attach the shape to a net by specifying a net name.
<i>Assign Region</i>	To attach the shape to a region.
<i>Raise Priority</i>	To assign 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.
<i>Parameters</i>	To specify parameter settings for the currently chosen shape.
<i>Report</i>	To generate a Dynamic Shape report.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Changing the Net Assigned to the Shape

To change the net assigned to s shape, perform the following steps:

1. Choose *Shape – Select Shape or Void/Cavity* ([shape select](#) command). The command window prompt displays the following message:

Pick a shape or void to edit

2. 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

3. Attach the shape to a net by specifying a net name in the *Assign Net Name* field, choosing a net name from the dropdown list, right-clicking to choose *Assign Nets* from the pop-up menu that displays, or clicking... to display the *Select Nets* dialog box from which you can choose a net.

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

4. Right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up to the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To work with another shape.
<i>Reject</i>	To reject or undo the edits you made.
<i>Delete Vertex</i>	To delete the selected vertex.
<i>Move</i>	To reposition the entire shape you chose. Right-click again to rotate the chosen shape by choosing <i>Rotate</i> from the pop-up menu that displays.
<i>Copy</i>	To duplicate the entire shape you chose.
<i>Copy to Layers</i>	To replicate a shape on the chosen subclass layers.
<i>Edit Boundary</i>	To redefine the boundary of the copper area shape or its voids.
<i>Defer Dynamic Fill</i>	To prevent the currently chosen shape from dynamically updating. Disabling this field dynamically updates the shape you are currently working with and filled if the shape boundary exists. This field is disabled for unfilled and non etch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later.
<i>Change Shape Type</i>	To modify a shape fill type from <i>Static Solid</i> to <i>Dynamic Copper</i> or visa versa.
<i>Assign Net</i>	To assign nets.
<i>Assign Region</i>	To attach the shape to a region.
<i>Raise Priority</i>	To assign 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.
<i>Parameters</i>	To specify parameter settings for the currently chosen shape.
<i>Report</i>	To generate a Dynamic Shape report.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Changing a Filled Rectangle to a Shape

To change a filled rectangle to a shape, follow these steps:

1. Choose *Shape – Select Shape or Void/Cavity* ([shape select](#) command). The command window prompt displays the following message:
Pick a shape or void to edit
2. Choose the graphic element by clicking on it. Handles appear on the chosen element, which has automatically been converted to a shape.
3. Right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up to the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To work with another shape.
<i>Reject</i>	To reject or undo the edits you made.
<i>Delete Vertex</i>	To delete the selected vertex.
<i>Move</i>	To reposition the entire shape you chose. Right-click again to rotate the chosen shape by choosing <i>Rotate</i> from the pop-up menu that displays.
<i>Copy</i>	To duplicate the entire shape you chose.
<i>Copy to Layers</i>	To replicate a shape on the chosen subclass layers.
<i>Edit Boundary</i>	To redefine the boundary of the copper area shape or its voids.
<i>Defer Dynamic Fill</i>	To prevent the currently chosen shape from dynamically updating. Disabling this field dynamically updates the shape you are currently working with and filled if the shape boundary exists. This field is disabled for unfilled and non etch shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later.
<i>Change Shape Type</i>	To modify a shape fill type from <i>Static Solid</i> to <i>Dynamic Copper</i> or visa versa.
<i>Assign Net</i>	To assign nets.
<i>Assign Region</i>	To attach the shape to a region.
<i>Raise Priority</i>	To assign 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.
<i>Parameters</i>	To specify parameter settings for the currently chosen shape.
<i>Report</i>	To generate a Dynamic Shape report.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Reviewing Shape Instance Parameters

Follow these steps to review shape instance parameters:

1. Choose *Shape – Select Shape or Void/Cavity* ([shape select](#) command). The command window prompt displays the following message:

Pick a shape or void to edit

2. 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

3. Right-click to choose *Parameters* from the pop-up menu.

4. Modify shape fill, void clearances, and thermal-relief connect line parameters in the dialog box that appears.

 You can also review or edit shape parameters in the Design Parameter Editor. Choose *Setup – Design Parameters* ([prmed](#) command), then click the *Shapes* tab. You do not need to select a shape first to access the Design Parameter Editor.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Moving an existing shape or void

To reposition a selected shape or void:

1. Choose *Shape – Select Shape or Void/Cavity* ([shape select](#) command). The command window prompt displays the following message:
Pick a shape or void to edit
2. Right-click and choose *Move* from the pop-up menu that displays.
3. To rotate the chosen shape, right-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-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.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Deleting a Shape

Perform these steps to delete a shape:

1. Use *Edit – Delete* ([delete](#) command).
2. In the *Options* panel, ensure that Shapes is the only element checked.
3. Left click to choose a shape to delete.
PCB and Package Designer highlights the chosen shape.
4. Right-click and choose *Done* from the pop-up menu.
PCB and Package Designer removes the shape from the design.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Moving a Segment

Follow these steps to move a segment:

1. Choose *Shape – Select Shape or Void/Cavity* ([shape select](#) command). The command window prompt displays the following message:

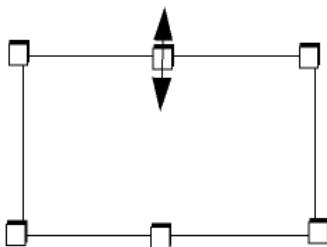
Pick a shape or void to edit

2. 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

3. Choose a segment. Edit handles then display on the newly chosen segment.

4. Left mouse click to move or resize a segment; the cursor shape changes as it moves over the shape or void with edit handles, as shown in the following figure.



5. Right-click to choose any of the options from the pop-up menu.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Running a Dynamic Shape Report

You can generate a dynamic shape report using *Tools – Reports* ([reports](#)) command. The report 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.

1. Choose *Shape – Select Shape or Void/Cavity* ([shape select](#)) command. The command window prompt displays the following message:

Pick a shape or void to edit

2. 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

3. Right-click to display the pop-up menu and choose *Reports*.

4. Choose *Dynamic Shapes* from the list.

5. Click *Report*.

The report displays onscreen.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

Identifying Out-of-date Dynamic Shapes

You can identify out-of-date dynamic shapes by following these steps:

1. Run the [status](#) command.
2. Click the *Out of Date Shapes* color box on the *Status* tab to produce a report, sorted by layer, showing the status of each dynamic shape on the board as follows:
 - Smooth: Ready for artwork.
 - Out of date: Update required.
 - No Etch: Shape has no etch, possibly due to a route keepout. Delete the dynamic shape or add etch to produce artwork.

Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

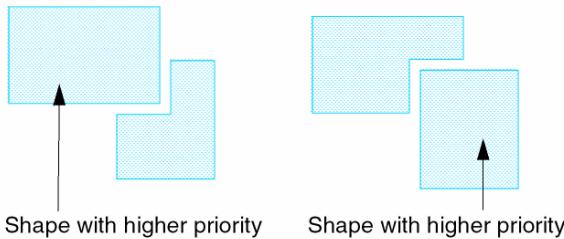
Assigning a Higher Voiding Priority to a Dynamic Shape

You can assign a higher voiding priority to a dynamic shape by following these steps:

1. Choose *Shape – Rectangular* (`shape add rect` command), *Shape – Polygon* (`shape add` command), or *Shape – Circular* (`shape add circle` command) to create a dynamic shape.
2. Choose *Shape – Select Shape or Void/Cavity*, then right-click and choose *Raise Priority* from the pop-up menu that appears. The command window prompt displays the following message:

Pick dynamic shape

3. Click on the overlapping dynamic shape to which to assign a higher priority.
The shape's priority is immediately updated as illustrated in the following figure.



Related Topics

- [shape select](#)
- [Shape Select Command: Options Panel](#)
- [Shape Copy to Layers Dialog Box](#)
- [Shape Select Command Tasks](#)

shape to cline

The `shape to cline` command converts shapes to clines. For example, you can convert DFX or GDSII geometries or shapes to clines using this command.

Related Topics

- [Converting Shapes to Clines](#)

Shape To Cline Command: Options Panel

Access Using

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

Remove converted shapes	Removes converted shapes and only retains the clines. Selected by default.
Merge shapes before converting	Merges overlapping shapes before converting them into clines. For example, if you select three shapes to be merged with this option selected and two of the shapes are overlapping, two clines will be created; one for the merged overlapping shapes and one for the shape that is not overlapping. Not selected by default.
Subtract endcap length from segments	Subtracts endcap length from segments. Not selected by default. The following images shows two clines converted from shapes of the same size, the first without selecting the option and the later one with the option selected. 
Create cline on shape layer	Creates clines on the shape layer. You can remove the selection and choose a layer from the list. Selected by default.

Converting Shapes to Clines

To convert shapes to clines:

1. Choose *Tools – Convert – Shape to Cline*.
2. Configure the Options pane.
3. Select the shapes to convert.
4. Choose *Done* from the pop-up menu

Related Topics

- [shape to cline](#)

shape to padstack

The `shape to padstack` command converts shapes to padstacks. You can create vias for the padstack and add the padstacks to the physical constraint list. For example, you can convert DFX or GDSII geometries or shapes to padstack using this command.

You can select multiple shapes on different layers to create a multi-layer via padstack; for example, spanning from top to bottom.

Related Topics

- [Converting Shapes to Padstacks](#)

Shape To Padstack Command: Options Panel

Access Using

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

Create via and remove shapes	Removes selected shapes after creating vias. Selected by default.
Add padstack to via list	Adds created padstacks to the physical constraints list. Selected by default.
Convert shape voids to mask layer flash	Creates pad with degassing holes. If shape has voids the outer boundary is converted into the filled metal regular pad on the layer and the voids are converted into a flash pad and placed on the appropriate user-defined mask layer linked to this layer. Not selected by default.
Shape name	Specifies the name of the pad shape.
Padstack name	Specifies the padstack name.
Layer	Specifies the layer the padstack has to be created on.
Origin	Specifies the origin of the padstack, in terms of <i>Center</i> , <i>Top Left</i> , <i>Top Right</i> , <i>Bottom Left</i> , or <i>Bottom Right</i> . For circular shape only <i>Center</i> is allowed. By default, <i>Center</i> is selected.

Converting Shapes to Padstacks

To convert shapes to padstacks:

1. Choose *Tools – Convert – Shape to Padstack*.
2. Configure the Options pane.
3. Select the shapes to convert.
4. Choose *Done* from the pop-up menu

Related Topics

- [shape to padstack](#)

shape to via

The `shape to via` command converts shapes, lines, or clines to vias or bond fingers. You have the option of converting all matching lines, clines, or shapes to vias or bond fingers. For example, you can convert DFX or GDSII geometries or shapes to vias using this command.

 The padstacks for the vias being created must already exist in the database. This replaces shapes that match the parameters with vias or fingers using a pre-defined via padstack. To create a padstack from the geometry, run the `shape to padstack` command first.

Related Topics

- [Converting Shapes to Vias](#)

Shape To Via Command: Options Panel**Access Using**

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

Convert matching shapes	Specifies that all matching shapes should be converted to vias or bond fingers. Selected by default.
Convert matching lines	Specifies that all matching lines should be converted to vias or bond fingers. Selected by default.
Convert matching clines	Specifies that all matching clines should be converted to vias or bond fingers. Selected by default.
Diameter	Specifies the diameter of the selected shape. You can edit this value.
Layer	Specifies the layer of the selected shape.
Via padstack	Specifies the padstack for the vias.
Create bond finger	Creates a bond finger instead of a via. Not selected by default.
Replace items	Converts the selected shapes and, if specified, all matching shapes, clines, or lines according to the configured options.

Converting Shapes to Vias

To convert shapes to vias:

1. Choose *Tools – Convert – Shape to Via*.
2. Configure the Options pane.
3. Select a shape to convert.
4. Click *Replace items*

Related Topics

- [shape to via](#)

shape vertex add

An internal Cadence engineering command.

shape void circle

The `shape void circle` command lets you create a circular element within an etch/conductor shape that is recognized as unfilled during penplotting and photoplotting. Use this command only when your design calls for etch/conductor connect lines to be on the same layer as an etch/conductor shape. Use this procedure to add a circular void area in a conductor shape.

 You can also create a circular shape within route or via keepout areas to allow routing in the void.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Adding a User-defined Circular Void to a Shape](#)

Shape Void Circle Command: Options Panel

Access Using

- Menu Path: *Shape – Manual Void/Cavity– Circular*



- Toolbar Icon:

Active Class and Subclass	Choose the proper etch/conductor layer. Color boxes in the subclass section align with the etch/conductor color on that particular subclass.
Shape fill	Disabled for this command.
Type	Disabled for this command.
Defer performing dynamic fill	Choose to prevent the currently chosen void from dynamically updating. Disabling this field causes the void you are currently adding to be dynamically updated and filled if the void boundary exists. This field is disabled for unfilled and non-etch/conductor shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <i>Esc</i> key to stop the process.
Assign net name	Enter a net to assign to the void, choose a net from the dropdown list, or click ... to display the <i>Select Net</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using identify nets (<i>Logic – Identify DC Nets</i>). Changing an assigned net dynamically fills and updates the void. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current void, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
Shape grid	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by running the define grid command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None</i> : Choose to create shapes off grid in user units, specified on the <i>Drawing Parameters</i> dialog box (see Creating Drawing Parameters). <i>Current Subclass Grid</i> : Choose to use the grid values defined for the active class/subclass. This is the default value.
Circular Shape Creation	Choose to set circular shape creation mode. <i>Draw Circle</i> : Choose to draw circular shape. This option is selected by default. <i>Place Circle</i> : Choose to place the circular shape of known radius by setting the <i>Radius</i> value. <i>Center/Radius</i> : Choose to create the circular shape with known center and radius by setting the <i>Radius</i> and <i>Center</i> values in the field below. <i>Create</i> : Click to create the circular shape. <i>Radius</i> : Set radius of the circular shape. <i>Center</i> : Set the origin of the circular shape.

Adding a User-Defined Circular Void to a Shape

Follow these steps to add a user defined circular void to a shape:

1. Verify the active class and subclass are correct.
2. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field.
3. Run `shape void circle`. The command window prompt displays the following message:

Pick a shape or void to edit

4. Choose a shape by clicking on it. The command window prompt displays the following message:

Pick void coordinates

- Choose options in the *Circular Shape Creation* to create the circular shape.

Option	Description
<i>Draw Circle</i>	<ol style="list-style-type: none"> Specify the center of circular shape by moving the cursor to the position where you want to be the circle center, and left click. The coordinates of the center are updated in the <i>Options</i> panel. Specify the radius of circular shape by moving cursor to the position and left click. The value of the radius of the circular shape is updated in the <i>Options</i> panel.
<i>Place Circle</i>	<ol style="list-style-type: none"> Specify the radius of circular shape in the <i>Radius</i> field in the <i>Options</i> panel. The circular shape is attached to the cursor. Left click to place the circular shape. The coordinates of the center are updated in the <i>Options</i> panel.
<i>Center/Radius</i>	<ol style="list-style-type: none"> Specify the center of circular shape in the <i>Center</i> field in the <i>Options</i> panel. You can also specify the center by moving the cursor you want to be the circle center, and left click. The coordinates of the center are updated in the <i>Options</i> panel. Specify the radius of circular shape in the <i>Radius</i> field in the <i>Options</i> panel or move the cursor to the position, and left click. The value of the radius of the circular shape is updated in the <i>Options</i> panel. Choose <i>Create</i> to add the circular shape with specified radius.

Right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Select Shape</i>	To complete the shape and make it selected for editing.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

Related Topics

- [shape void circle](#)

shape void copy

The `shape copy void` command copies a user-defined void that you chose in the active shape.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Copying Voids in Rectangular Patterns](#)
- [Copying Voids in Radial Patterns](#)

Shape Void Copy Command: Options Panel

Access Using

- Menu Path: *Shape – Manual Void/Cavity – Copy*

Use the fields on the *Options* panel to control how voids are copied. The choices vary depending on whether you set *Copy mode* to *Rectangular* or *Polar*.

<i>Copy mode</i>	Specifies the type of pattern you are copying. These are the choices:
<i>Rectangular</i>	Generates copies or adds voids in a rectangular grid array.
<i>Polar</i>	Generates copies around a user-defined point in angular increments.
Rectangular Options	
<i>Qty X</i>	Defines the number of columns to be created. When you set this field and the <i>Qty Y</i> field to 1, the layout editor places the elements in the cursor buffer to be placed dynamically.
<i>Qty Y</i>	Defines the number of rows to be created. When you set this field and the <i>Qty X</i> field to 1, the layout editor places the elements in the cursor buffer to be placed dynamically.
<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, as measured from the point on the element indicated by the <i>Copy origin</i> field. <i>Spacing Y</i> indicates the amount of distance in user units between items in a column, measured from the point on the element indicated by the <i>Copy origin</i> field.
<i>Order</i>	Indicates the direction in which the layout editor should create the copies in each row and column. <i>Order X</i> specifies the direction that rows are copied. <i>Right</i> is the default. <i>Order Y</i> specifies the direction that columns are copied. <i>Down</i> is the default.
<i>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 layout editor 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 layout editor snaps the figure to the nearest 90-degree increment.
<i>Copy origin</i>	Indicates the point on the element used to calculate distance for rows and columns. If you rotate the element before pasting it, this is also the point about which the element is rotated. The choices are:
	<i>Symbol</i> The 0,0 point of the element.
	<i>Body Center</i> The point at the center of an invisible boundary that the layout editor draws around the edge of the symbol.
	<i>User Pick</i> A point you clicked with the mouse.
	<i>Sym Pin #</i> A pin number you choose. You specify the number in the <i>Symbol pin #</i> field.
<i>Retain net of vias</i>	Keeps the net name assigned to a via when it is copied to a new location for applications such as GND stitching for example.
<i>Symbol pin #</i>	Specifies a pin number as the element's origin. Appears when you set <i>Copy origin</i> to <i>Sym Pin #</i> .
Polar Options	
<i>Direction</i>	Specifies a direction for the copied elements: <i>Cww</i> (counter-clockwise) or <i>Cw</i> .(clockwise).
<i>Copies</i>	Specifies how many copies of the element the layout editor makes. When set to 1 (the default), the layout editor places the element(s) in the cursor buffer to be placed dynamically.

<i>Angle</i>	Specifies the incremental angle to be used when placing the copies around the origin. Enter a number between 0 and 360 or choose a number from the pop-up. The layout editor allows accuracy for up to three decimal places.
<i>Copy origin</i>	Identifies the point on the element and each copy where the radius from the polar origin should intersect. The choices for this field are described previously in the table.

Related Topics

- [Copying Voids in Radial Patterns](#)

Copying Voids in Rectangular Patterns

Perform these steps to copy voids in rectangular patterns:

1. Run `shape void copy`. The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose a shape by clicking on it. The command window prompt displays the following message:

Pick Origin

3. In the *Options* panel, choose *Rectangular* and complete the other entries.
4. Adjust the Find filter as necessary.
5. Choose the element(s) to be copied. For a group of elements, you can use *Temp Group* from the pop-up menu or drag the cursor over an area to choose a group.
6. To rotate the elements, click right to display the pop-up menu and choose *Rotate*; then click to lock the elements in that position.
7. To relocate geometry to the opposite side of the board/substrate, click right to display the pop-up menu and choose *Mirror Geometry*. A mirror image of the element(s) displays at the cursor.
 - a. Position the mirrored element(s) and click to place it on the same subclass.
 - b. After selecting an element, click right and use the *Rotate* command on the pop-up menu to change the orientation of the mirrored element before placing it on the design.
 - c. Choose the new location for the element(s). Each mouse click pastes another copy of the element(s) on your design.
8. From the pop-up menu, choose *Done*.

Related Topics

- [shape void copy](#)

Copying Voids in Radial Patterns

Perform these steps to copy voids in radial patterns:

1. Run `shape void copy`. The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose a shape by clicking on it. The command window prompt displays the following message:

3. Select the elements to copy

4. In the *Options* panel, choose *Polar* and complete the other entries.

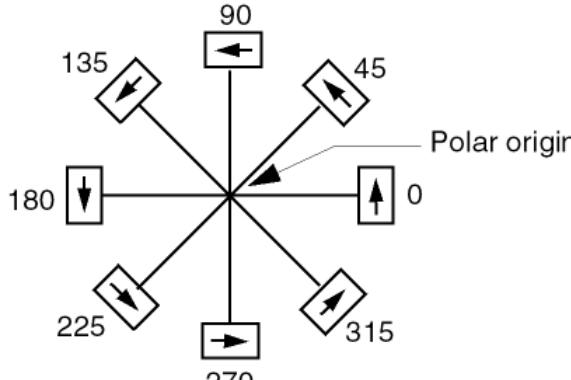
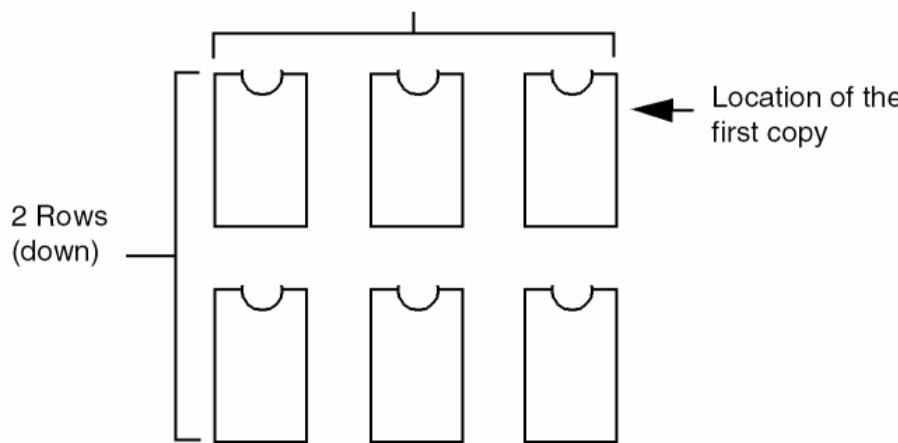
5. Adjust the Find filter as necessary.

6. Choose the element(s) to be copied. For a group of elements, you can use *Temp Group* from the pop-up menu or drag the cursor over an area to choose a group.

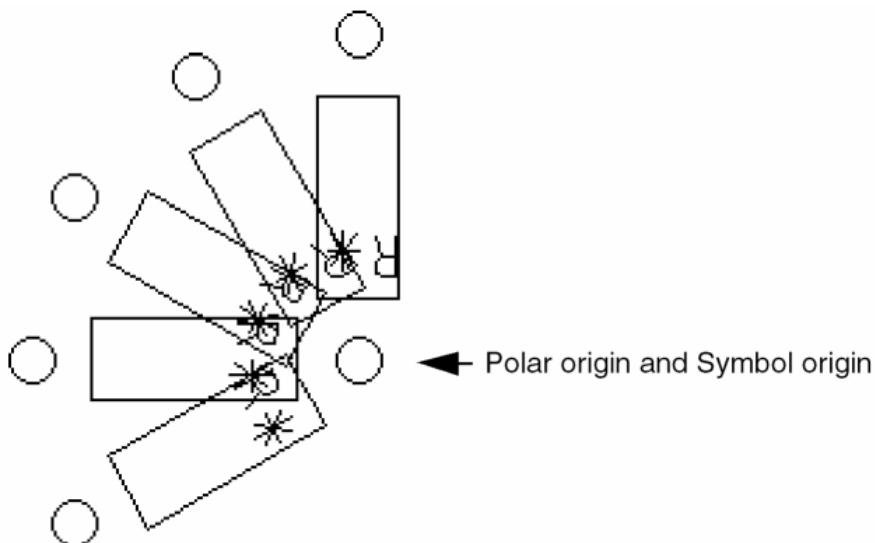
7. Click to specify the origin for the copied elements.

8. Choose the new location for the element(s). Each mouse click pastes another copy of the element(s) on your design. The layout editor creates a pattern of copies around the point of origin.

9. From the pop-up menu, choose *Done*.

Example	Figure	Description
Incremental Angles		The figure uses an angle value of 45 degrees and shows the possible locations for this setting
Copied Elements in a Rectangular Pattern	<p style="text-align: center;">3 Columns (left)</p>  <p>2 Rows (down)</p>	The figure shows the number of columns and direction is 3, Left and the number of rows and direction is 2, Down.

Copied Elements in a Radial Pattern



The figure shows radial pattern of copies generated by the layout editor.

Related Topics

- [shape void copy](#)
- [Shape Void Copy Command: Options Panel](#)

shape void delete

The `shape void delete` command deletes voids that you chose in the active shape.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Deleting a Void in a Shape](#)
- [Deleting All Voids](#)
- [Deleting Island Voids](#)

Shape Void Delete Command: Options Panel

Access Using

- Menu Path: *Shape – Manual Void/cavity – Delete*

Active Class and Subclass	Choose the proper etch/conductor layer. Color boxes in the subclass section align with the etch/conductor color on that particular subclass layer.
Shape fill	Disabled for this command.
Type	Disabled for this command.
Defer performing dynamic fill	Choose to prevent the currently chosen void from dynamically updating. Disabling this field causes the void you are currently adding to be dynamically updated and filled if the void boundary exists. This field is disabled for unfilled and non-etch/conductor shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <code>Esc</code> key to stop the process.
Assign net name	Enter a net to assign to the void, choose a net from the dropdown list, or click ... to display the <i>Select Net</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using identify nets (<i>Logic – Identify DC Nets</i>). Changing an assigned net dynamically fills and updates the void. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current void, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
Shape grid	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by running the define grid command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None</i> : Choose to create shapes off grid in user units, specified on the <i>Drawing Parameters</i> dialog box (see Creating Drawing Parameters). <i>Current Subclass Grid</i> : Choose to use the grid values defined for the active class/subclass. This is the default value.
Segment type	Disabled for this command.
Type	Disabled for this command.
Angle	Disabled for this command.
Arc radius	Disabled for this command.

Related Topics

- [Deleting All Voids](#)
- [Deleting Island Voids](#)

Deleting a Void in a Shape

To delete a void in a shape:

1. Run `shape void delete`. The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose a shape by clicking on it.
3. Click on the void, and it is deleted.

Related Topics

- [shape void delete](#)
- [Deleting Island Voids](#)

Deleting All Voids

To delete all void in a design, follow these steps:

1. Run `shape void delete`. The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose a shape by clicking on it.
3. Right-click to choose *Delete All Voids* from the pop-up menu that displays. All user-defined voids in a dynamic shape and all voids in a static shape are deleted.
4. Right-click to choose any of the following from the pop-up menu:

Done to exit the command.

Oops to back up to the last pick.

Cancel to delete any modifications made during this session.

Temp Group to choose a collection of elements.

Reject to undo the current selection and choose another element from those that are near the selection pick location.

Delete All Voids to remove all voids for the entire chosen shape.

Related Topics

- [shape void delete](#)
- [Shape Void Delete Command: Options Panel](#)

Deleting Island Voids

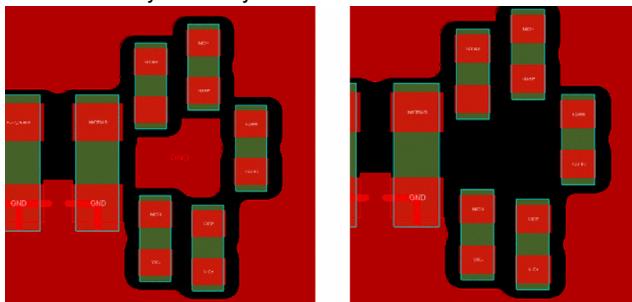
To delete island voids, perform these steps:

1. Run `shape void delete`. The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose a shape by clicking on it.
3. Right-click to choose *Delete Island Voids* from the pop-up menu that displays.

All voids are dynamically deleted and the number of the deleted voids are displayed in the command window.



Shape with Islands

Shape when Islands are deleted

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

Related Topics

- [shape void delete](#)
- [Shape Void Delete Command: Options Panel](#)
- [Deleting a Void in a Shape](#)

shape void element

The `shape void element` command lets you choose a pin or via and automatically creates an unfilled clearance hole for static (manual) shapes. You can also automatically generate voids for a positive shape by right-clicking and choosing *Void All* from the pop-up menu that displays.

Use this command only when your design calls for etch/conductor connect lines to be on the same layer as an etch/conductor shape. If the shape is on a negative etch/conductor layer, do not generate voids automatically. In Allegro X Advanced Package DesignerL, use this procedure to add a void around an element in a conductor shape.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Generating Voids for Static Positive Shapes on Positive Etch/Conductor Layers Automatically](#)
- [Debugging Improperly Voiding Vias In Dynamic Plane Shapes](#)

Shape Void Element Command: Options Panel

Access Using

- Menu Path: *Shape – Manual Void/Cavity – Element*

<i>Active Class and Subclass</i>	Choose the proper etch/conductor layer. Color boxes in the subclass section align with the etch/conductor color on that particular subclass layer.
<i>Shape fill</i>	Disabled for this command.
<i>Type</i>	Disabled for this command.
<i>Defer performing dynamic fill</i>	Choose to prevent the currently chosen void from dynamically updating. Disabling this field causes the void you are currently adding to be dynamically updated and filled if the void boundary exists. This field is disabled for unfilled and non-etch/conductor shapes.
	If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later.
	To cancel dynamic filling of complex shapes for a large design, you can use the <code>Esc</code> key to stop the process.
<i>Assign net name</i>	Enter a net to assign to the void, choose a net from the dropdown list, or click ... to display the <i>Select Net</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using identify nets (Logic – Identify DC Nets). Changing an assigned net dynamically fills and updates the void. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current void, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
<i>Shape grid</i>	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by running the <code>define grid</code> command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None</i> : Choose to create shapes off grid in user units, specified on the <i>Drawing Parameters</i> dialog box (see Creating Drawing Parameters). <i>Current Subclass Grid</i> : Choose to use the grid values defined for the active class/subclass. This is the default value.
<i>Segment type</i>	Disabled for this command.
<i>Type</i>	Disabled for this command.
<i>Angle</i>	Disabled for this command.
<i>Arc radius</i>	Disabled for this command.

Related Topics

- [Debugging Improperly Voiding Vias In Dynamic Plane Shapes](#)

Generating Voids for Static Positive Shapes on Positive Etch/Conductor Layers Automatically

You can generate voids for static positive shapes on positive etch/conductor layers automatically by following these steps:

1. Run `shape void element`. The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose a shape by clicking on it. The command window prompt displays the following message:

Pick element to be voided

3. Choose a pin or a via or an area to void by window.

To automatically generate voids for an entire positive shape, right-click to choose *Void All* from the pop-up menu. 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).

You can use *Edit – Undo* if the result is not desired, then reset shape parameters or change the shape outline to recreate a contiguous shape.

4. Right-click to choose any of the following from the pop-up menu:

Done to exit the command.

Oops to back up to the last pick.

Cancel to delete any modifications made during this session.

Temp Group to choose a collection of elements.

Reject to undo the current selection and choose another element from those that are near the selection pick location.

Complete to finalize the collection of elements chosen with Temp Group.

Void All to generate autovoiding for the entire chosen shape.

Parameters to specify parameter settings for the currently chosen shape.

Related Topics

- [shape void element](#)

Debugging Improperly Voiding Vias In Dynamic Plane Shapes

To debug improperly voiding vias in dynamic plane shapes:

1. Run the [status](#) command. On the *Status* tab, the *Fill mode* for *Shapes (Dynamic Copper Pour)* may be *Disabled*, and you may see a red box next to *Out of Date Shapes* along with the out of date count.
2. Click *Update to Smooth* to update the dynamic shapes, which will void all vias, pins, and clines accordingly.
3. Choose *Tools – Database Check* ([dbdoctor](#) command), and enable *Check Shape Outlines*.
4. Click *Viewlog* to review the [dbdoctor.log](#) containing the results of the database check.
5. Choose *Shape – Select Shape or Void/Cavity* ([shape select](#) command). The command window prompt displays the following message:

Pick a shape or void to edit

6. Choose the shape, right-click and choose *Move* from the pop-up menu that displays. Move the shape zero distance. (You can also use the *Edit – Move* ([fmove](#) 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.)
7. Right-click and choose *Done* from the pop-up menu that displays.
8. At The command window prompt, type:
`ix 0 <return>`.
9. Choose *Shape – Global Dynamic Params* ([shape global param](#) command) and verify/change the parameters on the *Clearances* tab from *Thermal/anti* to *DRC* or vice versa. Enter a value in the *Use Thermal Width Oversize of* field on the *Thermal Relief Connects* tab as required.
10. Delete a via and then place it again in the same X-Y location.
11. Choose *Shape – Change Shape Type* ([shape change type](#) command). On the *Options* panel, choose *To static solid* in the *Shape Fill Type* field.
12. Choose *Shape – Manual Void/Cavity – Element* ([shape void element](#) command) to void manually.

⚠ Performance problems are usually due to the number of shapes on a layer. When many dynamic shapes exist in the design, work in the *Disabled* shape fill mode until you are ready to generate artwork, and then choose *Update to Smooth* on the *Status* tab or on the *Shape Fill* tab of the *Global Dynamic Shape Parameters* dialog box.

Related Topics

- [shape void element](#)
- [Shape Void Element Command: Options Panel](#)

shape void move

The `shape void move` command moves a void that you have chosen in the active shape.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Moving an Existing Void](#)
- [Moving Multiple Voids](#)

Shape Void Move Command: Options Panel

Access Using

- Menu Path: *Shape – Manual Void/Cavity – Move*

<i>Ripup Etch</i>	Rips up any connection elements to the closest pin, T connection, or via, and sets <i>Stretch Etch</i> to <i>No</i> . If you do not choose this field, the program leaves all connections that are associated with the element on the board/substrate as dangling lines.
<i>Stretch Etch</i>	Rips up the first line segment of any connection attached to the element and adds an odd angle segment between the rotated element and the rest of the connection.
<i>Rotation Type</i>	Determines the mode of rotation. The options are: <i>Absolute</i> rotates the element once to place it at the angle specified in the <i>Angle</i> field. <i>Incremental</i> 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.
<i>Angle</i>	Determines the angle of rotation, but has a different meaning depending on the rotation mode: For <i>Incremental</i> mode, <i>Angle</i> specifies how many degrees comprise each increment as you rotate the element. For <i>Absolute</i> mode, <i>Angle</i> specifies the final degree of rotation from the 0,0 orientation. When you execute the command, the program immediately rotates the element to that angle. You can enter a number between 0 and 360, or you can choose one of the following numbers from the pop-up menu: 0, 45, 90, 135, 180, 215, 270, and 315. The program is accurate up to three decimal places.
<i>Point</i>	Indicates the anchor point around which the element turns. The options are: <i>Sym Origin</i> is the 0,0 point of the element (symbol). <i>Body Center</i> is the point at the center of an invisible boundary that the program draws around the edge of the element. <i>User Pick</i> is a mouse click or typed coordinates that indicate the point. <i>Sym Pin #</i> invokes a field where you enter the number. The <i>Symbol Pin #</i> field appears only when you choose <i>Sym Pin #</i> as the rotation point. Enter a pin number.

Related Topics

- [Moving Multiple Voids](#)

Moving an Existing Void

To move a void in the design, follow these steps:

1. Run `shape void move`. The command window prompt displays the following message:

Pick a shape or void to edit

2. Verify the entries in the *Options* panel.
3. Choose a shape by clicking on it. The command window prompt displays the following message:

Select element(s) to move

THE layout editor attaches the element to your cursor and

- o If the element has attached connect lines, ratsnest lines replace them.
- o If Stretch Etch is enabled in the *Options* panel, the segments that connect to the symbol/via appear as rubber bands and the reprobated lines go to the other end of the segment that was erased.
- o If Stretch Etch is disabled, the ratsnest lines reprobated (for example, from pin to pin on another symbol).

4. Choose the destination point.

The layout editor displays the element at its new location.

If any element had connected lines to other elements, the connected lines become permanently deleted, stretched and connected, or left alone depending on how the *Ripup Etch* and *Stretch Symbol Etch* fields are set in the *Options* panel.

5. Choose another element to be moved, or click right to display the pop-up menu, and choose *Done*.

Related Topics

- [shape void move](#)

Moving Multiple Voids

You can move multiple voids by following these steps:

1. Run `shape void move`. The command window prompt displays the following message:

Pick a shape or void to edit

2. Choose a shape by clicking on it. The command window prompt displays the following message:

Select element(s) to move

3. Verify the entries in the *Options* panel.
4. Click right to display the pop-up menu in the work area and choose *Temp Group*.
5. Choose each element in the group using the left mouse button.
6. When you have chosen all elements to move, click right to display the pop-up menu and choose *Complete*.
7. Identify a location as the origin of the entire group. The layout editor attaches the elements to your cursor. The command window prompt displays the following message:

Pick new location for element(s)

8. Identify the new location for the group or window of elements.
9. Choose another element or group of elements to move or click right to display the pop-up menu and click *Done*.

Related Topics

- [shape void move](#)
- [Shape Void Move Command: Options Panel](#)

shape void polygon

The `shape void polygon` command creates a non-copper polygon within the copper area.

 You can also create a polygon within route or via keepout areas to allow routing in the void.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Adding a Void Area in an Etch/Conductor Shape Interactively](#)

Shape Void Polygon Command: Options Panel

Access Using

- Menu Path: *Shape – Manual Void/Cavity – Polygon*

<i>Active Class and Subclass</i>	Choose the proper etch/conductor layer. Color boxes in the subclass section align with the etch/conductor color on that particular subclass layer.
<i>Shape fill</i>	Disabled for this command.
<i>Type</i>	Disabled for this command.
<i>Defer performing dynamic fill</i>	Choose to prevent the currently chosen void from dynamically updating. Disabling this field causes the void you are currently adding to be dynamically updated and filled if the shape boundary exists. This field is disabled for unfilled and non-etch/conductor shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <i>Esc</i> key to stop the process.
<i>Assign net name</i>	Enter a net to assign to the void, choose a net from the dropdown list, or click ... to display the <i>Select Net</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using identify nets (<i>Logic – Identify DC Nets</i>). Changing an assigned net dynamically fills and updates the void. Disabled if you choose a <i>Shape Fill Type</i> of Unfilled. If you do not assign a net to the current void, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current shape.
<i>Shape grid</i>	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by running the <i>define grid</i> command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None</i> : Choose to create shapes off grid in user units, specified on the <i>Drawing Parameters</i> dialog box (see Creating Drawing Parameters). <i>Current Subclass Grid</i> : Choose to use the grid values defined for the active class/subclass. This is the default value.
<i>Segment type</i>	The following line segment types are available:
<i>Type</i>	<i>Line</i> : Choose to use any angle line. <i>Line 45</i> : Choose to miter lines to a 45 degree at vertex locations. <i>Line Orthogonal</i> : Choose to create lines at 90 degree angles at vertex locations. <i>Arc</i> : Choose to create an arc. Available only when adding polygons. Once you enter an arc, this field automatically defaults to the previous line segment type specified in the <i>Type</i> field. Cursor position as it moves toward the arc end point determines arc direction (clockwise or counter clockwise).
<i>Angle</i>	Available only if you specified <i>Arc</i> as the line segment type in the <i>Type</i> field as an alternative to selecting the end point of the arc. Enter a value to create an arc from the start point with the specified angle. The arc is tangent to the start and end point, which determines the arc's direction.
<i>Arc radius</i>	Available only if you specified <i>Arc</i> as the line segment type in the <i>Type</i> field. Enter the next arc segment with a given radius. A zero value creates a tangent arc.

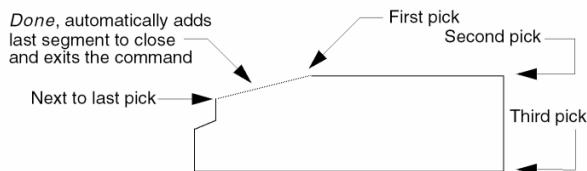
Adding a Void Area in an Etch/Conductor Shape Interactively

To add a void area in an etch/conductor shape interactively:

1. Verify the active class and subclass are correct.
2. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field.
3. Run `shape void polygon`. The command window prompt displays the following message:

Pick void coordinates

4. Left click at the vertices of the void outline that you want to create.



5. Click near the first pick to complete the void; then right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Assign Net</i>	To attach the shape to a net.
<i>Assign Region</i>	To attach the shape to a region.
<i>Arc</i>	To set the rubber band mode to arc.
<i>Snap pick to</i>	To snap your next mouse pick to the closest design element you choose from the sub-menu.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

Related Topics

- [shape void polygon](#)

shape void rectangle

The `shape void rectangle` command creates a non-copper rectangle within the copper area.

 You can also create a rectangular shape within route or via keepout areas to allow routing in the void.

For additional related information on working with shapes, see the *Preparing the Layout* user guide in your documentation set.

Related Topics

- [Adding a User-Defined \(Manual\) Rectangular Void to a Shape](#)

Shape Void Rectangle Command: Options Panel

Access Using

- Menu Path: *Shape – Manual Void/Cavity – Rectangular*

Active Class and Subclass	Choose the proper etch/conductor layer. Color boxes in the subclass section align with the etch/conductor color on that particular subclass layer.
Shape fill	Disabled for this command.
Type	Disabled for this command.
Defer performing dynamic fill	Choose to prevent the currently chosen void from dynamically updating. Disabling this field causes the shape you are currently adding to be dynamically updated and filled if the shape boundary exists. This field is disabled for unfilled and non-etch/conductor shapes. If <i>Defer dynamic fill</i> is disabled in the <i>Global Dynamic Shape Parameters</i> dialog box, subsequently created or modified shapes are not filled but marked as out of date to be filled later. To cancel dynamic filling of complex shapes for a large design, you can use the <i>Esc</i> key to stop the process.
Assign net name	Enter a net to assign to the void, choose a net from the dropdown list, or click ... to display the <i>Select Net</i> dialog box that lists all nets in the board design. The dropdown lists nets with a voltage property, assigned using <i>Identify nets</i> (<i>Logic – Identify DC Nets</i>). Changing an assigned net dynamically fills and updates the void. Disabled if you choose a <i>Shape Fill Type</i> of <i>Unfilled</i> . If you do not assign a net to the current void, the <i>Defer performing dynamic fill</i> field is automatically disabled for the current void.
Shape grid	Choose a grid increment for shape/void outlines or enter a value in database units. If the shape grid is set to <i>None</i> or to <i>Current Subclass Grid</i> , the subclass grid displays if you enable the <i>Grids On</i> field in the <i>Grids Display</i> dialog box, available by running the <i>Define Grid</i> command. If a shape grid is not entered, the grid for the current subclass is used. Up to five grid entries can be entered during any session. Exiting clears the grid settings from memory. Once the shape editing session ends, the working grid reverts back to the original database settings. <i>None</i> : Choose to create shapes off grid in user units, specified on the <i>Drawing Parameters</i> dialog box (see <i>Creating Drawing Parameters</i>). <i>Current Subclass Grid</i> : Choose to use the grid values defined for the active class/subclass. This is the default value.
Segment type	Disabled for this command
Type	Disabled for this command
Angle	Disabled for this command
Arc radius	Disabled for this command

Adding a User-Defined (Manual) Rectangular Void to a Shape

Perform these steps to add a user-defined rectangular void to a shape:

1. Verify the active class and subclass are correct.
2. Choose a grid increment for shape/void outlines or enter a value in database units in the *Shape Grid* field.
3. Run `shape void rectangle`. The command window prompt displays the following message:

Pick a shape or void to edit

4. Choose a shape by clicking on it. The command window prompt displays the following message:

Pick void coordinates

5. Left click and drag the cursor to the desired void size.
6. Left click near the first pick to complete the void; then right-click to choose any of the following from the pop-up menu:

Option	Description
<i>Done</i>	To exit the command.
<i>Oops</i>	To back up the last pick.
<i>Cancel</i>	To delete any modifications made during this session.
<i>Next</i>	To complete the shape and create another shape.
<i>Complete</i>	To continue editing the shape using the handles that display.
<i>Parameters</i>	To override global parameter settings and apply custom parameter settings to the shape with which you are currently working.

Related Topics

- [shape void rectangle](#)

shape zigzag

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

The `shape zigzag` command creates zigzag notches on the selected layers of the design. This fulfils one of the requirements of silicon manufacturing not to have long and parallel edges between wide metal areas of different nets.

Related Topics

- [shapeedit](#)
- [Creating Zigzag Patterns Between Long Metal Shapes](#)

Shape Zigzag Command: Options Panel

Access Using

- Menu path: *Si Layout – Shape Zigzag*

Generate zigzags	Creates zigzag notches on the selected layers with the given settings.
Remove zigzags	Removes any route keepouts tagged as zigzags in the design and restores the boundary of the dynamic shape to its original shape. For static shapes that have edges of the existing shape boundary or void boundary, the edge must be straightened manually.
Delete resolved DRC markers	Select this check box to remove the DRCs and markers on the DRC rule layer that has been resolved by the tool. This check box is deselected by default.
Add DRC markers at blocked locations	Select this check box to add DRCs where a needed notch cannot be added. This is useful if it is not possible to create a zigzag on a shape edge that has lot of vias along the edge of the shape. In such cases, the void cannot be placed. This check box is selected by default.
Update static shapes	Select this check box to force the shapes to cut out the holes instantly. This is helpful in case of static shapes that are only updated by the user when manually requested because cutouts in the shape's outline become permanent. This check box is selected by default.

Layer Table

Layer	Displays the name of the conductor and mask layers.
DRC Rule	Displays PVS DRC rules to be used on the corresponding layer.
Notch width	Specify the width of the notches to be created. Width is parallel to the shape edge that is being notched. The default value is 25um.
Notch depth	Specify the depth of the notches to be created for the corresponding layer. Depth is perpendicular to the shape edge that is being notched. The default value is 25um.
Max length	Specify the maximum straight edge length allowed. This ensures that the maximum notches are placed at most this far apart. The default value is 500um.
Max separation	Specify the maximum distance between two shapes' parallel edges that are required to be zigzagged. If the distance between the two edges is more than this value, they are allowed to be long, straight segments. Default value is 50um.
Remove DRCs	Removes the DRCs from the enabled layer's linked DRC layer when processing is complete. DRCs can also be removed manually using the edit commands or PVS interface.

Related Topics

- [shapeedit](#)

Creating Zigzag Patterns Between Long Metal Shapes

If a metal plane edge violates the max parallel length rule with an adjacent metal plane edge, the metal plane outline pattern needs to be removed by creating a zigzag pattern between the parallel metal shapes.

To create a zigzag pattern between two long or parallel metal shapes, follow these steps:

1. Choose *Si Layout– Shape Zigzag*.
2. Set the values for the zigzags in the *Layer* table of the *Options* panel.
3. Click *Generate Zigzags*.

Zigzag shapes are created between the long or parallel metal shapes.

Related Topics

- [browse drcs](#)
- [shapeedit](#)
- [shape zigzag](#)

shapeupdate

An internal Cadence engineering command.

shapeedit

The `shapeedit` command activates the *Shape Edit* application mode that enables you to easily edit the shape boundaries. You can slide shape segments with or without corners, move multiple segments, and add notches to the shapes. When in the *Shape Edit* application mode, you can perform operations on shapes using *Options* panel in conjunction with the context-sensitive menus.

You can access the `shapeedit` command in one of the following ways:

- Choose Setup – Application Mode – Shape Edit.
- Right-click on the canvas and choose Application Mode – Shape Edit from the pop-up menu.
- Click the Shapeedit toolbar icon.
- Type `shapeedit` in the Console window and press `Enter`.
- Click the application mode in the status bar, and choose Shape edit from the pop-up menu.

For more information on using the shape edit application mode, see the [Getting Started with Physical Design](#) user guide in your documentation set.

Related Topics

- [Shape Edit Tasks](#)

Shape Edit Application Mode: Options Panel

Access Using

- Menu Path: Setup – Application Mode – Shape Edit



- Toolbar Icon:

⚠ By default, the `shapeedit` command selects segment. You can select a shape by choosing one of the following:

- Use the `TAB` key to navigate between shape and segments.
- Right-click on a segment and choose Shape.

<i>Enable zone boundary editing</i>	Enables zones as shapes for editing.
<i>Segment Commands</i>	
<i>Click</i>	Applies command on a segment using single-click (or double- click). You can select one of the following commands: <ul style="list-style-type: none"> ◦ <i>Add Notch</i>: Select to add a notch on a segment. ◦ <i>Move</i>: Select to move a segment. ◦ <i>Slide</i>: Select to slide a segment. ◦ <i>Remove/Extend</i>: Select to remove or extend a segment. ◦ <i>None</i>: Select if you do not want to apply a command
<i>Drag</i>	Applies command on a segment using drag. You can select one of the following commands: <ul style="list-style-type: none"> • <i>Move</i>: Select to move a segment. • <i>Slide</i>: Select to slide a segment. • <i>None</i>: Select if you do not want to apply a command
<i>Vertex Commands</i>	
<i>Click</i>	Applies command on a vertex using single click (or double click). You can select one of the following commands: <ul style="list-style-type: none"> ◦ <i>Move</i>: Select to move a vertex. ◦ <i>Chamfer/Round</i>: Select to chamfer and round the corners of a shape. ◦ <i>Delete</i>: Select to delete the vertex. ◦ <i>None</i>: Select if you do not want to apply a command
<i>Drag</i>	Applies command on a vertex using drag. You can select one of the following commands: <ul style="list-style-type: none"> ◦ <i>Move</i>: Select to move a vertex. ◦ <i>None</i>: Select if you do not want to apply a command.
<i>Slide</i>	
<i>Extend Selection (hold Shift to toggle)</i>	Select to extend a segment with chamfered corners.
<i>Auto Join (hold Ctrl to toggle)</i>	Select to automatically join dissimilar segments.
<i>Move</i>	
<i>Free Vertex</i>	Select to move vertex.

S Commands

S Commands--shapeedit

Corners	
Chamfer	Select to chamfer the corners. You can select one of the following options: <ul style="list-style-type: none">◦ <i>Trim (T)</i>: Specifies the trim length.◦ <i>Chamfer(C)</i>: Available if the Chamfer option is selected. Specifies the chamfered length.
Round	Select to round off the corners. You can select one of the following options: <ul style="list-style-type: none">◦ <i>Trim (T)</i>: Specifies the trim length.◦ <i>Round (R)</i>: Available if the Round option is selected. Specifies the round off length.
<i>Set trim size by cursor</i>	Select to trim the corners using the cursor.

Shape Edit Tasks

You can perform the following tasks using the `shapeedit` command:

- Sliding a Segment
- Extending a Segment
- Adding a Notch
- Moving Vertex
- Chamfering Corners
- Rounding Corners
- Chamfering All Corners
- Rounding All Corners
- Joining Segments
- Sliding Multiple Segments

Related Topics

- [shapeedit](#)

Sliding a Segment

To slide a segment:

1. Choose a shape segment.
2. Right-click and choose Slide Segment.
3. Drag the mouse to slide the segment to the required side and click.

You can also slide a segment using the options available in the Options panel. To slide a segment, use one of the following methods:

1. In Segment Commands, choose Slide in the Click field.
2. Drag the mouse to slide the segment to the required side and click.

Or

1. In Segments Commands, choose Slide in the Drag field.
2. Drag the mouse to slide the segment to the required side and click.

Related Topics

- [shapeedit](#)
- [Shape Edit Application Mode: Options Panel](#)
- [Shape Edit Tasks](#)

Extending a Segment

You can extend a segment with and without chamfered corners.

To extend a segment with chamfered corners:

1. In Segment Commands, choose Slide in the Click and Drag fields.
2. Select the Extend Selection option in the Slide section.
3. Choose a shape segment.
4. Drag the mouse to slide the segment to the required side and click.

To extend a segment without chamfered corners:

1. In Segment Commands, choose Slide in the Click and Drag fields.
2. Uncheck the Extend Selection option in the Slide section.
3. Choose a shape segment.
4. Drag the mouse to slide the segment to the required side and click.

Related Topics

- [shapeedit](#)
- [Shape Edit Application Mode: Options Panel](#)
- [Shape Edit Tasks](#)

Adding a Notch

You can add a notch on a straight segment and extend it inward or outward by following these steps:

1. In Segment Commands, select Add notch in the Click field.
2. Click on a segment.
A small square appears on the segment.
3. Take the cursor to a different location on the same segment and click.

 The gap between the two square boxes defines the notch width.

4. To add a notch, choose one of the following:
 - To define the notch inside the shape, drag the mouse inside the shape to the required shape and click.
 - To define the notch outside the shape, drag the mouse outside the shape to the required shape and click.

Related Topics

- [shapeedit](#)
- [Shape Edit Application Mode: Options Panel](#)
- [Shape Edit Tasks](#)

Moving Vertex

You can increase the width and length of a shape by moving the vertex of the shape:

1. Take the cursor to the bottom right corner of the shape.
The cursor shape changes.
2. Right-click and choose Move Vertex.
3. Drag the mouse to extend the shape to the required size and click.

Related Topics

- [shapeedit](#)
- [Shape Edit Application Mode: Options Panel](#)
- [Shape Edit Tasks](#)

Chamfering Corners

If the segments are at right angle, you can round off, that is, chamfered or trim the interior corner at forty five degrees. Follow these steps:

1. Select Chamfer/Round in the Click field.
2. Select Chamfer in the corners section.
3. Select Trim(T) and enter a value.
4. To trim, choose one of the following:
 - Click on a corner.
 - Right-click on a corner and choose Trim Vertex.

The segments are rounded off at forty five degrees.

5. Select Chamfer in the corners section.
6. To chamfer, choose one of the following:
 - Click on a corner.
 - Right-click on a corner and choose Trim Vertex.

The segments are chamfered as per the value defined in the Chamfer(C) field.

You can also trim the corners using the cursor.

1. Select a vertex location.
A moveable line appears at the vertex location.
2. Move the cursor to trim the vertex to the required size and click.

Related Topics

- [shapeedit](#)
- [Shape Edit Application Mode: Options Panel](#)
- [Shape Edit Tasks](#)

Chamfering All Corners

You can automatically chamfer all corners of a shape by performing these steps:

1. Select Chamfer in the Corners section.
2. Select Trim and enter a value.
3. Right-click on a segment and choose Shape – Trim Corners.

Related Topics

- [shapeedit](#)
- [Shape Edit Application Mode: Options Panel](#)
- [Shape Edit Tasks](#)

Rounding Corners

You can round off the exterior corner of a shape by following these steps:

1. Select Round in the Corners section.
2. Uncheck the Set Trim Size by cursor option.
3. To round off the corners, choose one of the following:
 - o Click on a corner.
 - o Right-click on a corner and choose Trim Vertex.

You can also change the corners to any types.

1. Select Radius(R) and enter a value.
2. Click on a ninety degree corner and continue to move the cursor unless you get the required type.
3. Click to round off.

Related Topics

- [shapeedit](#)
- [Shape Edit Application Mode: Options Panel](#)
- [Shape Edit Tasks](#)

Rounding All Corners

To automatically round off all corners of a shape, follow these steps:

1. Select Round in the Corners section.
2. Select Trim and enter a value.
3. Choose a segment.
4. Right-click on a segment and choose Shape – Trim Corners.

Related Topics

- [shapeedit](#)
- [Shape Edit Application Mode: Options Panel](#)
- [Shape Edit Tasks](#)

Joining Segments

To join segments, follow these steps:

1. Select Slide in the Click field.
2. Select Auto Join in the Slide section.
3. Click a vertical segment and move the cursor to the left side until you notice a joining line.
4. Click to join the segments.

Related Topics

- [shapeedit](#)
- [Shape Edit Application Mode: Options Panel](#)
- [Shape Edit Tasks](#)

Sliding Multiple Segments

You can slide dissimilar segments without interrupting their structure by following these steps:

1. Press Shift and select the segments.
2. Right-click and choose Move segment(s).
3. Move the cursor to slide the group of segments to the required side.
4. Click to finish.

Related Topics

- [shapeedit](#)
- [Shape Edit Application Mode: Options Panel](#)
- [Shape Edit Tasks](#)

shape freeze

The `shape freeze` command preserves the state of dynamic shapes without changing them to static shapes. This command prevents dynamic shapes from updating when interacting with other objects. As a result, it maintains design intent by protecting critical circuitry drawn using shapes. When a dynamic shape is frozen, new objects entering the dynamic shape area do not void and generate a DRC error similar to a static shape. You can manually modify shape boundaries on frozen dynamic shapes to maintain current voiding while avoiding new voids.

The command is also available as a context-sensitive menu option *Freeze Shape* when a shape is selected.

 The shape instance parameter cannot be modified for a frozen shape.

Use the `shape unfreeze` command to remove the frozen state of dynamic shapes.

Access Using

- Menu Path: *Shape – Freeze Shape(s)*

shape unfreeze

The `shape unfreeze` command removes the frozen state of dynamic shapes.

The command is also available as a context-sensitive menu option *Unfreeze Shape* when a shape is selected.

Access Using

- Menu Path: *Shape – Unfreeze Shape(s)*

shell

The `shell` command is used to open a window to the host operating system. The window provides full access to all host system utilities. You can manipulate the window opened with `shell` like any other port in the native windowing system. Additionally, you can move the window outside the boundary of your user interface and turn the window into an icon on the desktop.

Opening a Window to the Host Operating System

1. Type `shell` at The command window prompt.
A terminal window opens on your desktop.

shorting via array

You can use the `shorting via array` command to create shape-to-shape shorting via arrays, and to delete or update the arrays. The shapes selected must be on the same net but on different layers with overlapping areas.

Shape Shorting Via Array Dialog Box

Access Using

- Menu Path: *Manufacture – Shape Via Shorting*

Via array parameters	
Padstack	Lists the padstacks that start or stop between layers containing the selected shapes and present in the via list of Constraint Manager for at least one constraint set. If no shapes are selected, it lists all the padstacks in the design. If the current selection becomes invalid for the selected shapes, the first padstack in the list is selected based on alphabetical order.
Spacing	Specifies the air gap between vias in the via array. The value is calculated based on the via padstack and the pitch specified in the <i>Pitch</i> field. Specify a value that is at least equal to the via to via DRC constraint of the net.
Pitch	Specifies the center to center distance between vias in the shorting via array.
Array Angle	Specifies the angle of the array. By default the value is 0 degrees resulting in a grid structure.
Rotate vias at array angle	Specifies angle by which vias will be rotated. Use this for padstacks with non-circular pads to maintain consistent separation if Array Angle is not 0 degrees.
Starting Position	<p>Specifies the reference point for the via array. The available options are: <i>Upper Left</i>, <i>Upper Right</i>, <i>Lower Left</i>, <i>Lower Right</i>, and <i>Custom Point</i>.</p> <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <input checked="" type="checkbox"/> Use <i>Custom Point</i> to offset via arrays between different layer pairings or if multiple via arrays connect to a single plane shape. </div>
Offset X	Specifies the X value relative to the specified starting position.
Offset Y	Specifies the Y value relative to the specified starting position.
Custom Point X	Specifies the absolute database X value for the reference position of the via arrays. This is available if starting position is <i>Custom Point</i> .
Custom Point Y	Specifies the absolute database Y value for the reference position of the via arrays. This is available if starting position is <i>Custom Point</i> .
Via Spacing Constraints	
Use DRC constraint values	Causes the tool to use the spacing constraints specified in the DRC constraint for the database. If the addition of vias at a location results in new DRCs, vias will not be placed at that location.
Via to Shape Boundary	Specifies the value for the vias to clear the outline of the two shorted planes. Enabled only if <i>Use DRC constraint values</i> is not selected. The default is via to shape spacing on the top layer of the design.
Via to Conductor (Shape Layer)	Specifies the value for the vias to clear other routing objects on the same layer as the shapes being connected. Enabled only if <i>Use DRC constraint values</i> is not selected. The default is via to cline spacing on the top layer of the design.
Via to Conductor(Internal Layer)	Specifies the value for the vias to clear routing objects on all routing layers between the layers containing the shapes being connected. Enabled only if <i>Use DRC constraint values</i> is not selected. The default is via to cline spacing on the top layer of the design.
Update form values from saved shape pair settings	Updates the form with parameters stored in the database for the selected shapes. The parameters are stored in the database when a shorting via array is created.
Add Vias	Adds shorting array of vias between two selected shapes. Existing shorting vias, if any, are removed to prevent duplicate vias in the database.

S Commands
S Commands--shorting via array

Remove Vias	Removes shorting via array for specified shapes. If a single shape is selected, all shorting vias between the selected shape and any other shape will be removed. If two shapes are selected, only vias connecting the two shapes will be removed.
OK	Saves all changes and exits the dialog box.
Cancel	Exits without saving any changes.
Help	Displays the help topic for this command.

show allpanes

Restores the *Options*, *Worldview*, *Find*, *Visibility*, and *Command* foldable window panes to display in the positions in which you last viewed them. To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default* (`reset dockwindows` command).

Syntax

```
show allpanes
```

Access Using

- Menu Path: *View – Windows – Show All*

Displaying All Foldable Window Panes

1. Choose *View – Windows – Show All*.

The *Options*, *View*, *Find*, *Visibility*, and *Command* foldable window panes display in the positions in which you last viewed them.

showhide dcm

An internal Cadence engineering command.

showhide find

The `showhide find` command toggles the visibility of the *Find* panel.

A check mark next to *View – Windows – Find* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Find* panel by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Find* panel to expand it, or clicking the X to hide it.

 To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default* (`reset dockwindows` command).

For more information on the *Find* panel, see the *Getting Started with Physical Design* user guide in your documentation set.

Syntax

```
showhide find [show] [hide]
```

showhide_find	Displays or hides the pane, depending on its current state.
show	Displays the pane if it is hidden. If it is already visible, no action occurs.
hide	Hides the pane if it is visible. If it is already hidden, no action occurs.

Access Using

- Menu Path: *View – Windows – Find*

Controlling the Visibility of the Find Panel

1. Choose *View – Windows*.
 - a. To hide the pane, click *Find* if a check mark appears next to it.
 - b. To display the pane, click *Find* if no check mark appears next to it.

showhide options

The `showhide options` command toggles the visibility of the *Options* panel.

A check mark next to *View – Windows – Options* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Options* panel by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Options* panel to expand it, or clicking the X to hide it.

 To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default to Default* (`reset dockwindows` command).

For more information on the *Options* panel, see the *Getting Started with Physical Design* user guide in your documentation set.

Syntax

```
showhide options [show] [hide]
```

showhide_options	Displays or hides the pane, depending on its current state.
show	Displays the pane if it is hidden. If it is already visible, no action occurs.
hide	Hides the pane if it is visible. If it is already hidden, no action occurs.

Access Using

- Menu Path: *View – Windows – Options*

Controlling the Visibility of the Options Panel

1. Choose *View – Windows*.
 - a. To hide the pane, click *Options* if a check mark appears next to it.
 - b. To display the pane, click *Options* if no check mark appears next to it.

showhide text

The `showhide text` command toggles the visibility of the *Command* window pane.

A check mark next to *View – Windows – Command* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Command* window pane by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Command* window pane to expand it, or clicking the X to hide it.

 To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default* (`reset dockwindows` command).

For more information on the *Command* window pane, see the *Getting Started with Physical Design* user guide in your documentation set.

Syntax

`showhide text [show] [hide]`

showhide_text	Displays or hides the pane, depending on its current state.
show	Displays the pane if it is hidden. If it is already visible, no action occurs.
hide	Hides the pane if it is visible. If it is already hidden, no action occurs.

Access Using

- Menu Path: *View – Windows – Command*

Controlling the Visibility of the Command Window Pane

1. Choose *View – Windows*.
 - a. To hide the pane, click *Command* if a check mark appears next to it.
 - b. To display the pane, click *Command* if no check mark appears next to it.

showhide view

The `showhide view` command toggles the visibility of the *Worldview* window pane.

A check mark next to *View – Windows – Worldview* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Worldview* window pane by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Worldview* window pane to expand it, or clicking the X to hide it.

 To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default UI* (`reset dockwindows` command).

For more information on the *Worldview* window pane, see the *Getting Started with Physical Design* user guide in your documentation set.

Syntax

```
showhide view [show] [hide]
```

showhide_view	Displays or hides the pane, depending on its current state.
show	Displays the pane if it is hidden. If it is already visible, no action occurs.
hide	Hides the pane if it is visible. If it is already hidden, no action occurs.

Access Using

- Menu Path: *View – Windows – Worldview*

Controlling the Visibility of the Worldwide Window Pane

1. Choose *View – Windows*.
 - a. To hide the pane, click *Worldview* if a check mark appears next to it.
 - b. To display the pane, click *Worldview* if no check mark appears next to it.

showhide views1

The showhide views1 command opens a second work area. You can perform most of the actions inside the Split View window. You can also zoom and pan in the Split View window independent of the main design window.

A check mark next to *View – Split View* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Split View* window pane by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Split View* window pane to expand it, or clicking the X to hide it.

Toggles the visibility of the *Split View* window pane.

Access Using

- Menu Path: *View – Split View*

Controlling the Visibility of the Split View Window Pane

1. Choose *View – Split View*.
 - a. To hide the pane, click *Split View* if a check mark appears next to it.
 - b. To display the pane, click *Split View* if no check mark appears next to it.

showhide vis

The `showhide vis` command toggles the visibility of the *Visibility* window pane.

A check mark next to *View – Windows – Visibility* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Visibility* window pane by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Visibility* window pane to expand it, or clicking the X to hide it.

 To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default UI* (`reset dockwindows` command).

For more information on the *Visibility* window pane, see the *Getting Started with Physical Design* user guide in your documentation set.

Syntax

```
showhide vis [show] [hide]
```

showhide_vis	Displays or hides the pane, depending on its current state.
show	Displays the pane if it is hidden. If it is already visible, no action occurs.
hide	Hides the pane if it is visible. If it is already hidden, no action occurs.

Access Using

- Menu Path: *View – Windows – Visibility*

Controlling the Visibility of the Visibility Window Pane

1. Choose *View – Windows*.
 - a. To hide the pane, click *Visibility* if a check mark appears next to it.
 - b. To display the pane, click *Visibility* if no check mark appears next to it.

showhide_workflow

The `showhide_workflow` command restores the *Design Workflow* window pane to display in the positions in which you last viewed it.

Syntax

`showhide_workflow`

Access Using

- Menu Path: *View – Windows – Design Workflow*

show element

The `show element` command lets you list the attributes of a graphic element. It displays all values relevant to the element, such as its graphic coordinates, segment coordinates (for lines, connect lines, rectangles, and shapes), segment length, center and radius (for arcs), symbol type and reference designator (for package symbols), attached properties.

The `show element` command shows the schedule for user schedule nets. For pins and vias, the command also displays backdrill data.

You can enable hyperlinks to display external data in the Show Element window. Add the `COMMENT` property on an object with value as the address to the external link. For example, select a net and add `COMMENT = https://www.cadence.com` using `property_edit` command. Open the Show Element dialog for the net and click the link, it opens the external link in your default browser. You must ensure that environment variables `allegro_html` and `allegro_html_qt` are set for this to be applicable.

Related Topics

- [Displaying Design Attributes for an Object](#)
- [Finding an Object by its Property](#)
- [Finding an Object by its Name](#)

Show Element Dialog Box

Access Using

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

The Show Element dialog box is a text box. It contains the following controls:

<i>File – Save As</i>	Saves the information in a text file. When you issue this command, the layout editor prompts you for a file name and appends the .txt extension.
<i>File – Print</i>	Prints the contents of the window on either UNIX or Windows systems. Use the User Preferences Editor dialog box to set the <code>print_unix_command</code> environment variable governing UNIX printing or the <code>print_nt_extension</code> environment variable governing Windows printing.
<i>File – Stick</i>	Makes the window remain on screen until you close the window, or the program terminates. Use this option to compare information between two windows. For example, you may use <code>show element</code> to obtain information about two design elements and use <i>File – Stick</i> to compare the contents of each window.

You can click on the x y coordinates in the Show Element dialog box and zoom center on the location in the Design window.

To be able to search a text file when you use the *File – File Viewer*, *File – Viewlog*, or *Display – Element* menu commands, be sure to set the `allegro_html` environment variable by choosing *Setup – User Preferences*.

To be able to access a web link as the value of a property, be sure to set the `allegro_html` environment variable by choosing *Setup – User Preferences*. For additional information on storing web links as the value of a property, see the *Creating Design Rules* user guide in your product documentation.

Find By Name/Property

Use this dialog box to set up search criteria so you can find element types quickly.

<i>Object Type</i>	Defines the element type you want to select.
<i>Available Objects</i>	Lists all the available elements in the design.
<i>Name Filter</i>	Lets you narrow the element list of names by typing in names, parts of names, and using wildcards.
<i>Value Filter</i>	Lets you narrow the element list of values by typing in values, parts of values, and using wildcards.
<i>All -></i>	Lets you move all the <i>Available Objects</i> into the <i>Selected Object</i> list.
<i><-All</i>	Lets you move all the <i>Selected Objects</i> into the <i>Available Object</i> list.
<i>Selected Objects</i>	Lists all the elements you chose.

Double clicking an element in either the *Available Object* list or the *Selected Object* list results in the element moving to the other column.

When you click *Apply*, the Show Element dialog box appears and the Find by Name/Property dialog box remains open. When you click *OK*, the elements are found but the Find by Name/Property dialog box closes.

Related Topics

- [Finding an Object by its Property](#)
- [Finding an Object by its Name](#)

Displaying Design Attributes for an Object

This procedure lets you display element attributes. You can also find instances of inherited properties on parent and child elements using this method. This depends on where you start to search for inherited properties. If you add the FIXED property to a net and, by inheritance, to its associated pin, only the first instance of the inherited property (attached to the pin) is printed. Since the attachment does not exist on the pin, it is reported as being inherited from the net.

Follow these steps to display design attributes for an object:

1. Run the `show element` command.
2. In the Find filter, choose the design elements you want to display.
3. Position the cursor over an element and click to select.
The element is highlighted and the Show Element dialog box appears. It contains all values relevant to the element you picked.
4. Choose additional elements for display or click right and choose *Done* from the pop-up menu.

 You can print a listing of the highlighted design element or you can save the listing to a file.

Related Topics

- [show element](#)
- [Finding an Object by its Name](#)

Finding an Object by its Property

To find an object by its property, follow these steps:

1. Run the `show element` command.
2. Click *More* in the Find Filter.
The Find by Name/Property dialog box appears.
3. Choose the property from the *Available Properties* list box.
The property appears in the *Name* field.
4. To display all elements that have the chosen property, click *Apply*.
A Show Element dialog box appears, listing all elements to which the chosen property currently is attached.
Any elements on the design that have the chosen property are highlighted. If there are no such elements, a message is displayed in the command console:
No instances of *<property_name>* found.
5. To display attributes for the chosen element, click `Show`.
The Find by Property Show dialog box appears.

Related Topics

- [show element](#)
- [Show Element Dialog Box](#)

Finding an Object by its Name

To find an object by its name, follow these steps:

1. Click the arrow next to the drop-down list box at the bottom of the Find Filter.
2. Choose the type of element from the list.
3. Enter the name of the element in the *Name* field to the right of the drop-down list box.
4. Click *Enter*.

The Show Element dialog box appears displaying the name of the selected elements including other details such as, location, connected shapes if any, or logical path (displays the id or name of the object).

Related Topics

- [show element](#)
- [Show Element Dialog Box](#)
- [Displaying Design Attributes for an Object](#)

show measure

The `show measure` command lets you calculate the distance between two user-defined points on your design and displays the following information:

- Distance
- Total distance
- Manhattan distance
- Change along the x-axis
- Change along the y-axis
- Pick Angle

Related Topics

- [Measuring Distance between Two Points in the Design](#)

Measure Dialog Box

Access Using

- Menu Path: *Display – Measure*



- Toolbar Icon:

<i>Dist</i>	Displays the distance between two markers shown on the elements you picked.
<i>Total Dist</i>	Displays the accumulated total of all values displayed in the <i>Dist</i> field since you chose the second element or since you last chosen Next from the pop-up.
<i>Manhattan Dist.</i>	Displays the absolute sum of the x-distance and the y-distance between two markers. This is always a positive value
<i>Dx</i>	Displays the absolute x-distance (horizontal) between two markers.
<i>Dy</i>	Displays the absolute y-distance (vertical) between two markers.
	Note: Manhattan Dist = Dx + Dy
<i>Pick Angle</i>	Displays the angle between two markers. This field is useful when doing offset routing.
<i>Width</i>	Displays the width of line segments along a connect line.

If you have a connection path joining two elements, the following options appear on the Measure dialog box:

<i>Etch/Conductor Dist</i>	Displays the distance along the center lines of the connect lines connecting the two elements.
<i>Total Etch/Conductor</i>	Displays the accumulated connection path length from the first selection you made.
<i>Via Count</i>	Displays the number of vias on the path joining the last two points you picked.
<i>Air Gap</i>	Displays the minimum distance between the two elements you picked. If either element is a DRC marker, NC Drill figure, or a point not on any element, then a message displays indicating that no Air Gap was measured. A similar message displays if both picks are on the same etch/conductor type element. ⚠ Air gap is not measured when pad is not defined for the selected element.
<i>On Subclass</i>	Displays the subclass that is common to both elements, if they have one. This field does not display if there is no common subclass.

Measuring Distance between Two Points in the Design

You can measure the distance between two points in your design by following these steps:

1. Run `show measure`.
2. Adjust the Find Filter to choose specific design elements.

3. Position the cursor and click to highlight the first element.

The Measure dialog box displays and identifies the element and its location.

4. Position the cursor and click to highlight the second element.

The Measure dialog box is updated with the second element and its location, and displays the distance between the two points you chose.

The following temporary markers on each element appear:

- A cross indicates the center of a pad or the vertex of a connect line or filled rectangle.
- A square at the nearest grid point identifies all other picks.

If you pick two different elements and an air gap has been defined between them, a line showing the air gap between the nearest points on the two elements is displayed.

The command finds the connecting path, if it exists, between the two elements you pick, highlights it, and displays the distance in the Dist field of the Measure dialog box. If more than one connecting path joins the two elements, one of them is found and highlighted.

- a. To measure any other path, indicate it by picking intermediate points along it and read the Total Dist field of the Measure dialog box.
5. When you are finished, click right to display the pop-up menu, and choose Done.

Related Topics

- [show measure](#)

show parasitic

The `show parasitic` command lets you calculate capacitance between any two conductor (including connect lines, filled rectangles, or shapes) elements in your design. The program displays the result on the Parasitic Calculator dialog box.

⚠ Capacitance to shapes is based on a rectangular approximation to the shape area. 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 capacitance).

ⓘ The program cannot calculate capacitance if you have not defined a shield layer.

Related Topics

- [Measuring Capacitance Between Two Conductor Elements in the Design](#)

Parasitics Calculator Dialog Box

Access Using

- Menu Path: *Display – Parasitic*

The Parasitics Calculator is a text display box that contains the following controls:

<i>File</i> – <i>Save As</i>	Saves the information in a text file. When you issue this command, the layout editor prompts you for a file name and appends the .txt extension.
<i>File</i> – <i>Print</i>	Prints the contents of the window on either UNIX or Windows systems. Use the User Preferences Editor dialog box to set the print_unix_command environment variable governing UNIX printing or the print_nt_extension environment variable governing Windows printing.
<i>File</i> – <i>Stick</i>	Makes the window remain on screen until you close the window, or the program terminates. Use this option to compare information between two windows. For example, you may use show element to obtain information about two design elements and use File – Stick to compare the contents of each window.

Measuring Capacitance Between Two Conductor Elements in the Design

Perform the following steps to measure capacitance between two conductor elements:

1. Run `show parasitic`.
To focus on certain etch/conductor elements, adjust the Find Filter.
2. Position the cursor and click to highlight the first etch/conductor element.
The Parasitics Calculator dialog box is displayed. The dialog box displays
 - Impedance
 - Inductance
 - Capacitance to shield layer
 - Propagation delay
 - Resistance values
3. Position the cursor and click to highlight the second etch/conductor element.
The dialog box displays new values and calculates the capacitance to the previous etch/conductor element.
4. Continue to choose etch/conductor elements, or click right to display the pop-up menu and choose *Done*.

Related Topics

- [show parasitic](#)

show property

The `show property` command identifies the properties in your current design in the Show Property dialog box. You can list all design elements assigned to a property/value or view a property definition.

For more details about properties, see the *Creating Design Rules* user guide in your product documentation.

Related Topics

- [Finding Elements With A Specific Property/value](#)
- [Displaying Properties Graphically](#)

Show Property Dialog Box

Access Using

- Menu Path: *Display – Property*

Use this dialog box to find elements with a specific property/value or view the definition of a property.

Information Tab

<i>Available Properties</i>	Displays a list of all the layout editor properties. Click a property to choose it. The property name appears in the <i>Name</i> field.
<i>Filter</i>	Limits the properties you want displayed in the <i>Available Properties</i> list.
<i>Name</i>	Searches for the property name entered in this field.
<i>Value</i>	Searches for the property value entered in this field. A property must be defined in the <i>Name</i> field before this field is active.
<i>Type</i>	Indicates the property type after you have chosen a property.
<i>Sort By</i>	Sorts elements in one of the following ways:
<i>Element</i>	(Default) Lists properties by design element.
<i>Property</i>	Lists design elements by property.
<i>Show Val</i>	Displays a list of all the elements that have the chosen property/value. The list appears sorted in a separate window that remains open until you close it.
<i>Show Def</i>	Displays the definition of the chosen property, which appears in a separate window that remains open until you close it.
<i>Reset</i>	Clears the fields and resets to the defaults.

Graphics Tab

<i>Available Properties</i>	Displays a list of all the layout editor properties. Click on a property to choose it.
<i>Filter</i>	Limits the properties on display in the Available Properties list.
<i>Selected Properties</i>	Displays the name of the property for which to create text.
<i>Subclass</i>	Identifies the manufacturing subclass on which to create text for the chosen properties.
<i>Text Block</i>	Specifies the size of the text block.
<i>Text name</i>	Specifies the name of the text block.
<i>Property Name</i>	If chosen, property text includes both property name and value.
<i>Reset</i>	Clears the fields and resets to the defaults.
<i>Create</i>	Click to create text for properties listed in <i>Available Properties</i> .
<i>Delete</i>	Deletes all text on the subclass.

Related Topics

- [Displaying Properties Graphically](#)

Finding Elements With A Specific Property/value

To find elements with a specific property or value, perform these steps:

1. Choose *Display – Property* (`show property` command).
The Show Property dialog box appears.
2. Click the Information tab.
3. Choose a property from the *Available Properties* list. –or– Enter a property name in the *Name* field.
You can enter the property name in uppercase or lowercase.

⚠ You can click `Filter` to limit the listed properties. By default, all properties appear.
4. If needed, enter a property value in the *Value* field.
5. If needed, change the *Sort by* method.
6. Click *Show Val* for a list of elements that have the property—and its value, if specified. –or– Click *Show Def* for a definition of the property.
The Show window appears.
7. Click *OK* to close the Show Property dialog box.
To allow you to view property information while using other commands, the Show window does not disappear when you close the main Show Property dialog box. Close the Show window when you are done.

Related Topics

- [show property](#)

Displaying Properties Graphically

To display properties graphically, perform the following steps:

1. Choose *Display – Property* ([show property](#) command).
2. Click the Graphics tab.
3. Choose a property from the *Available Properties* list, moving it to the *Selected Properties* section, which displays the name of the property for which to create text.
4. Choose a manufacturing subclass on which to create text for the chosen properties in the *Subclass* field. If you specify a user-defined subclass to which to add properties, you must define them up prior to instantiating any properties using *Setup – Subclasses* ([define subclass](#) command).
5. Choose a value in the *Text Block* field, to specify the size of the text.
6. Specify a name for the text block in the *Text name* field.
7. Enable the *Property Name* field to allow property text to include both the property name and value.
8. Click *Create* to create text. The status bar in the dialog box shows the number of text instances added.
9. Click *OK* to close the dialog box.
10. Choose *Display – Color Visibility* or click the color icon in the tool bar to display the Color dialog box.
11. In the *Package Geometry* section, click the ASSEMBLY TOP and BOTTOM subclasses to display them.
12. Set the *Global Visibility* to *All Invisible*.
13. Click Yes in the confirmator that appears.
14. Set *Group* to *Manufacturing* and click any user-defined subclasses to display them; otherwise, the layout editor adds the text instances to the PROPERTIES subclass by default.
15. Click *Apply* on the Color dialog box.
16. Click the Show Element icon. Set the *Find Filter* to *All Off* and enable *Text*.
17. Window select to zoom in. The elements with the property name and value text appear.

Related Topics

- [show property](#)
- [Show Property Dialog Box](#)

show rlc

The `show rlc` command is used with the `add connect` command. It displays the total values of parasitic resistance, inductance, and capacitance on the chosen net.

show symbol drc

The `show symbol drc` command makes all pin-to-pin minimum spacing DRC markers for large symbols visible. This applies to all the symbol definitions and instances in the design.

When a symbol with a high pin count (pin count over 10,000) is imported to the design, the `NODRC_SYM_SAME_PIN` property is applied to it by default. This property hides the pin-to-pin minimum spacing DRC markers for the pins on the symbol. This command removes the property from the symbol to show the symbol DRCs.

Access Using

- Menu Path: *Display – Large Symbol DRCs – Show*

Related Topics

- [NODRC_SYM_SAME_PIN](#)

show waived drcs

The `show waived drcs` command lets you display all waived DRC error markers on the board. This command is the opposite of the `blank waived drcs` command.

For more information on waiving DRC errors, see [waive drc](#), [blank waived drcs](#), [restore waived drc](#), and the [Creating Design Rules](#) user guide in your documentation set.

Access Using

- Menu Path: *Display – Waive DRCs – Show*

Showing Waived DRC Error Markers in the Design

 This command displays waived DRC errors that already exist in the design but are invisible, but will not waive DRC errors.

1. Run the `show waived drcs` command.
The waived DRC error markers appear on the board.

signal 3dmodel

The `signal 3dmodel` command displays the 3-D Interconnect Modeling dialog box and enables you to generate 3D package and interconnect device model files suitable for PCB-level simulation.

- ⓘ** Software, licensing, and support for *Sentinel-NPE* is not provided by Cadence Design Systems, but rather the 3rd party. Cadence's software will check to see whether the proper *Sentinel-NPE* engine is in place and, if so, invoke it. You must ensure that you have *Sentinel-NPE* field solver and models installed on your operating system.

Package Model Formats

The 3D Field Solver outputs package model files in the following formats.

- Spice circuit models for single or coupled nets
 - Multi-section narrow band models
 - Wide band models
- IBIS package format
- DML format
 - RLGC
 - Subcircuit wrapped
- S-Parameter in Touchstone format

For further details on the DML formats along with DML package model examples, refer to Appendix B of your product documentation.

Model Parasitics Report

When you generate a 3D package or interconnect device model, a Parasitics report is automatically generated. You can access this report by opening the file `<model_name>.csv` located in your current working directory. The file is written in a tab or blank space-separated format and can be easily loaded into an Microsoft Excel® spreadsheet.

The head record line is in the following format with units specified within parentheses.

```
Neti Netj Rij(mOhm) Lij(nH) Cij(pF) Gij(uMho) Td(ns)
```

The data record line is in the following format.

```
<net_name_1>, <net_name_2>, <R_value>,<L_value>,<C_value>,<G_value>
```

- ⚠** If `<net_name_1>` and `<net_name_2>` are identical, the RLGC are self-coupling parasitic values. Otherwise, they are mutual-coupling parasitic values.

Sample Report

```
Parasitic Extraction Results ----- Solder Name
```

```
Neti,NetJ,Rij (mOhm),Lij (nH),Cij (pF),Gij (uMho),
H,H,8.053e-02,1.924e-09,1.507e-09,1.236e-05,
H,G,0.000e+00,1.031e-09,1.783e-15,6.521e-08,
H,A,0.000e+00,3.599e-10,1.258e-15,2.239e-08,
G,G,7.939e-02,1.889e-09,2.236e-09,1.862e-05,
G,A,0.000e+00,6.087e-11,1.858e-17,6.268e-11,
A,A,7.011e-02,1.669e-09,2.231e-09,1.271e-05,
```

```
G,F,0.000e+00,2.909e-10,1.367e-15,3.376e-08,  
H,F,0.000e+00,3.949e-10,1.226e-17,9.083e-12,  
F,F,7.022e-02,1.650e-09,1.024e-09,1.261e-05,  
F,E,0.000e+00,9.406e-10,2.578e-15,8.377e-08,  
G,E,0.000e+00,3.885e-10,1.767e-17,3.736e-10,  
E,E,7.015e-02,1.666e-09,5.857e-10,1.162e-05,  
E,D,0.000e+00,6.116e-10,1.504e-15,3.011e-08,  
F,D,0.000e+00,4.453e-10,2.633e-17,2.804e-10,  
D,D,8.477e-02,2.003e-09,2.104e-09,1.686e-05,  
D,C,0.000e+00,6.794e-10,3.487e-15,1.403e-07,  
E,C,0.000e+00,3.949e-10,1.244e-17,2.368e-11,  
C,C,8.431e-02,1.963e-09,7.954e-10,9.889e-06,  
C,B,0.000e+00,6.780e-10,1.652e-15,3.528e-08,  
D,B,0.000e+00,4.418e-10,2.738e-17,1.531e-10,  
B,B,6.994e-02,1.664e-09,1.239e-09,1.248e-05,  
B,A,0.000e+00,1.060e-09,4.219e-15,1.412e-07,  
C,A,0.000e+00,4.583e-10,4.282e-17,1.567e-10,  
H,B,0.000e+00,4.332e-10,5.193e-17,7.592e-11,
```

Related Topics

- [Creating a 3D Package Device Model For PCB-level Simulation](#)
- [Creating a 3D Package Interconnect Device Model For PCB-level Simulation](#)
- [Grouping Pins in a Multiport Net](#)
- [Enabling Mapping Between Package Bump Pads And Die Bump Pads](#)

Signal 3D Model Dialog Boxes

Access Using

- Menu Path: *Analyze – 3-D Modeling*
- [3-D Interconnect Modeling Dialog Box](#)
- [3-D Modeling Parameters Dialog Box](#)
- [3-D Modeling Port Group Dialog Box](#)
- [Wire Bond Profile Editor](#)

3-D Interconnect Modeling Dialog Box

Package Model Tab

Option	Description	
<i>Select method to create Package Model</i>	Specifies the method to generate the package model.	
<i>Package Model for the whole design</i>	Creates a Package Model for the entire design	
<i>Package Model by nets</i>	Creates a Package Model only for the nets you specify in the net controls described below.	
	<i>Net</i>	Specifies a net name pattern. Click the down-arrow to select patterns previously entered.
	<i>List of Nets</i>	Displays a file browser that is used to select a netlist file.
	<i>Net Browser</i>	Displays the Signal Select Browser that selects nets in the current design.
<i>Model name</i>	Specifies a name for the package model.	
<i>Package Model Type</i>	Specifies the type of package model.	
<i>Load into existing device library</i>	When enabled, a newly generated DML package model is loaded into the SigNoise device library automatically.	
<i>Create Model</i>	Initiates package model generation.	
<i>Parameters</i>	Displays the 3-D Modeling Parameters dialog box and enables you to set the modeling parameters used by the 3D Field Solver.	
<i>Port Group</i>	Displays the Port Group dialog box from which you can assign pins to port groups	

Net Model Tab

Option	Description
Select method to create Net Model	<p>Specifies the method to generate the net model. Note:</p> <ul style="list-style-type: none"> ◦ If you choose <i>Single or coupled net model for chosen net(s)</i>, the <i>Number of coupling nets</i> value (as specified in the 3-D Modeling Parameters dialog box) is not used. One model is generated for all the chosen nets, with coupling among them modeled as well. ◦ If you choose <i>Coupled net model for chosen nets and neighbor nets</i>, for each chosen net, the specified number of neighbor nets are searched for. One model is generated for all chosen nets and their found neighbor nets. <p>⚠ This mode is much slower because finding the neighbor nets requires the 3D Field Solver to mesh all the nets in the package.</p> <p>Net selection options are:</p>
Net	Specifies a net name pattern. Click the down-arrow to select patterns previously entered.
List of Nets	Displays a file browser that can be used to select a netlist file.
Net Browser	Displays the Signal Select Browser to select nets in the current design.
Model name	Specifies a name for the net model.
Net Model Type	Specifies the type of net model. The default is DML Laplace.
Create Package Terminal Map File	Generates a text file that maps the nodes in the 3D field solver subcircuit file to the bump pad names on the die. This allows IC power analysis tools to link the power/ground model in the package to the power grid circuit of the silicon in order to perform post-route simulation with package effects.
Load into existing device library	When enabled, loads newly generated DML net models into the SigNoise device library.
Create Model	Generates the net model and displays the file in a text window.
Parameters	Displays the 3-D Modeling Parameters dialog box and enables you to set the modeling parameters used by the 3D Field Solver.

Neighbor Net Calculations

If you choose to include neighbor nets in your net model, the 3D Field Solver uses a formula to determine the 30 nearest neighbor nets from each net and saves them in the probable order of magnitude of mutual inductances to be calculated (not already calculated). This method is only an approximation of neighbor inclusion with true mutual inductances calculated at a later time (during FEM generation and after CAD parsing).

Therefore, when you perform a whole package analysis with the option to let the 3D Field Solver determine neighbor nets, some nets that are a relatively similar in distance from a reference net may be picked erroneously. To overcome this, you need to go back to the 3-D Interconnect Modeling dialog box and use the *Single or coupled net model for chosen net(s)* option, and then manually specify the neighbors/coupling nets.

3-D Modeling 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, float pins are ignored during simulation.

Option	Description
<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>	
<i>Group/Type Filter</i>	Determines which pins of a specified type are displayed in the list box. Choices are * (all), reference, float, 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

Wire Bond Profile Editor

The Wire Bond Profile Editor appears when you click the *WireBond Profile* button in the Ball tab of the 3-D Modeling Parameters dialog box.

Profiles in Use / Available	
<i>Active Profile</i>	Specifies a list of all profile names currently defined in the design or the specified technology file (<i>Master Definitions</i>).
<i>Add</i>	Lets you add a new wire profile name to the list of profiles and to the design. This becomes the <i>Active Profile</i> definition for editing.
<i>Copy</i>	Lets you add a new wire profile name to the list. The profile is a copy of the existing active profile with the new name that you provide.
<i>Delete</i>	Lets you remove the active wire profile from the design. ⚠ You cannot delete the wire profile if it is currently assigned to one or more bond wires in the design.
<i>Master Definitions</i>	Specifies the location of the <code>.xml</code> technology file, that lists the definitions of the wire profiles that you are working on.
<i>Load</i>	Lets you load another <code>.xml</code> technology file. The wire profiles currently used in the design are still available. If any profile names used in the design are defined in the new <i>Master Definitions</i> file, you are prompted about whether or not to load the new definitions.
<i>Save</i>	Lets you save the current wire profile settings to a specified technology file. Typically, only the library group of your company will use this part of the Wire Profile Editor.
<i>Definition</i>	
<i>Direction</i>	Specifies whether this profile defines a <i>Forward Bond</i> (wire runs from die pad to bond finger) or a <i>Reverse Bond</i> (wire runs from bond finger to die pad). The default setting is <i>Forward Bond</i> .
<i>Material</i>	Specifies the wire material.
<i>Diameter</i>	Specifies the wire diameter. Use a positive integer.
<i>Refresh from Master</i>	Click this button to return to the original specification in the technology file if you have customized a wire profile in the current design. If the profile is not defined in the current technology file, this button is disabled.
<i>Movement Type</i>	This section describes the movement type with the horizontal and vertical planes used for each step.
<i>Horizontal</i>	Provides a menu with these options to specify the movement type in the horizontal plane (X/Y axes along the length of the wire): <ul style="list-style-type: none"> • Length – Specifies a constant distance movement along the horizontal wire from the previous point in the model. • Percent – Specifies a constant distance movement along the horizontal wire from the previous point in the model, but specifies the distance as a percentage of the wire's length. • Angle – Specifies a change in angle of the given amount, with positive moving away from the substrate and negative moving towards it. • Switch – Indicates to the tool that it should take the next point from the other end of the wire. For example, if you are moving from the beginning of the wire, the next point starts at the other end of the wire.
<i>Value</i>	Lets you specify the value associated with the <i>Horizontal</i> movement type.

<i>Vertical</i>	Provides a menu with these options to specify the vertical movement type: <ul style="list-style-type: none"> Length – Specifies a constant distance movement vertically from the previous point in the model. Percent – Specifies a constant distance movement vertically from the previous point in the model, but specifies the distance as a percentage of the wire's length. Angle – Specifies a change in angle of the given amount, with positive moving away from the substrate and negative moving towards it. Switch – Indicates to the tool that it should take the next point from the other end of the wire. For example, if you are moving from the beginning of the wire, the next point starts at the other end of the wire.
<i>Value</i>	Lets you specify the value associated with the vertical movement type.
	<div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;">  If you set the <code>wire_profile_ui_points_in_rows</code> user preference environment variable, the step appears in a row instead of a column. </div>
<i>Example</i>	
<i>Sample Start Height</i>	<i>Specifies the starting height or the distance from the top of the substrate (used for the graphical example). By default, this setting is 200 UM to simulate an average die height.</i>
<i>Sample Wire Length</i>	<i>Specifies the wire length to use for drawing the graphical example. This default setting is the maximum wire length constraint value in the current database.</i>
<i>Current example</i>	This graphics panel illustrates the approximate look of the wire profile as currently specified. Because each bond wire that uses this profile may have a unique length, this visualization is an approximation only, based on the samples given.
<i>Apply</i>	Saves the changes made to profiles. If you exit by clicking <i>Cancel</i> after you already clicked <i>Apply</i> , you do not cancel any changes that you made.
<i>OK</i>	Accepts the changes to the wire profiles and records any modifications in the database.
<i>Cancel</i>	Cancels all changes made. This does not include any changes saved to disk through the XML-saving interface within the dialog box. Only database changes are cancelled.
<i>Help</i>	Displays the help topic for this command.

Related Topics

- [Creating a 3D Package Interconnect Device Model For PCB-level Simulation](#)
- [Grouping Pins in a Multiport Net](#)
- [Enabling Mapping Between Package Bump Pads And Die Bump Pads](#)

Creating a 3D Package Device Model For PCB-level Simulation

-  Before running any of the procedures described below, delete any existing plating bars from your design.

To create a 3D package device model for PCB-level simulation, follow these steps:

1. Choose *Analyze – 3-D Modeling*.
The 3-D Interconnect Modeling dialog box appears with the Package Model tab displayed.
2. In the *Select method to create Package Model* area, choose a model creation method. If choosing *Package Model by nets*, do one the following. Otherwise proceed to step 3.
 - a. Enter a name pattern in the *Net* field or choose a previously entered pattern from the drop-down list.
- or -
 - b. Click *List of Nets* to display a file browser to select a netlist file.
- or -
 - c. Click *Net Browser* to display the Signal Select Browser to select nets in the current design.
- or -
 - d. Select nets in the design canvas by mouse pick or window capture.

The chosen nets appear in the *Selected Nets* window.

3. In the *Model name* field, enter the name for your package model.
 4. In the *Package Model Type* area, choose the model type.
 5. If you want to have your package model loaded automatically into the SigNoise device library, click the *Load into the existing device library* button.
-  This option is not available for IBIS package models.
6. Click the *Parameters* button to display the 3-D Modeling Parameters and modify the conditions under which the Package Model will be created.
 7. Click *Create Model*.
The package model generates.
- or -
The model generation fails and an error message appears.

Related Topics

- [signal 3dmodel](#)
- [Grouping Pins in a Multiport Net](#)
- [Enabling Mapping Between Package Bump Pads And Die Bump Pads](#)

Creating a 3D Package Interconnect Device Model For PCB-level Simulation

To create a 3D package interconnect device model for PCB-level simulation, perform the following steps:

1. Choose *Analyze – 3-D Modeling*.
The 3-D Interconnect Modeling dialog box appears.
2. Click the *Net Model* tab.
3.
 - a. Enter a name pattern in the *Net* field or choose a previously entered pattern from the drop-down list.
- or -
 - b. Click *List of Nets* to display a file browser to select a netlist file.
- or -
 - c. Click *Net Browser* to display the Signal Select Browser to select nets in the current design.
- or -
 - d. Select nets in the design canvas by mouse pick or window capture. In the *Select method to create Net Model* area, choose a model creation method, then do one of the following sub steps.

The chosen nets appear in the *Selected Nets* window.

4. In the *Model name* field, enter the name for your package interconnect model.
5. If you want to have your package interconnect model loaded automatically into the SigNoise device library, click the *Load into the existing device library* button.
6. Click the *Parameters* button to display the 3-D Modeling Parameters and modify the conditions under which the Package Model will be created.
7. Click *Create Model*.

The package interconnect model generates with its related SPICE file (containing an RLC lumped subvariety).

Related Topics

- [signal 3dmodel](#)
- [Signal 3D Model Dialog Boxes](#)
- [Enabling Mapping Between Package Bump Pads And Die Bump Pads](#)

Grouping Pins in a Multiport Net

To group pins in a multiport net:

1. Choose *Analyze – 3-D Modeling*.
The 3-D Interconnect Modeling dialog box appears.
2. Click *Port Group*
The Port Group dialog box appears with a listing of the nets in the current design.
3. In the Selection Area, choose a net. (You can alternatively select a net directly from the design.)
A listing of the pins in the selected net appear in the left-side Port Group Assignment list box.
4. Specify a group type in the filter drop-down to narrow the pin list. (Optional)
5. Highlight the pins you want to place into a new group. –or– Click *All* to move the entire list to the right-side Port Group Assignment list box.
6. Select a new group type from the drop-down menu.

 One of your group types must be designated as a reference group.

All the listed pins are converted to the new group type.

7. To remove the list of pins in either list box, click *Clear*. (You can redisplay the pins by reselecting the appropriate net.)

Related Topics

- [signal 3dmodel](#)
- [Signal 3D Model Dialog Boxes](#)
- [Creating a 3D Package Device Model For PCB-level Simulation](#)

Enabling Mapping Between Package Bump Pads And Die Bump Pads

 Your design must contain die(s) upon which 3D extraction was previously performed.

You can enable mapping between package bump pads and die bump pads by following these steps:

1. Choose *Analyze – 3-D Modeling*.
The 3-D Interconnect Modeling dialog box appears.
2. Check *Create Package Terminal Map File*.
3. Click *Create Model*.
A .ptmf file is created in your current working directory.

Example package terminal map file

```
I8 100.0, 300.0
I7 pin A
I6 -100.0, 300.0
I5 pin B
O2 pin C
I4 300.0, 100.0
I2 pin D
I1 100.0, -300.0
```

Related Topics

- [signal 3dmodel](#)
- [Signal 3D Model Dialog Boxes](#)
- [Creating a 3D Package Device Model For PCB-level Simulation](#)
- [Creating a 3D Package Interconnect Device Model For PCB-level Simulation](#)

signal atimes

The `signal atimes` command displays the Crosstalk Active Times Import dialog box for loading timing data into the design database.

 This command is not available in L Series products.

Related Topics

- [Importing Loading Timing Data to the Design Database](#)

Crosstalk Active Times Import Dialog Box

Access Using

- Menu Path: *File – Import – Active Times*

Use this dialog box to load timing data into the design database.

When you import timing data, a mapping file is created that associates Verilog hierarchical names and simulation ranges to net names in the design database. The simulation ranges are loaded in the design database as values for the XTALK_ACTIVE_TIME property.

<i>Active Times Data File</i>	Enter the name of the active times data file generated by Verilog-XL or NC-Verilog.
<i>Net Name Map File</i>	Click to choose Net Name Map File and enter the name of the mapping file to be created.

Importing Loading Timing Data to the Design Database

Perform the following steps to import loading timing data to the design database:

1. Run `signal atimes`.
The Crosstalk Active Times Import dialog box appears.
2. Enter the name of the active times data file in the *Active Times Data File* field.
3. Click to choose *Net Name Map File* and enter the name of the mapping file.
4. Click *OK*.

The simulation ranges are loaded in the design database as values for the `XTALK_ACTIVE_TIME` property of the nets listed in the net names mapping file.

Related Topics

- [signal atimes](#)

signal audit

The `signal audit` command runs the *SI Design Audit* wizard, which helps you run an audit on all or selected nets in a design. The wizard helps you audit specific nets in the layout to verify that they are set up properly for extraction and simulation.

Related Topics

- [Signal Audit Tasks](#)

Signal Audit Dialog Boxes

Access Using

- Menu Path: *Setup – SI Design Audit*
- Menu Path: *Tools – Utilities – Keyboard Commands*. Choose *signal audit* in Command Browser.
- Toolbar Icon: 

The SI Design Audit wizard enables you to perform the following tasks:

- Controlling the Tests to Run
- Selecting Xnets and Nets to Audit
- Viewing Audit Errors
- Highlighting Errors
- Resolving Errors
- Viewing Error Report
- Importing Ignored Errors

Signal Audit Tasks

1. Run `signal audit`.

The SI Design Audit wizard is displayed.

In this wizard you can perform an audit on selected nets and Xnets and check for any missing models. A report is displayed for that net indicating the current status. The SI Design Audit wizard walks you through the steps to:

- [Controlling the Tests to Run](#)
- [Selecting Xnets and Nets to Audit](#)
- [Viewing Audit Errors](#)
- [Highlighting Errors](#)
- [Resolving Errors](#)
- [Viewing Error Report](#)
- [Importing Ignored Errors](#)

Related Topics

- [signal audit](#)

Controlling the Tests to Run

You can control the tests you wish to run in the first page of the SI Design Audit wizard. All the tests you can run are organized into categories in a tree structure. You can run tests on various categories, such as Layer Stack, Power & Ground Nets and Components. Each category lists the tests you can run for the category. For example, you can run the test to check for illegal VOLTAGE property value on power and ground nets and the test for layer thickness on cross-sections.

1. Click a category to select or deselect it.
2. Choose the tests to run or ignore from the list of tests for the category.
By default, all the tests are selected. The list of tests to be run is saved with the drawing. These tests are displayed when you run the SI Design Audit command again.
3. Click *Next* to move to the next page of the wizard.

Related Topics

- [signal audit](#)
- [Signal Audit Dialog Boxes](#)
- [Signal Audit Tasks](#)

Selecting Xnets and Nets to Audit

After deciding on the tests to run, you select the Xnets and nets to be audited in the second page of the wizard.

-  If you disable all the Xnet/net-based tests, such as those in the Components, Xnets, InterConnect, SI Models, and Diff Pairs categories in page 1, this page does not appear.

You can select Xnets and nets individually, by bus, by differential pair, or for an entire design.

1. Click the individual Xnet/net or the entire bus or differential pair to include them in the audit.

By default, Xnets and nets which are members of a bus or a differential pair are shown as members of the bus or differential pair, respectively.

You can control the display of bus and differential pairs in this list.

2. Choose the *Show Buses* or *Show Diff Pairs* options to display or hide buses and differential pairs.

You can also import a text file containing Xnets and nets that are to be selected. The file must contain each Xnet and net name on a separate line, and have `.lst` extension.

3. Choose the *Import Xnets/Nets to be Selected* button to import the Xnet and net names from an external file.

4. Browse to the file and click *Open*.

All the Xnets and nets specified in the file are selected.

Additionally, you can export the currently selected Xnet and net names to an external file (`.lst`) by clicking the Export Selected Xnets/Nets button.

5. Select the *Include Coupled Xnets* option in case you selected *Simulation* or *Estimated Crosstalk Simulation* in the list of audits to be performed.

6. To view a list of coupled Xnets, click the *List Coupled Xnets* button.

This command searches for Xnets that are coupled to each of the selected Xnets. A progress bar appears to show the progress of this search.

In case the list of Xnets is too large, it might take a while before the list is processed. You can choose to cancel the search of coupled Xnets. The Allegro *Stop* button is activated and you can click this any time before the search completes to stop the command from running any further.

When you stop the command, the following message appears and no report is generated or displayed.

If you do not stop the search, the coupled Xnets report is generated.

7. Click *Next* to run the selected audit tests on the specified Xnets and nets.

Related Topics

- [signal audit](#)
- [Signal Audit Dialog Boxes](#)
- [Signal Audit Tasks](#)

Viewing Audit Errors

The audit tests are run and the results are shown on the Audit Errors page of the wizard.

This page displays a list of errors encountered, ignored, or resolved during the audit process.

GUI Element	Description
Status Filter	<p>Optionally restricts the list of errors to show only Unresolved, Resolved, or Ignored errors:</p> <ul style="list-style-type: none"> • All: Displays all the errors. This is the default selection. • Unresolved: Displays all the errors that still exist. • Resolved: Displays all the errors that are resolved. • Ignored: Displays all the errors that are to be ignored.
Test Filter	<p>Displays category-specific test results. You can display results for the following categories:</p> <ul style="list-style-type: none"> • All Tests • Power & Ground Nets • Cross-Section • Components • Xnets • Interconnect • SI Preferences • SI Models • Diff Pairs
Sort By	<p>Sorts the list of errors by either the test category or the status.</p>
Status	<p>Shows the current status of an error. You can right-click the status and choose to either resolve or ignore the error by choosing the appropriate command from the drop-down list:</p> <ul style="list-style-type: none"> • Resolve Error: Automatically resolves the selected error. If the error is resolved, the status changes to Resolved (Green). Else, the status remains Unresolved (Red). • Ignore Error: Ignores the selected error. The status changes to Ignored (Yellow). The ignored errors are saved with the drawing. On subsequent runs of the <i>SI Design Audit</i> command, such errors are displayed with the Ignored status.
Error Messages	<p>Provides a description for each error.</p>
Resolve Errors	<p>Provides options to resolve errors:</p> <ul style="list-style-type: none"> • All: Automatically resolves all the errors reported. • Selected: Resolves errors automatically. • Manually: Lets you manually resolve the errors. Depending on the type of error, a dialog opens where you can fix the error. When the separate dialog is closed, the test is run again. If the error no longer exists, the status of the error changes to Resolved.
Ignore Errors	<p>Directs the tool to ignore an error.</p> <ul style="list-style-type: none"> • All: Ignores all unresolved errors. The status of all the errors is changed to Ignored. • Selected: Ignores the selected unresolved error(s).
Show Resolution	<p>Displays a message that describes how the selected error was resolved.</p>

Report	Creates a report that shows each error with its status and test category in a text editor.
--------	--

Related Topics

- [signal audit](#)
- [Signal Audit Dialog Boxes](#)
- [Signal Audit Tasks](#)

Highlighting Errors

The tool can highlight a net that has been flagged with a missing voltage property error.

1. Right-click an error and choose the *More Info About Error* popup command.
The net is zoomed-in and highlighted in the drawing.

Related Topics

- [signal audit](#)
- [Signal Audit Dialog Boxes](#)
- [Signal Audit Tasks](#)

Resolving Errors

To resolve an error:

1. In the list of errors, select the error to be resolved or ignored.
2. Click the *Selected* button under *Ignore Errors* to ignore an error. Or click *All* to ignore all the errors.
3. To resolve the selected error, click *All*, *Selected*, or *Manually* as required.
On clicking the *Selected* button, the Select Errors to be Resolved dialog appears.
If you select *All*, a suggested resolution for all the errors is listed in the dialog box.
4. You can select or deselect the errors you want to resolve. When you are done, click *OK*.

The status of all the resolved errors changes. Some of the errors that remain unresolved need to be resolved manually.

You can also resolve errors manually. For example, to resolve a pin use mismatch error, do the following:

1. Select the message in the list of errors.
2. Click *Manually*.
The Change Pin Use of a Pin dialog appears.
3. Select the pin from the Pins of net GND list.
4. Click the *Change all pins to Power* or *Change all pins to Ground* as appropriate to resolve the error.
5. Click *OK*.

If you choose multiple nets to resolve, they are resolved one at a time. A warning message to this effect appears when you select several messages from the list in one go and choose to resolve them manually.

Related Topics

- [signal audit](#)
- [Signal Audit Dialog Boxes](#)
- [Signal Audit Tasks](#)

Viewing Error Report

You can create and view a report of errors. To create a report:

1. Click the *Report* button.

A report is generated on the fly that shows each error with its status and test category. The report is displayed in a text editor. You can save the report as a text file and also print it.

Related Topics

- [signal audit](#)
- [Signal Audit Dialog Boxes](#)
- [Signal Audit Tasks](#)

Importing Ignored Errors

You can transfer the list of ignored errors from one drawing to another.

1. After you select errors and choose the command to ignore them during audit, can click *Import Report* and load the report file (.txt file) created earlier.
Each ignored error in the report file in the current errors list is searched. If found and the error is currently Unresolved, its state is changed to Ignored.

Related Topics

- [signal audit](#)
- [Signal Audit Dialog Boxes](#)
- [Signal Audit Tasks](#)

signal bus setup

The `signal bus setup` command lets you identify the source synchronous buses in your layout, and provide data required for you to perform the analysis. You enter this data by way of the Signal Bus Setup and the Stimulus Setup dialog boxes.

The functionality embodied in the setup command is required before you can perform a simulation of a source synchronous bus, the command for which is [signal bus sim](#). If you have already set up buses in your design for simulation, setup is not required.

You can find additional information on source synchronous bus analysis in the [Allegro PCB SI User Guide](#).

Related Topics

- [Setting Up A Simulation Bus](#)
- [Setting Up A Signal Bus](#)

Signal Bus Setup Dialog Boxes

Access Using

- Menu Path: *Analyze – Bus Setup*

The signal bus setup command supports the following three dialog boxes:

- Signal Bus Setup Dialog Box
- Create Simulation Buses Dialog Box
- Stimulus Setup Dialog Box

Signal Bus Setup Dialog Box

This dialog box consists of a common bus selection area, three tab pages, Export/Import controls, and common functional buttons. All are described in the sections below.

Select Bus to Setup Area

Option	Description
<i>Bus Name</i>	Identifies the bus you are setting up. The drop-down menu contains a list of all the buses previously defined in the active design, as well as any buses you have created for simulation purposes with the <i>Create Simulation Bus</i> option.
<i>Bus Direction</i>	Allows you to define whether the selected bus is unidirectional or bidirectional. The default is bidirectional.
<i>Controller Refdes</i>	Identifies the component in the selected bus that serves as the controller. The drop-down menu contains a list of the reference designators of all the components of class IC connected by the selected bus. If the selected bus contains components in multiple designs; that is, is a system bus, the full system refdes name is displayed.
<i>Switch On</i>	Defines the edge of the clock on which the bus data is latched: either <i>Rising Edge</i> , <i>Falling Edge</i> , or <i>Both Edges</i> . The default is <i>Rising Edge</i> .
<i>Derating Table File</i>	Defines the name of the file that contains the derating table. This file must have a <i>.dat</i> extension and must exist in one of the directories defined by the SIGDAT Allegro directory search path. The browse button to the right of the field opens a browser that displays all the files with a <i>.dat</i> extension in all of the directories defined by the SIGDAT search path.
<i>Create Simulation Bus</i>	Opens the Create Simulation Buses dialog box to create new buses available <i>only</i> for bus simulations. See Create Simulation Buses Dialog Box for additional information.
<i>Assign Bus Stimulus</i>	Opens the Stimulus Setup dialog box which displays all the nets in the selected bus along with their associated clock or strobe nets.

Assign Bus Component Buffer Model Tab

Use the controls in this tab to define the IO buffer models that you will use for the pins in the selected bus. You must define three models for each pin: one model defines the pin as a driver, another model defines the pin as a receiver, and a third when the pin is in standby mode.

⚠ It is assumed that the IBIS Device model assigned to each of the components in the selected bus contains model selectors that define the legal IO buffers for each pin of the model that is being assigned. If this is not the case, an error message is generated which states:

The signal models assigned to the components in this bus don't have model selectors defined for the pins of the bus. This means that the default model assigned to each of the bus pins will be used for the driver, receiver and standby states.

Option	Description
<i>Assign By</i>	Allows you to assign models in one of two ways: <i>Model Selector: Component</i> : The selection you choose dictates the configuration of the columns in the tab.
<i>Column Filters</i>	Each column supports a filtering mechanism that allows you to select from a drop-down menu any or all the listed values. You can also enter a wild card value to filter for specific values.
<i>Component Model</i> (displayed when <i>Model Selector</i> assignment method is chosen)	Lists the IBIS Device models assigned to the components in the selected bus. You cannot edit these values. In this mode (<i>Model Selector</i> assignment), you can select a driver, active receiver, and standby receiver model for each of the model selectors referenced by the nets in the bus you are setting up. Right-click in the column header to sort the cell contents in ascending order.

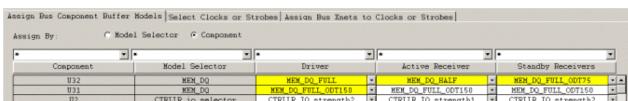
S Commands

S Commands--signal bus setup

<i>Component</i> (displayed when <i>Component</i> assignment method is chosen)	Lists the component reference designators in the selected bus. In this mode (<i>Component</i> assignment), you can change the model assignment for specific components. This makes it possible for different components that reference the same model selector to have different models selected for bus simulation. Note that when such a state exists, the Driver and/or Receiver fields referencing the different models are displayed as ".....".
<i>Model Selector</i>	The model selectors defined in each of the IBIS Device models that are referenced by the data and clock pins of the selected bus. You cannot edit these values. Right-click in the column header to sort the table based on the content of this column.
<i>Driver</i>	Defines for each of the pins in the bus that have been assigned the device model and selector, the name of the IO buffer model to be used when the pin is driving. Drop-down menus for each of the <i>Driver</i> cells list the IO buffer models defined for the given model selector. The default selection for each cell is the model defined as the default for the given selector model. Right-click in the column header to sort the table based on the content of this column.
<i>Receiver</i>	Defines for each of the pins in the bus that have been assigned the device model and selector, the name of the IO buffer model to be used when the pin is receiving. Drop-down menus for each of the <i>Receiver</i> cells list the IO buffer models defined for the given model selector. The default selection for each cell is the model defined as the default for the given selector model. Right-click in the column header to sort the table based on the content of this column.
<i>Standby</i>	Defines for each of the pins in the bus that have been assigned the device model and selector, the name of the IO buffer model to be used when the pin is in standby mode. Drop-down menus for each of the <i>Standby</i> cells list the IO buffer models defined for the given model selector. The default selection for each cell is the model defined as the default for the given selector model. Right-click in the column header to sort the table based on the content of this column.
<i>Buffer Model To Be Assigned</i>	Defines for each of their associated columns a selected IO buffer for all the <i>Driver</i> , <i>Receiver</i> , and <i>Standby</i> entries in the respective cells. If there are several model selectors that require the same IO buffer model assignment, you can filter the <i>Component Model</i> and <i>Model Selector</i> columns to display only the desired rows. You can then select from the <i>Buffer Model To Be Assigned</i> field the model you wish to assign as the driver.
<i>Assign</i>	Sets all the cells in the associated column to the selected IO model.
<i>Export</i>	Writes out a .csv file in spreadsheet format containing all the displayed buffer model data in the columns of this tab. You can view and edit this file.
<i>Import</i>	Reads into the dialog box a specified file of valid data that populates the columns in this tab. The file must be a .csv file in spreadsheet format. File content is verified for validity before replacing the current buffer model assignments in the dialog box. It must be in the same format as the .csv file that is created by the <i>Export</i> command.

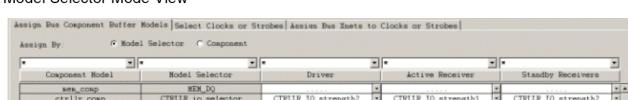
Assignment Mode Views When Referencing Different Models

Component Mode View



Driver and receivers referencing model mem_comp and model selector MEM_DQ for component refdes U32 are different than driver and receivers referencing mem_comp and MEM_DQ for component refdes U31. This state results in the different driver and receivers displayed as "....." when viewed in Model Selector mode (below).

Model Selector Mode View



Select Clock or Strobes Tab

Use the controls in this tab to identify the clock or strobe nets associated with the selected bus; clock nets if unidirectional or strobe nets for bidirectional. The purpose of this tab is to select the nets that are clocks or strobes for the selected bus.

Option	Description
<i>List Filters</i>	Allows you to define a filter for the <i>Potential</i> and <i>Selected</i> lists.

S Commands
S Commands--signal bus setup

Potential Clock/Strobe Nets	Displays all the nets assigned to the controller component except for the nets that are members of the selected bus.
Selected Clock/Strobe Nets	Displays all the nets that you have selected as clock or strobe nets from the <i>Potential</i> list.

Assign Bus Xnets to Clocks or Strobes Tab

Use the controls in this tab to assign groups of nets in the selected bus to a clock or strobe.

Option	Description
Clock/Strobe Name	Identifies the clock or strobe net to associate with the assigned bus nets. The drop-down menu contains a list of all the clock or strobe nets you selected in the <i>Select</i> tab.
List Filters	Allows you to define a filter for the <i>Unassigned</i> and <i>Assigned</i> lists.
Unassigned Bus Xnets	Displays a list of all the nets in the selected bus that you have not assigned to a clock or a strobe.
Assigned Bus Xnets	Displays the nets that are currently assigned to the clock or strobe that is identified by the <i>Clock/Strobe Name</i> field.
Export	Writes out a <code>.csv</code> file in spreadsheet format containing all the displayed clock/strobe-to-bus-net data in the columns of this tab. You can view and edit this file.
Import	Reads into the dialog box a specified file of valid data that populates the columns in this tab. The file must be a <code>.csv</code> file in spreadsheet format. File content is verified for validity before replacing the current buffer model assignments in the dialog box. It must be in the same format as the <code>.csv</code> file that is created by the <i>Export</i> command.

Common Functional Buttons

OK	Saves the data you have entered in the tabs of the dialog box and completes the command.
Apply	Saves the data you have entered in the tabs of the dialog box without terminating the command.
Cancel	Discards the information you have entered into the dialog box since you last executed <i>Apply</i> (or since the dialog was opened if you have not applied changes) and terminates the command.

Create Simulation Buses Dialog Box

Use this dialog box to create new buses available *only* for bus simulations. Nets and Xnets in simulation buses can be members of multiple buses. Once you have created buses with this option, they are displayed on the pull-down list of the *Bus Name* field and in the Show Element window (along with standard Allegro buses) when you select a net that is a member of a simulated bus.

 These buses do *not* appear as elements in your drawing layout.

Option	Description
Add Bus/Delete Bus	These controls let you create or to delete the buses that you will use for simulation purposes only. The bus that is highlighted in the Buses list appears as the selected bus.
Items lists	The left pane contains a list of the elements in the design that are not part of the selected bus. Items in the right pane are. You can move individual items from one side to the other by clicking on them, or use the <i>All</i> buttons.
Filter	The standard wildcard characters (* ?) support strings that let you limit the list of design elements in the Items panes.
Apply	This saves any changes (additions, deletions, etc.) you have made to the simulation buses.
OK	This saves your changes and closes the dialog box.

Setting Up A Simulation Bus

Buses in your design must have been previously created in Concept, Constraint Manager, or by using [Logic — Identify Buses](#) in SI or PCB Editor. You can create buses for simulation purposes only by way of the Create Simulation Buses dialog box, described below.

1. Choose *Analyze — Bus Setup*.
The Signal Bus Setup dialog box appears.
2. Click *Create Simulation Bus*.
The *Create Simulation Buses* dialog box appears.
3. Create the simulation buses using the controls the dialog boxes.
4. Click *Apply* following the creation of each simulation bus to save the change.
5. When finished, click *OK* to close the dialog box.
6. Proceed to [Setting Up A Signal Bus](#).

Related Topics

- [Signal Bus Setup Dialog Boxes](#)
- [signal bus setup](#)

Setting Up A Signal Bus

Follow these steps to set up a signal bus:

1. Choose *Analyze — Bus Setup*.
The Signal Bus Setup dialog box appears.
2. Select the bus you wish to simulate from the *Bus Name* drop-down menu.
A bus that was previously set up will display its saved settings. For buses that have not been set up, the dialog box will contain some standard defaults.
3. Configure the setup parameters using the controls in the dialog boxes. You must configure the controls in each tab of the dialog box. Use the *Apply* button to save your settings as you make them. Changes that are not applied will not be saved if the dialog box is closed inadvertently.
4. Click *Assign Bus Stimulus* to assign and/or edit custom stimulus values for the nets in the selected bus. (Remember that you can save bus and pin parameters using the *Export* control in this and the *Signal Bus Setup* dialog box.)
5. When you have completed your setup of the selected bus and are ready to simulate, click *OK* to save all your changes.
The dialog box closes and SI returns to an idle state.
6. If you are now ready to simulate the bus, choose *Analyze — Bus Simulate*.
The Analysis Bus Simulation dialog box appears.

Related Topics

- [signal bus setup](#)
- [Signal Bus Setup Dialog Boxes](#)

signal bus sim

The `signal bus sim` command lets you simulate for analysis the source synchronous buses in your layout. Before you run this command, you must have set up the buses as described in [signal bus setup](#).

You can find additional information on source synchronous bus analysis in the [Allegro PCB SI User Guide](#).

Related Topics

- [Simulating the Source Synchronous Buses in your Design for Analysis](#)

Analysis Bus Simulation Dialog Box

Access Using

- Menu Path: *Analyze – Bus Simulate*

You perform configuration and simulation by way of the Analysis Bus Simulation dialog box, which contains the following controls:

Option	Description
Case Selection Area	
Current Case	Displays the selected simulation case. The drop-down menu displays a list of all available simulation cases.
Bus to Simulate Area	
Bus Name	Displays the selected bus. The drop-down menu displays a list of all previously set up buses in the design.
Bus Setup	Opens the signal bus setup dialog box.
Assign Bus Stimulus	Opens the Stimulus Setup dialog box.
Fast/Typical/Slow Mode	Defines the simulation mode for device operating conditions.
Receiver Selection Area	
All Receivers	All receivers in the simulation use the Active Receiver model. This supports cases in which ODT settings do not apply. This is the default selection.
Each Receiver	Simulates each net in the selected bus separately, one at a time. One receiver uses the Active Receiver model (either ODT or input) and the rest of the receivers use the Standby Receivers model (input or ODT). The ODT settings of the components determine which IO buffers are used when a buffer is driving, active, or in standby mode.
Simulation Type Area	Determines the type of simulation you want to run.
Reflection	Runs a Reflection simulation.
Comprehensive	Runs a Comprehensive simulation. This report returns identical data to that in the Reflection simulation, but additionally takes into account the coupling effects in the various measurements.
Simulation Output Area	

S Commands
S Commands--signal bus sim

<i>Show Report</i>	Creates and displays an output report summarizing the results of the simulation. This is the default selection.
<i>Show Waveform</i>	Creates and displays a waveform in SigWave.
<i>Save Circuit Files</i>	Saves the circuit files created by the simulation.
<i>Simulate</i>	Executes the simulation for the selected bus.
<i>Preferences</i>	Opens the Analysis Preferences dialog box.
<i>OK</i>	Saves your changes and completes the command.
<i>Cancel</i>	Terminates the command without saving any changes.

Simulating the Source Synchronous Buses in your Design for Analysis

 You must have previously set up the selected bus for simulation. If you have not already done so, see [signal bus setup](#) for information.

You can simulate the source synchronous buses in your design for analysis by following these steps:

1. Choose *Analyze – Bus Simulate* to open the Analysis Bus Simulation dialog box.
2. Configure the parameters of the dialog box controls as described in the dialog box section, above.
3. If you wish to modify any setup parameters, you can do so from this dialog box by clicking *Bus Setup* and/or *Assign Bus Stimulus*. See [signal bus setup](#) for information these dialog boxes. You can make additional modifications to your simulation parameters from the Analysis Preferences dialog box by clicking *Preferences*. See [signal prefs](#) for information.
4. When your configuration is complete, select a simulation output mode and click *Simulate*.

The results of your simulation will be displayed as a report and/or a waveform. You can also save the circuit files created by the simulation.

Related Topics

- [signal bus sim](#)

signal demiaudit

See [signal audit](#), [signal bus setup](#), [signal audit](#), [signal lib audit](#), and [signal libs audit](#).

signal pkg_model

The `signal pkg_model` command enables you to generate 3D package device model files suitable for PCB-level simulation.

Package Model Formats

The 3D Field Solver currently outputs package model files in the following formats.

- IBIS RLGC (you must translate and load these models)
- DML RLGC Package Model
- DML Subckt Package Model

Model Parasitics Report

When you generate a 3D package device model, a Parasitics report is automatically generated. You can access this report by opening the file `<model_name>.csv` located in your current working directory. The file is written in a comma-separated format and can be easily loaded into an Microsoft Excel® spreadsheet.

The head record line is in the following format with units specified within parentheses.

```
Net i,Net j,Rij (mOhm),Lij (nH),Cij (pF),Gij (uMho)
```

The data record line is in the following format.

```
<net_name_1>, <net_name_2>, <R_value>,<L_value>,<C_value>,<G_value>
```

⚠️ If `<net_name_1>` and `<net_name_2>` are identical, the RLGC are self-coupling parasitic values. Otherwise, they are mutual-coupling parasitic values.

Sample Report

Parasitic Extraction Results ----- Solder Name

```
Neti,NetJ,Rij (mOhm),Lij (nH),Cij (pF),Gij (uMho),
H,H,8.053e-02,1.924e-09,1.507e-09,1.236e-05,
H,G,0.000e+00,1.031e-09,1.783e-15,6.521e-08,
H,A,0.000e+00,3.599e-10,1.258e-15,2.239e-08,
G,G,7.939e-02,1.889e-09,2.236e-09,1.862e-05,
G,A,0.000e+00,6.087e-11,1.858e-17,6.268e-11,
A,A,7.011e-02,1.669e-09,2.231e-09,1.271e-05,
G,F,0.000e+00,2.909e-10,1.367e-15,3.376e-08,
H,F,0.000e+00,3.949e-10,1.226e-17,9.083e-12,
F,F,7.022e-02,1.650e-09,1.024e-09,1.261e-05,
F,E,0.000e+00,9.406e-10,2.578e-15,8.377e-08,
G,E,0.000e+00,3.885e-10,1.767e-17,3.736e-10,
E,E,7.015e-02,1.666e-09,5.857e-10,1.162e-05,
E,D,0.000e+00,6.116e-10,1.504e-15,3.011e-08,
F,D,0.000e+00,4.453e-10,2.633e-17,2.804e-10,
D,D,8.477e-02,2.003e-09,2.104e-09,1.686e-05,
D,C,0.000e+00,6.794e-10,3.487e-15,1.403e-07,
E,C,0.000e+00,3.949e-10,1.244e-17,2.368e-11,
```

```
C,C,8.431e-02,1.963e-09,7.954e-10,9.889e-06,  
C,B,0.000e+00,6.780e-10,1.652e-15,3.528e-08,  
D,B,0.000e+00,4.418e-10,2.738e-17,1.531e-10,  
B,B,6.994e-02,1.664e-09,1.239e-09,1.248e-05,  
B,A,0.000e+00,1.060e-09,4.219e-15,1.412e-07,  
C,A,0.000e+00,4.583e-10,4.282e-17,1.567e-10,  
H,B,0.000e+00,4.332e-10,5.193e-17,7.592e-11,
```

Related Topics

- [Creating a 3D Package Model For PCB-level Simulation](#)

Signal PKG Model Dialog Boxes

Access Using

- Menu Path: *Analyze – 3-D Package Model*

3-D Package Modeling Dialog Box

Option	Description
<i>Model name</i>	Specifies a name for the package model.
<i>Package model Type</i>	Specifies the type of package model to generate. The default is DML RLGC.
<i>Create Model</i>	Generates the package model.
Parameters	Displays the 3-D Modeling Parameters dialog box and enables you to set the modeling parameters used by the 3D Field Solver.

3-D Modeling Parameters Dialog Box

Use this dialog box to set 3D modeling parameters used by the 3D Field Solver.

⚠ The functionality contained in this dialog box is also available in [signal 3dmodel](#) and in [signal prefs](#). If you opened this dialog while in either of those commands, please use the links above to navigate back to the appropriate documentation.

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

General Tab

Option	Description	
<i>Solder Ball Location</i>	Specifies how the package is oriented relative to the PCB. ⚠ 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. Options are:	
	<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>	Specifies the number of coupling nets to model. A value of <code>1</code> indicates a single line. A value of <code>2</code> indicates a single neighbor net. Note: <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>	Specifies a minimum via diameter. The default is either <code>2mil</code> or <code>50um</code> (depending upon the drawing unit type). <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> ▲ If the via diameter is less than the default, 90% of the diameter for the smallest pad in the design is used. </div>							
<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 <code>0</code> , 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: <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="padding: 5px;"><i>Coarse</i></td><td style="padding: 5px;">Fastest, but least accurate.</td></tr> <tr> <td style="padding: 5px;"><i>Fine</i></td><td style="padding: 5px;">Default, with good accuracy.</td></tr> <tr> <td style="padding: 5px;"><i>Finest</i></td><td style="padding: 5px;">Slowest, but most accurate.</td></tr> </table>		<i>Coarse</i>	Fastest, but least accurate.	<i>Fine</i>	Default, with good accuracy.	<i>Finest</i>	Slowest, but most accurate.
<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. <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> ▲ A large box produces a more accurate model but increases model processing time. </div>							
<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> . <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> ▲ A large box produces a more accurate model but increases model processing time. </div>							
	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> . <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> ▲ A large box produces a more accurate model but increases model processing time. </div>							
	The default is <i>Medium</i> .							
<i>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.							
<i>Start frequency</i>	Enter a value to specify the start frequency. The default is <code>0Hz</code> . 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 <code>50ohm</code> .							

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 a `.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.

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

External Ground Tab

Option	Description
<i>Include PCB plane</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</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.

⚠ Having one plane, both planes, or no planes is permissible.

SI Ignore Layers Tab

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

Creating a 3D Package Model For PCB-level Simulation

- ⓘ Before running the procedure below, delete any existing plating bars from your design.

To create a 3D package model for PCB-level simulation, follow these steps:

1. Choose *Analyze – 3-D Package Model*.
The 3-D Package Modeling dialog box appears.
2. In the *Model name* field, enter the name for your package model.
3. In the *Package model type* area, choose the model type.
4. Click the *Parameters* button to display the 3-D Modeling Parameters and modify the conditions under which the Package Model will be created.
5. Click *Create Model*.
The package model is generated.
- or -
The model generation fails and an error message appears.

Related Topics

- [signal pkg_model](#)

signal prefs

The `signal prefs` command displays the Analysis Preferences dialog box.

Use this dialog box to:

- Set default IBIS IOCell models, determine whether default IOCell models are used, and determine how buffer delays are obtained.
- Define preferences for routed and unrouted interconnect modeling and crosstalk checks and determine whether to do plane modeling.
- Enable co-planer waveguide extraction using the Electromagnetic Solution Two-Dimensional Full Wave field solver (EMS2DFW).
- Set simulation defaults for pulse stimuli, simulation duration, waveform resolution, threshold measurement for delays, and debug mode. You can also define parameter Set default units of measure for reports.
- Set defaults for EMI single net simulations. You can also determine whether advanced EMI simulations are performed and set defaults for them.

 This command is not available in L Series products

For more information about the Analysis Preferences dialog box and setting advanced preferences, see *Setting Simulation Preferences in the Floorplanner* in your product documentation.

Related Topics

- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

Signal Prefs Dialog Boxes

Analysis Preferences Dialog Box	Fast/Typical/Slow Simulations Definition Dialog Box	EMS2D Preferences Dialog Box
3-D Modeling Parameters Dialog Box	Advanced Preferences Dialog Box	

Related Topics

- [Signal Prefs Tasks](#)

Analysis Preferences Dialog Box

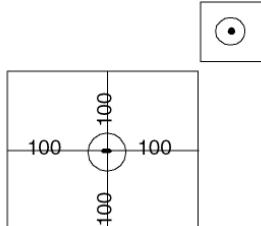
[[Device Models Tab](#)] [[InterconnectModels Tab](#)] [[Power Integrity Tab](#)]

Device Models Tab

Option	Function
<i>Use Defaults for Missing</i>	When checked, SigNoise uses specified default models for devices without specific model assignments. The default is checked. When unchecked, simulations proceed only for nets with models assigned for all components.
<i>IN</i>	Defines the default IOCell model for a pin with the PINUSE property value of IN. The default is CDSDefaultInput.
<i>OUT</i>	Defines the default IOCell model for a pin with a PINUSE property value of OUT. The default is CDSDefault Output.
<i>BI</i>	Defines the default IOCell model for a pin with a PINUSE property value of BI. The default is CDSDefaultIO.
<i>TRI</i>	Defines the default IOCell model for a pin with a PINUSE property value of TRI. The default is CDSDefaultTristate.
<i>OCL</i>	Defines the default IOCell model for a pin with a PINUSE value of OCL. The default is CDSDefaultOpenDrain.
<i>OCA</i>	Defines the default IOCell model for a pin with a PINUSE value of OCA. The default is CDSDefaultOpenSource.
<i>Browse Models</i>	Displays the SI Model Browser dialog box where you can locate and select IOCell models.
<i>Buffer Delays</i>	Specifies how SigNoise obtains buffer delays for the simulation. <i>From Library</i> specifies that the buffer delay is obtained from the model stored in the library. This is the default. <i>On-the-fly</i> specifies that the buffer delay is calculated using the IBIS model's standard load circuit. <i>No Buffer Delay</i> assumes 0ns buffer delays.

InterconnectModels Tab

Option	Function
<i>Percent Manhattan</i>	Sets the percent of manhattan distance value for unrouted transmission lines. The default is 100%.
<i>Default Impedance</i>	Sets the default impedance value. A typical value for most technologies is 20-75 Ohms. The default is 60.
<i>Default Prop Velocity</i>	Sets the default propagation velocity for unrouted transmission lines. The default is 1.4142e+08M/S.
<i>Default Diff - Impedance</i>	Sets the default differential impedance for unrouted transmission lines. The default is 100 ohms.
<i>Default Diff - Velocity</i>	Sets the default differential velocity for unrouted transmission lines. The default is 1.4142e+08 M/S.
<i>Cutoff Frequency</i>	Indicates the bandwidth within which interconnect parasitics are to be solved. The default is 0GHz.
<i>Shape Mesh Size</i>	Indicates the boundary element size when modeling routed traces, which may be considered as shapes if the traces are 40 mils or wider. The default is 50 mils.
	<p>⚠ The default mesh size is 50 mils to improve simulator performance. To get more accurate models for shapes and thick traces, adjust this parameter to a lower value.</p>

<i>Diff Pair /Via Coupling Window</i>	<p>Indicates the size of the search window used for locating Diffpair neighbor nets based on a minimum coupled length, as illustrated here. The default is 100 mils.</p> 
<i>Geometry Window</i>	<p>Displays the distance away from the primary net that SigNoise searches for neighbor nets when searching for sources of crosstalk. SigNoise takes into account the nets on either side of the primary net as well as the nets on layers above and below the primary net. The default distance is 10 mil.</p> <p>⚠ Allegro platform products recognize different geometry window settings in board file segments (that is, boards containing multiple .mcm packages) or in multiple boards in a coupled configuration. The result is a detailed crosstalk report that considers the different geometry window settings in each of the .brd/.mcm. See Setting Geometry Windows At The Drawing Level, below, for details.</p>
<i>Min Coupled Length</i>	<p>Displays the minimum length for which a primary net segment and a neighbor net segment fall within the geometry window. The neighbor net segment falling within the geometry window must run parallel, without bendovers, for at least the specified Min Coupled Length, in order for the net pair to be analyzed for crosstalk. The default is 300 mil.</p>
<i>Min Neighbor Capacitance</i>	<p>Displays the minimum mutual capacitance value, which is the minimum amount of capacitive coupling between traces for SigNoise to look for crosstalk. The capacitance value is read from the RLGC matrix inside the package model. The default is 0.1pF.</p>
<i>Algorithm Model Generation</i>	<p>This option is On by default. It enables the retrieval of algorithm-based models for use in simulation when no traditional interconnect model matching the search criteria can be found. For additional information, see Algorithm-Based Modeling in the PCB SI User Guide.</p>
<i>Enable CPW Extraction</i>	<p>This option operates in conjunction with both the Bem2d and Ems2d field solvers. In conjunction with the Bem2d field solver, the extractor attempts to detect coplanar waveguides (CPWs) during the extraction process. Any generated models containing CPWs are then simulated using the Ems2d field solver. (CPWs are more fully described in the PCB SI User Guide section, Dynamic Analysis with the EMS2D Full Wave Field Solver.) Generated models that do not contain one or more CPWs are simulated using the Bem2d field solver. In conjunction with the Ems2d field solver, the extractor employs only Ems2d to generate models.</p>
<i>Trace Solver</i>	<p>These options allow you to select a trace solver for simulation. <i>Bem2d</i>: Specifies the Boundary Element 2.5D field solver based on static and quasi-TEM conditions for single and coupled trace geometry extractions. This option does not solve for coplanar waveguides. <i>Ems2d</i>: Specifies the Electromagnetic Solution Full Wave field solver. This option <i>does</i> solve for coplanar waveguides. <i>Sentinel-NPE</i>: Specifies the 3D Field Solver for package designs.</p> <p> ⓘ Sentinel-NPE 3-D field solver is supplied and supported by a third-party vendor. You can use the Sentinel-NPE 3-D field solver in Allegro Package Designer+. Before you begin, you must ensure that you have the Sentinel-NPE field solver installed on your operating system.</p> <p>⚠ Before initiating 3D package modeling and simulation, refer to the "3D Field Solver Setup Guidelines" in the Allegro PCB SI User Guide.</p>
<i>Via Solver</i>	<p>Displays the Via Solver you select in the Via Model Setup dialog.</p>
<i>Coupled Vias</i>	<p>Displays the status of coupled vias, whether enabled or disabled. This value depends on the state of the <i>Enable Coupled Vias</i> check box in the Via Model Setup dialog.</p>
<i>Preferences</i>	<p>Launches the EMS2D Preferences Dialog Box, from where you can set various frequency settings for the EMS2d field solver.</p>

<i>Via Modeling Setup</i>	Opens the Via Model Setup dialog from where you specify how vias are modeled.
<i>Differential Extraction Mode</i>	When enabled (checked), specifies that differential nets be extracted only as a pair. When disabled, differential nets can be extracted individually.
<i>Diffpair Topology Simplification</i>	Specifies that the minimum space of all coupled traces of the extracted topology be used first, and that the unbalanced max length be a few times (default to 8) of this minimal length.
<i>Plane Modeling</i>	Specifies whether or not ground plane modeling is active. Be sure that <i>Do Plane Modeling</i> is chosen only when you are actively using ground plane analysis. Checking this causes SSN simulations to use an RLC mesh representation of the power and ground planes, which models the delivery of the supply current to components. Unchecking this models the power and ground planes as ideal voltage sources in SSN simulations.

Power Integrity Tab

Option	Function
<i>MultiNode grid size</i>	The granularity with which Power Integrity meshes planes during multi-node analysis. The number you select influences the maximum possible mesh cell size; the higher the number, the smaller the size. The values in this field denote the x-axis (length) and y-axis (width) directions. Therefore, a value of 8x8 indicates a grid mesh of 8 equally-sized cells across and 8 equally-sized cells down <i>when the Adaptive Level value is set to 1</i> . See <i>Gridding Planes for Multi-node Simulation</i> in your product documentation for a full explanation of adaptive meshing and when to use it.
<i>Adapt Level</i>	The number of divisions into which each grid cell is divided for adaptive meshing. The number you select influences the minimum possible mesh cell size; the higher the number, the smaller the size. For example, if you set <i>Multi-node grid size</i> to 8x8 and <i>Adapt Level</i> to 4, the regular base mesh becomes (8 x 8) x 4, or 32x32. The drop-down menu, in addition to numerical values, includes pre-defined settings which automatically set the multi-node grid size and adaptive level (grid size:level). They are: <i>Open</i> — Equivalent to 1x1:128 <i>Low</i> — Equivalent to 8x8:4 <i>Med</i> — Equivalent to 8x8:8 <i>High</i> — Equivalent to 16x16:8. See <i>Gridding Planes for Multi-node Simulation</i> in your product documentation for a full explanation of adaptive meshing and when to use it.
<i>Min. plane/board area</i>	The minimum area of a shape on a <i>conductor layer</i> . See DC Net/Plane Association During analysis, Power Integrity always considers shapes on <i>plane layers</i> . This field works in conjunction with the <i>include planes on conductor layers</i> field, below.
<i>Include planes on conductor layers</i>	Whether to include conductor layers in analysis. When checked, Power Integrity considers conductor layers in analysis. This field works in conjunction with the <i>Min plane/board</i> field, above.
<i>Voltage Supply</i>	The voltage passed to the simulator, if required.
<i>Temperature</i>	The temperature value passed to the simulator, if required.
<i>Frequency-Domain Impedance Range</i>	Provides additional information in the <code>powerAnalysis.run</code> directory on phase/real/imaginary/magnitude impedance in the frequency domain. See <i>Time and Frequency-Domain Simulation</i> in your product documentation for a full explanation of this feature.
<i>Time-Domain Voltage Ripple Display</i>	Enables the controls for either the Trapezoidal or Gaussian noise current pulse (you cannot select both). Selection of this option impacts other controls in the following manner: <ul style="list-style-type: none"> • Sets the <i>Voltage Supply</i> control to zero if a value had not previously been set for the voltage of the power plane upon which the voltage ripple will be added. • Sets the default upper limit of the analysis frequency range as the reciprocal of the time value input in either the fastest rise/fall field or the Gaussian width field. The default value of the lower limit is set to 1000Hz. See <i>Time and Frequency-Domain Simulation</i> in your product documentation for a full explanation of this feature.

<i>Trapezoidal Noise Current Pulse/ Fastest Tr/Tf</i>	Enables you to set the fastest rise and fall time of the noise pulse. The default value is 500ps. This value does not necessarily represent the actual switching current profile. It is used to help you validate the effectiveness of decapacitance, not intend to obtain the actual voltage variation of the power supply.
<i>Gaussian Noise Current Pulse/ Gaussian Width</i>	Enables you to set the width that represents the frequency range of the noise pulse. The default value is 1ns. This value does not necessarily represent the actual switching current profile. It is used to help you validate the effectiveness of decapacitance, not intend to obtain the actual voltage variation of the power supply.
<i>Upper (analysis frequency range) limit</i>	The upper frequency range of the simulation.
<i>Lower (analysis frequency range) limit</i>	The lower frequency range of the simulation.
<i>Multiplane Connector Resistance</i>	Series resistance parameter for the connector model used to simulate plane-to-plane connections during multi-plane pair simulation.
<i>Multiplane Connector Inductance</i>	Series inductance parameter for the connector model used to simulate plane-to-plane connections during multi-plane pair simulation.
<i>Corner Frequency</i>	Frequency up to which the target impedance is constant.
<i>Slope dB/Decade</i>	Ramp up of target impedance after the corner frequency.
<i>Calculate on Place</i>	When chosen, Power Integrity calculates the mounted inductance for that capacitor, and the PQ_MOUNTED_INDUCTANCE property is added.
<i>Default Assignment</i>	Default mounted inductance applied to all capacitors that do not have a mounted inductance property.

EMS2D Preferences Dialog Box

The settings in this dialog box determine how the Ems2d field solver will analyze for net extraction.

Frequency Settings	
<i>Default Frequencies</i>	Directs Ems2d to use standard Bem2d settings to solve the model.
<i>Frequency Parameters</i>	
<i>Start Frequency</i>	Specifies the frequency of the start point with respect to the # of Frequency Points. The remaining points are at equal intervals between the start frequency and end frequency.
<i>End Frequency</i>	Specifies the frequency of the end point with respect to the No. of Frequency Points. The remaining points are at equal intervals between the start frequency and end frequency.
<i># of Frequency Points</i>	Specifies the number of frequency points for which to generate the model.
<i>Frequency Point File</i>	Lets you select a frequency point file to provide specific frequency points in GHz. Frequency point files are ASCII-text files that you create using a <code>.frequency</code> extension. The files can reside at a location of your choice. The format of the file should resemble this example: <code>0.0001 0.0002 0.001 0.002 1 2 10 20 . . .</code>
<i>Fast Frequency Sweep (Reduced Order Model)</i>	Directs the Ems2d to scale the computation when selecting the frequency points between the start and end frequencies.
<i>Output SParameter</i>	Outputs into your signoise.run directory an S-Parameter Touchstone file for each model you generate. You can then view the wave form in SigWave.

Via Model Extraction Setup Dialog Box

The settings in this dialog box determines how to extract and model vias for simulation.

Single Via Tab	
<i>Model Generation Options</i>	Activates specific dialog box options according to the model generation type selected (<i>Closed Form</i> or <i>Analytical Solution</i>).
<i>Output Format</i>	<ul style="list-style-type: none"> • <i>Wide Band Equivalent Circuit</i> Specifies the use of wideband equivalent circuit syntax in the via model output format. Format Details: <ul style="list-style-type: none"> • <i>Start Frequency</i> for MGH applications recommended at 10MHz. • <i>End Frequency</i> should be about $2/t_{rise}$ ($1/t_{rise}$ minimum). Set to $5/t_{rise}$ for greater accuracy (similar to when you use a fine waveform resolution like 5ps or 10ps). • Leaving <i>Approx Order</i> set to 10 is recommended. You can increase it to 12 if <i>End Frequency</i> goes beyond 20GHz for improved accuracy. • There is some loss of accuracy compared to the S Parameter format. However, simulation time is significantly faster. • There is some risk of instability with this type of model. Convergence issues are possible if the frequency range is stretched too far. • <i>Narrow Band Equivalent Circuit</i> Specifies the use of narrow band equivalent circuit syntax in the via model output format. Format Details: <ul style="list-style-type: none"> • The narrowband model is derived from the <i>Target Frequency</i>. <ul style="list-style-type: none"> ◦ Use a target frequency that is near the middle of the energy content. ◦ Good rule of thumb is $1/(1000 \cdot \text{risetime})$. For a driver with 100ps rise times a target frequency of 10MHz is recommended. ◦ If <i>Target Frequency</i> is too high, then low frequency (DC losses) are dramatically overestimated. ◦ If <i>Target Frequency</i> is too low, then high frequency effects (skin effect and dielectric loss) are underestimated. However, these are small effects in a via. • This is the least accurate of the via model formats. However, it is very stable and simulates very quickly.

S Commands

S Commands--signal prefs

	<table border="1"> <tr> <td><i>Frequency Dependent Parameters</i></td><td></td></tr> <tr> <td>• <i>Target Frequency</i></td><td>This field is active only when you have selected <i>Narrow Band Equivalent Circuit</i> as your via model type. The default value is 10MHz.</td></tr> <tr> <td>• <i>Start Frequency</i></td><td>With Wideband Equivalent Circuit selected, specifies the start frequency for the equivalent circuit (RLC values). With S Parameter selected, specifies the frequency of the start point with respect to the No. of Frequency Points. The remaining points are at equal intervals between the start frequency and end frequency.</td></tr> <tr> <td>• <i># of Frequency Points Approximate Order</i></td><td>When the output format is set to <i>S Parameters</i>, specifies the number of frequency points for which to generate S parameters. When the output format is set to <i>Wideband Equivalent Circuit</i>, specifies the order of the equivalent circuit generated. The <i>Approximation Order</i> value must be within the range of 1 to 15 inclusive. The higher the order - the more accurate the solution at the cost of processing time.</td></tr> <tr> <td>• <i>Reference Impedance</i></td><td>Specifies the reference impedance used for generating the model.</td></tr> <tr> <td>• <i>End Frequency</i></td><td>With Wideband Equivalent Circuit selected, specifies the end frequency for the equivalent circuit (RLC values). With S Parameter selected, specifies the frequency of the end point with respect to the No. of Frequency Points. The remaining points are at equal intervals between the start frequency and end frequency.</td></tr> </table>	<i>Frequency Dependent Parameters</i>		• <i>Target Frequency</i>	This field is active only when you have selected <i>Narrow Band Equivalent Circuit</i> as your via model type. The default value is 10MHz.	• <i>Start Frequency</i>	With Wideband Equivalent Circuit selected, specifies the start frequency for the equivalent circuit (RLC values). With S Parameter selected, specifies the frequency of the start point with respect to the No. of Frequency Points. The remaining points are at equal intervals between the start frequency and end frequency.	• <i># of Frequency Points Approximate Order</i>	When the output format is set to <i>S Parameters</i> , specifies the number of frequency points for which to generate S parameters. When the output format is set to <i>Wideband Equivalent Circuit</i> , specifies the order of the equivalent circuit generated. The <i>Approximation Order</i> value must be within the range of 1 to 15 inclusive. The higher the order - the more accurate the solution at the cost of processing time.	• <i>Reference Impedance</i>	Specifies the reference impedance used for generating the model.	• <i>End Frequency</i>	With Wideband Equivalent Circuit selected, specifies the end frequency for the equivalent circuit (RLC values). With S Parameter selected, specifies the frequency of the end point with respect to the No. of Frequency Points. The remaining points are at equal intervals between the start frequency and end frequency.
<i>Frequency Dependent Parameters</i>													
• <i>Target Frequency</i>	This field is active only when you have selected <i>Narrow Band Equivalent Circuit</i> as your via model type. The default value is 10MHz.												
• <i>Start Frequency</i>	With Wideband Equivalent Circuit selected, specifies the start frequency for the equivalent circuit (RLC values). With S Parameter selected, specifies the frequency of the start point with respect to the No. of Frequency Points. The remaining points are at equal intervals between the start frequency and end frequency.												
• <i># of Frequency Points Approximate Order</i>	When the output format is set to <i>S Parameters</i> , specifies the number of frequency points for which to generate S parameters. When the output format is set to <i>Wideband Equivalent Circuit</i> , specifies the order of the equivalent circuit generated. The <i>Approximation Order</i> value must be within the range of 1 to 15 inclusive. The higher the order - the more accurate the solution at the cost of processing time.												
• <i>Reference Impedance</i>	Specifies the reference impedance used for generating the model.												
• <i>End Frequency</i>	With Wideband Equivalent Circuit selected, specifies the end frequency for the equivalent circuit (RLC values). With S Parameter selected, specifies the frequency of the end point with respect to the No. of Frequency Points. The remaining points are at equal intervals between the start frequency and end frequency.												
• <i>Frequency Sweep Type</i>	Displays the scale used for selecting the frequency points between the start and end frequencies.												
• <i>Step Size</i>	View-only field that displays the frequency step time based on start and end frequencies and number of frequency points. (The recommended frequency step size is 10MHz.) Specifically, the equation used is $(\text{end_frequency} - \text{start_frequency}) / (\#\text{_of__frequency__points})$ If the number of frequency points is 1, the step size should be 0.												
Coupled Via Tab													
<i>Model Generation Options</i>	Activates specific dialog box options according to the model generation type selected (<i>Analytical Solution</i> or <i>Coupled Disabled</i>). If you select the default, <i>Coupled Disabled</i> , setup is configured according to the settings in the Single Via tab controls.												

<i>Output Format</i>	<ul style="list-style-type: none"> <i>S-Parameters</i> <p>Specifies that S Parameter syntax be used in the via model output format. This is the only format that supports coupled via models. Format Details:</p> <ul style="list-style-type: none"> The most accurate via format. Accurately captures the via behavior over the entire frequency range. Expect slower simulation performance over circuit-based formats as more processing is required. Start Frequency for multi-gigahertz applications is recommended at 10MHz. If DC convergence issues occur, you can drop to 1MHz (but no lower than 0.1MHz). End Frequency should be about $2/t_{rise}$ ($1/t_{rise}$ minimum). Go up to $5/t_{rise}$ for greater accuracy, similar to when you use a fine waveform resolution like 5ps or 10ps. No. of Freq Points should be 128 points for most via models (this is the default value) <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p>⚠ If you are to include S-Parameter via models in larger S Parameter circuits, their accuracy must be similar to that of the desired final circuit. Otherwise, unpredictable results may occur.</p> </div>
<i>Wide Band Equivalent Circuit</i>	Not available for coupled vias.
<i>Narrow Band Equivalent Circuit</i>	Not available for coupled vias.
<i>Frequency Dependent Parameters</i>	
<i>Start Frequency</i>	Specifies the frequency of the start point with respect to the # of Frequency Points.

# of Frequency Points	Specifies the number of frequency points for which to generate S parameters. The value must be within the range of 1 to 15 inclusive. The higher the order - the more accurate the solution at the cost of processing time.
Reference Impedance	Specifies the reference impedance used for generating the model.
End Frequency	Specifies the frequency of the end point with respect to the No. of Frequency Points
Frequency Sweep Type	Displays the scale used for selecting the frequency points between the start and end frequencies.
Step Size	View-only field that displays the frequency step time based on start and end frequencies and number of frequency points. (The recommended frequency step size is 10MHz.) Specifically, the equation used is $(\text{end_frequency} - \text{start_frequency}) / (\#\text{_of__frequency__points})$ If the number of frequency points is 1, the step size should be 0.

Simulation Tab

Option	Function
Pulse Cycle Count	Sets the pulse number to measure from a series of pulses. This value controls the simulation duration so that the requested number of pulses propagates before the simulation stops. The default is 1.
Pulse Clock Frequency	Displays the frequency of the pulse stimuli for nets that have no specific pulse rate assigned. The default is 50 MHz.
Pulse Duty Cycle	Sets the length of the high portion of the cycle as a fraction. (0.5 represents equal high and low portions of the cycle.)
Pulse/Step Offset	Sets the launch time offset for the primary driver.
Fixed Duration	When checked the specified value determines the length of time a simulation will run. If unchecked, SigNoise determines the duration dynamically for each simulation.
Waveform Resolution (Time)	Sets the waveform resolution as the default or as one of the specified values. Controls how many data points are generated by the simulation and how far apart they are in time. Values are: Default (Pulse Cycle Period/100), 0.5ps, 1ps, 2ps, 5ps, 10ps, 20ps, 50ps, 100ps, 200ps, 500ps, 1ns, 2ns, 5ns, 10ns. Choosing a small value results in larger waveform files and longer simulation times. You can change the default divisor value with the ANL_WAVE_RESOLUTION_FACTOR environment variable.
Measure Delays At	Specifies the voltage threshold from which SigNoise measures delays. Input Thresholds specifies that SigNoise measure delays at the Input Logic Thresholds, Vil and Vih. V Measure specifies that SigNoise measure delays at the predetermined Buffer Delay Measurement Threshold, Vmeas or Vmeasure.

Driver/Receiver Pin Measurement Location	<p>Designates measurement locations for driver/receiver pin locations. The default position for both is <i>Model Defined</i>. You can select this setting, or model/pin/die. (The first time you change the default setting, driver/receiver resets to the same element. You can then make subsequent changes to different settings.) Choices are:</p> <ul style="list-style-type: none"> • <i>Model Defined</i>: The driver/receiver pin measurement location is defined by design context and the content in the related component DML model. • <i>Pin</i>: The pin measurement location is at the external pin node. • <i>Die</i>: The pin measurement location is at the internal die node, if present.
Advanced Measurements Settings	<p>Accesses the Advanced Measurement Parameters dialog box. From here, you can set measurement parameters that govern glitch controls that can assist you in finding correct cycles in your waveform.</p> <p>The <i>Glitch Tolerance</i> setting is a relative percentage of the faster of the rising and falling edges of each IO cell buffer model you need to measure. When a glitch occurs between the starting and ending points of a cycle, a glitch violation is reported if the value of the glitch exceeds the tolerance percentage entered in the Glitch Tolerance field. The glitch is <i>not</i> reported as a cycle.</p> <p>You can verify the status of glitch parameters in the Reflection Summary Report (for <i>Glitch</i>) and in the Delay report (for <i>GlitchRise</i> and <i>GlitchFall</i>):</p> <ul style="list-style-type: none"> • <i>Glitch</i> is the tolerance check of the rising and falling waveform • <i>GlitchRise</i> is the tolerance check on the rising waveform. If no glitch occurs in the rising waveform, the report denotes a PASS in the GlitchRise column. If one does occur, it reports a FAIL. • <i>GlitchFall</i> is the tolerance check on the falling waveform. If no glitch occurs in the falling waveform, the report denotes a PASS in the GlitchFall column. If one does occur, it reports a FAIL. <p>Glitch tolerance values are saved in the topology file and in the <code>sigxp.run</code> case management directory. If the tolerance values in these locations differ, the tolerance in the topology file takes precedent.</p>
Run Simulations in Debug Mode	When checked, SigNoise ensures that prerequisites are in place by running a net audit before running the simulation. The default is unchecked.
Report Source Sampling Data	When checked, SigNoise reports data for the primary driver as if it is also a receiver. The default is unchecked.
Prefer fastest aggressor on victim component	<p>When checked, SigNoise tries to find an aggressor driver that resides on the same component as the victim driver. If there are multiple aggressor drivers on the same component as the victim, the fastest or alphabetically first of these is chosen. If no aggressor drivers lie on the same component, then the fastest aggressor on the Xnet is chosen. The default is checked.</p> <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p>⚠ This feature helps to generate correct crosstalk test patterns where the neighbor nets lie on a common bus. In this case, signals for all nets on the bus are usually driven from one component at a time.</p> </div>
Fast/Typical/Slow	Opens the F/T/S Simulations Definition dialog box for setting default values fast, typical, and slow simulation modes.
Simulator	<p>Allows you to choose a simulator for models. Choices are Tlsm, Hspice, and Spectre.</p> <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p> ⓘ The Spectre interface is supported only on Sun Solaris 8 and 9, HP UX 11.0 and 11.11i, and Linux RHEL 3.0. Spectre is not bundled with the <i>Allegro PCB PDN Analysis</i> option. Both driver and receiver models must be Spectre models wrapped in DML.</p> </div>
Set Simulator Preferences	Opens the Advanced Simulator Preferences dialog box for Spectre (if supported) and Hspice.

Advanced Simulator Preferences Dialog Box

When you select the Spectre (not available on Windows, see above) or Hspice simulator options, you can open the *Advanced Simulator Preferences* dialog box for the chosen simulator. The controls in this dialog box let you impose simulator-specific preferences in addition to generic simulator preferences.

 You must have the Spectre and/or Hspice simulators specified in your path, as well as any libraries used in the simulator circuits.

Option	Function
<i>Command</i>	Displays the command syntax of the chosen simulator. A default command is displayed initially. You can edit this field to add/modify options.
<i>.TRAN options START =</i>	Specifies the transient sweep start time interval over which the simulation occurs. Left unset, the simulator assumes START to be zero (0).
<i>Use initial condition</i>	When checked, directs the simulator to use the initial conditions of circuit components and interconnects specified in the data statements of the model.
<i>Set Options</i>	Opens a text editor from where you can specify the <code>.options</code> statements that will be written to the beginning of the generated main simulator file. Each <code>.option</code> statement must be on a separate line

Option	Function
<i>Command</i>	Displays the command syntax of the chosen simulator. A default command is displayed initially. You can edit this field to add/modify options.
<i>.TRAN options START =</i>	Specifies the transient sweep start time interval over which the simulation occurs. Left unset, the simulator assumes START to be zero (0).
<i>Use initial condition</i>	When checked, directs the simulator to use the initial conditions of circuit components and interconnects specified in the data statements of the model.
<i>Set Options</i>	Opens a text editor from where you can specify the <code>.options</code> statements that will be written to the beginning of the generated main simulator file. Each <code>.option</code> statement must be on a separate line

Units Tab

Option	Function
<i>Design Voltage</i>	Displays a pull-down menu of available units for voltage: <i>mV</i> and <i>V</i> . The default is <i>V</i> . Power supply voltages use this value.
<i>Noise Voltage</i>	Displays a pull-down menu of available units for noise voltage: <i>mV</i> and <i>V</i> . The default is <i>mV</i> . Crosstalk, overshoot, and undershoot use this value.
<i>Design Resistance</i>	Displays a pull-down menu of available units for resistance: <i>mOhm</i> and <i>Ohm</i> . The default is <i>Ohm</i> . Resistor values use this value.
<i>Parasitic Resistance</i>	Displays a pull-down menu of available units for parasitic resistance: <i>mOhm</i> and <i>Ohm</i> . The default is <i>mOhm</i> . Package R uses this value.
<i>Design Capacitance</i>	Displays a pull-down menu of available units for capacitance: <i>pF</i> , <i>nF</i> , and <i>uF</i> . The default is <i>pF</i> . Capacitor values use this value.
<i>Parasitic Capacitance</i>	Displays a pull-down menu of available units for parasitic capacitance: <i>pF</i> , <i>nF</i> , and <i>uF</i> . The default is <i>pF</i> . Package C uses this value.
<i>Parasitic Inductance</i>	Displays a pull-down menu of available units for parasitic inductance: <i>nH</i> , <i>pH</i> , and <i>uH</i> . The default is <i>nH</i> . Package L uses this value.
<i>Delay Time</i>	Displays a pull-down menu of available units for delay time: <i>ns</i> , <i>uS</i> , <i>mS</i> , and <i>S</i> . The default is <i>ns</i> .
<i>Length</i>	Displays a pull-down menu of available units for length: <i>mil</i> , <i>Meter</i> , <i>mm</i> , and <i>uM</i> . The default is <i>Meter</i> .
<i>Spacing</i>	Displays a pull-down menu of available units for spacing: <i>mil</i> , <i>Meter</i> , <i>mm</i> , and <i>uM</i> . The default is <i>mil</i> .

EMI Tab

Option	Function
<i>EMI Regulation</i>	Specifies one of six available EMI regulations against which to evaluate the design: FCC Class A, FCC Class B, CISPR Class A, CISPR Class B, VCCI Class 1, and VCCI Class 2. The chosen regulation determines which regulation curve is superimposed on the emission level in the SigWave display and the reported pass/fail status of the emission level.
<i>Design Margin</i>	Sets the design margin value. The Design Margin value is subtracted from the regulation curve and affects the graphic display of the regulation curve as well as the pass/fail status of the report. The default is 10dB.
<i>Analysis Distance</i>	Sets the distance between the board and the receiving antenna in the measurements setup. The Analysis Distance value takes precedence over any measurement distance specified by the regulation. The default is 3M.
<i>Emulate OATS</i>	Displays whether or not an Open Area Test Site (OATS) test scenario is enabled and specified in the Advanced Preferences dialog box. The default is No.
<i>Compute Near Fields</i>	Displays whether or not computation of near field EMI effects is enabled and specified in the Advanced Preferences dialog box. The default is No.
<i>Compute Current Distribution</i>	Displays whether or not computation of current distribution in the frequency domain is enabled and specified in the Advanced Preferences dialog box. The default is No.

Button	Function

S Commands
S Commands--signal prefs

<i>Set Advanced Preferences</i>	Opens the Advanced Preferences dialog box (see below) for the specification of: <ul style="list-style-type: none">• General control settings for EMI computations• Control settings for OATS (Open Area Test Site) test scenario• Control settings for computation of near field EMI effects
---------------------------------	--

Advanced Preferences Dialog Box

[[General Tab](#)] [[OATS Tab](#)] [[Near Field Tab](#)]

General Tab

Option	Function
<i>Compute Current Distribution</i>	Specifies whether or not to calculate the current distribution as a function of frequency. When disabled (unchecked), the current distribution is not calculated. When enabled (checked), current distribution is calculated.
<i>Plane Shielding</i>	Specifies a plane shielding value that approximates the radiation (in dB) expected from stripline traces. The default is 15dB. 15 dB is an expected typical value, provided adequate ground vias and bypass capacitors have been used. You can modify this value to account for superior or inferior shielding.

OATS Tab

Option	Function
<i>Emulate OATS</i>	Specifies whether or not an Open Area Test Site is to be emulated. When disabled (unchecked), a standard EMI single net simulation is performed which searches for the maximum radiation. This is the most desirable choice for design purposes. When enabled (checked) specifies that OATS is enabled. You must provide values for the remaining parameters in this area to specify the test scenario to be validated using simulation.
<i>Antenna Height</i>	Displays the height of the antenna above the measurement ground plane. The default is 1 meter.
<i>Antenna Polarization</i>	Specifies the polarization for the receiving antenna as either vertical or horizontal. The default is <i>Vertical</i> .
<i>Board Height</i>	Displays the height of the center of the board above the measurement ground plane. Any user-specified value must be positive. The default is 0.8 meter.
<i>Board Orientation</i>	Specifies the orientation of the board with respect to the measurement ground plane. Horizontal - Face Up The top layer of the board faces up. Horizontal - Face Down The top layer of the board faces down. Vertical - Bottom Edge Down The bottom edge of the board is horizontal and rests on the turntable. Vertical - Left Edge Down The left edge of the board is horizontal and rests on the turntable. Vertical - Top Edge Down The top edge of the board is horizontal and rests on the turntable. Vertical - Right Edge Down The right edge of the board is horizontal and rests on the turntable. Options are:
	<i>Horizontal - Face Up</i> The top layer of the board faces up.
	<i>Horizontal - Face Down</i> The top layer of the board faces down.
	<i>Vertical - Bottom Edge Down</i> The bottom edge of the board is horizontal and rests on the turntable.
	<i>Vertical - Left Edge Down</i> The left edge of the board is horizontal and rests on the turntable.
	<i>Vertical - Top Edge Down</i> The top edge of the board is horizontal and rests on the turntable.
	<i>Vertical - Right Edge Down</i> The right edge of the board is horizontal and rests on the turntable.
<i>Board Azimuth degrees</i>	Displays the azimuth angle from the axis where the analysis distance is measured in the measurement setup. The range is 0 to 360 degrees. The default is 0 degrees.

Near Field Tab

Option	Function

S Commands
S Commands--signal prefs

<i>Compute Near Fields</i>	Specifies whether or not to calculate near field EMI effects. When disabled (unchecked), near fields are not calculated. When enabled (checked), near fields are calculated.	
<i>Near Field Measurement Plane</i>	Use this area to specify the measurement plane, a rectangular area over which to measure near field effects. Options are:	
	<i>Xmin</i>	The x coordinate of the lower left corner of the measurement plane. The default is the x coordinate of the lower left corner of the board.
	<i>Xmax</i>	The x coordinate of the upper right corner of the measurement plane.
	<i>Xinc</i>	The spacing increment between measurement points on the measurement plane along the x axis. The default is 250mils.
	<i>Ymin</i>	The y coordinate of the lower left corner of the measurement plane. The default is the y coordinate of the lower left corner of the board.
	<i>Ymax</i>	The y coordinate of the upper right corner of the measurement plane. The default is the y coordinate of the upper right corner of the board.
	<i>Yinc</i>	The spacing increment between measurement points on the measurement plane along the y axis. The default is 250mils.
	<i>Z</i>	The distance between the surface of the board and the measurement plane. A positive value places the plane above the top surface of the board. A negative value places the plane below the bottom surface of the board. The default is 500mils above the board.
	<i>Reset</i>	Resets the Xmin, Ymin and Xmax,Ymax values to their default values, the board extents as seen on the board display.
<i>Frequency Range</i>	Use this area to specify a range of frequencies over which to evaluate near field effects. The values you specify are rounded off to the nearest integer multiple of the clock frequency. The measurement frequency is incremented by the clock frequency through the frequency range. Options are:	
	<i>Fmin</i>	The minimum frequency at which to evaluate near fields. The default is the clock frequency in MHz.
	<i>Fmax</i>	The maximum frequency at which to evaluate near fields. The default is three times the clock frequency in MHz.
<i>Electric Field</i>	Use this area to select one or more components of the electric field for calculation. Units are DbuVolts/m. You can measure both electric and magnetic effects by making selections in both the Electric Field and Magnetic Field areas. Options are:	
	<i> E </i>	Calculates the total magnitude of the electric field.
	<i>Ex</i>	Calculates the x-axis component of the electric field.
	<i>Ey</i>	Calculates the y-axis component of the electric field.
	<i>Ez</i>	Calculates the z-axis component of the electric field.
<i>Magnetic Field</i>	Use this area to select one or more components of the magnetic field for calculation. Units are dbuAmps/m. You can measure both electric and magnetic effects by making selections in both the Electric Field and Magnetic Field areas. Options are:	
	<i> H </i>	Calculates the total magnitude of the magnetic field.
	<i>Hx</i>	Calculates the x-axis component of the magnetic field.
	<i>Hy</i>	Calculates the y-axis component of the magnetic field.

S Commands

S Commands--signal prefs

	<i>Hz</i>	Calculates the z-axis component of the magnetic field.
--	-----------	--

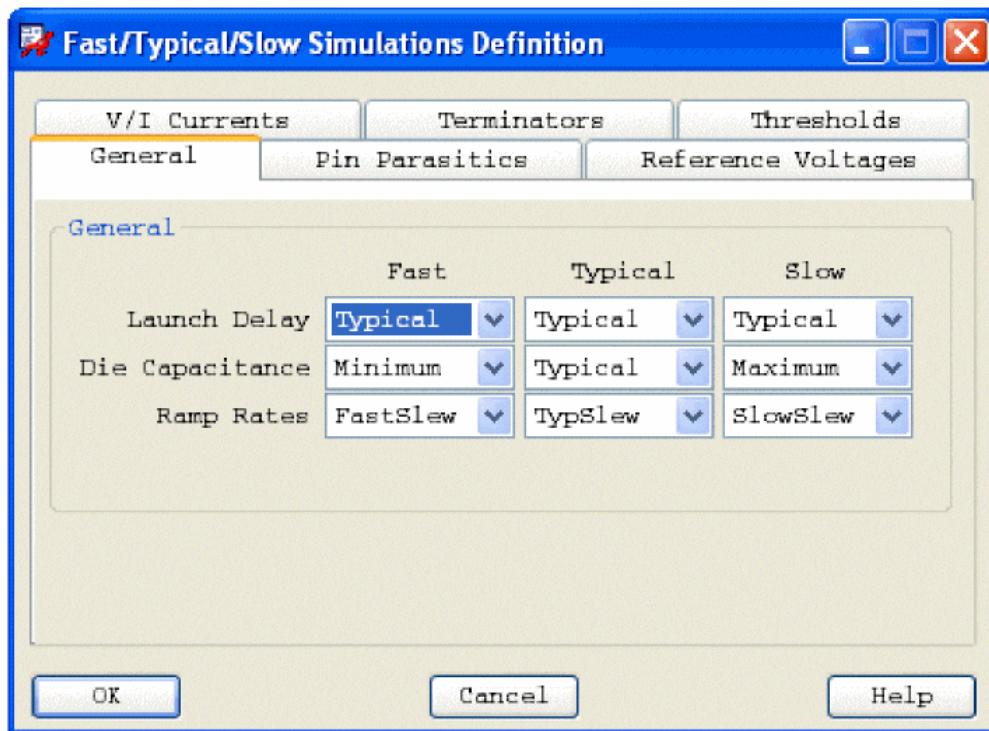
Fast/Typical/Slow Simulations Definition Dialog Box

You can represent device operating conditions by simulating in Fast, Typical, and Slow modes. The device model data is given as minimum, typical, and maximum values. This form controls the selection of model values for each simulation mode. For example, minimum Die Capacitance usually results in the fastest operating mode.

General Tab	Pin Parasitics Tab	Reference Voltages Tab
V/I Currents Tab	Terminators Tab	Thresholds Tab

General Tab

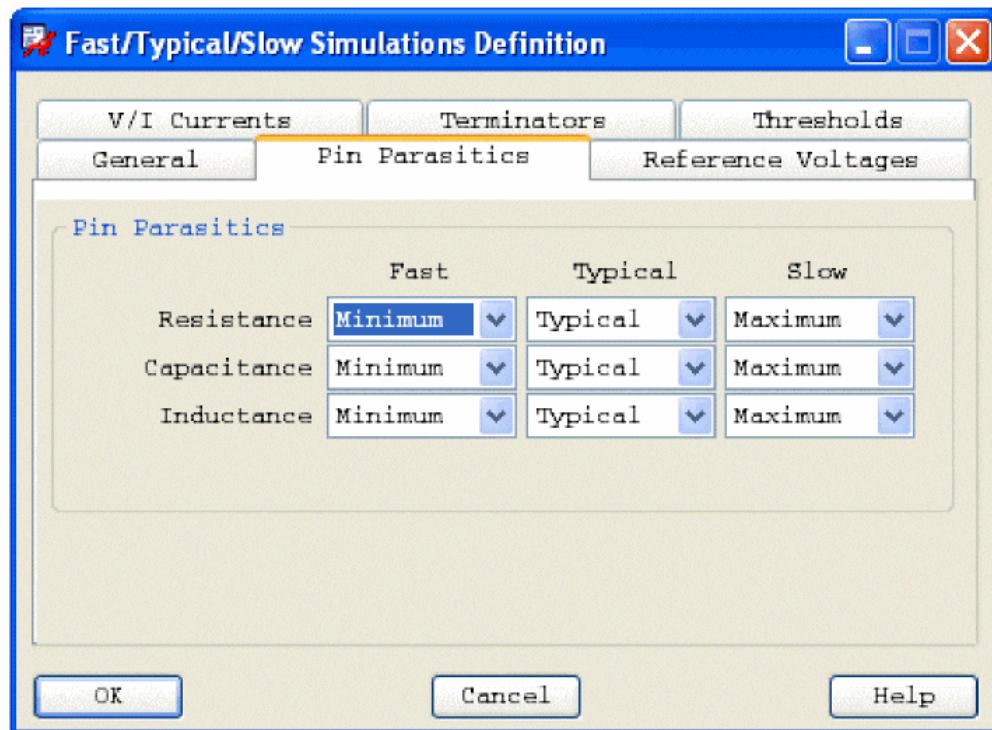
Use the General tab to define fast, typical, and slow simulation speed mode values for the Launch Delay, Die Capacitance, and Ramp Rate properties.



Option	Function
<i>Launch Delay</i>	Displays pull-down menus of launch delay values: Minimum, Typical, and Maximum.
<i>Die Capacitance</i>	Displays pull-down menus of die capacitance values: Minimum, Typical, and Maximum.
<i>Ramp Rates</i>	Displays pull-down menus of ramp rate values: FastSlew, TypSlew, and SlowSlew.

Pin Parasitics Tab

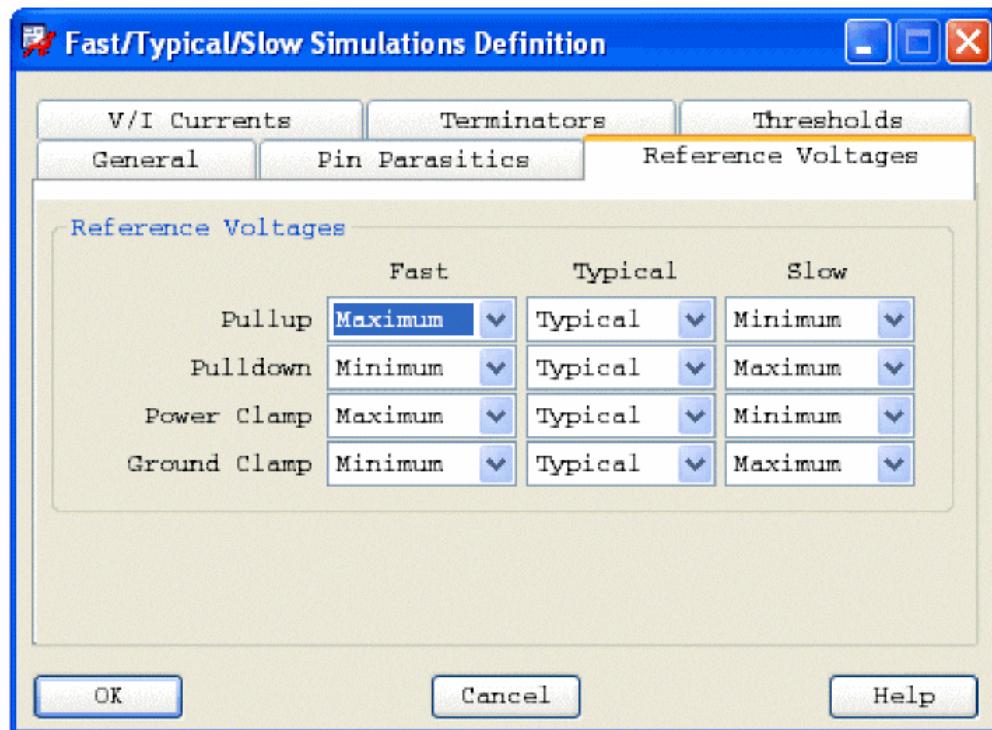
Use the Pin Parasitics tab to define fast, typical, and slow simulation speed mode values for the Resistance, Capacitance, and Inductance properties.



Option	Function
<i>Resistance</i>	Displays pull-down menus of resistance values: Minimum, Typical, and Maximum.
<i>Capacitance</i>	Displays pull-down menus of capacitance values: Minimum, Typical, and Maximum.
<i>Inductance</i>	Displays pull-down menus of inductance values: Minimum, Typical, and Maximum.

Reference Voltages Tab

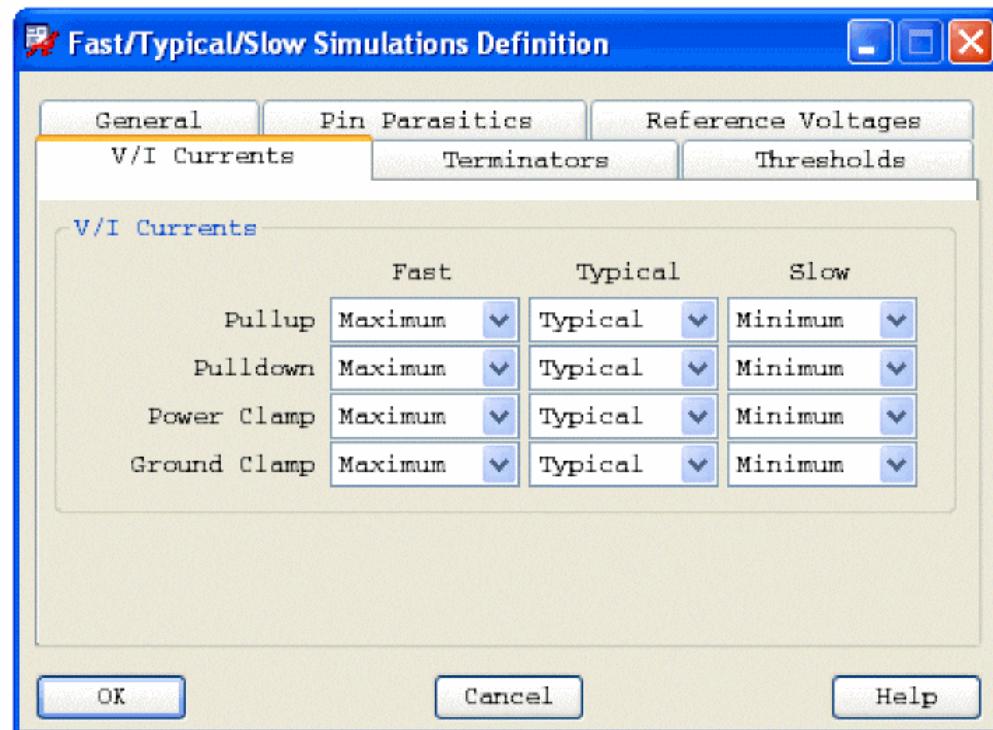
Use the Reference Voltages tab to define fast, typical, and slow simulation speed mode values for the Pullup, Pulldown, Power Clamp, and Ground Clamp properties.



Option	Function
<i>Pullup</i>	Displays pull-down menus of pullup values: Minimum, Typical, and Maximum.
<i>Pulldown</i>	Displays pull-down menus of pulldown values: Minimum, Typical, and Maximum.
<i>Power Clamp</i>	Displays pull-down menus of power clamp values: Minimum, Typical, and Maximum.
<i>Ground Clamp</i>	Displays pull-down menus of ground clamp values: Minimum, Typical, and Maximum.

V/I Currents Tab

Use the V/I Currents tab to define fast, typical, and slow simulation speed mode values for the Pullup, Pulldown, Power Clamp, and Ground Clamp properties.



Option	Function
<i>Pullup</i>	Displays pull-down menus of pullup values: Typ-Z, Low-Z, High-Z, and TempCntl.
<i>Pulldown</i>	Displays pull-down menus of pulldown values: Typ-Z, Low-Z, High-Z, and TempCntl.
<i>Power Clamp</i>	Displays pull-down menus of power clamp values: Typ-Z, Low-Z, High-Z, and TempCntl.
<i>Ground Clamp</i>	Displays pull-down menus of ground clamp values: Typ-Z, Low-Z, High-Z, and TempCntl.

If the simulation type is Temperature Controlled (TempCntl), the options in the Typical column of the form are used, except for the V/I currents. In this case, the V/I curve used is interpolated between the three given curves based on temperatures for each IOCell and the *VIReferenceTemperature* parameter.

To use *TempCntl*, you need to set the `J_TEMPERATURE` property of the component to the desired temperature. All the pins of the component inherit the property. The operating temperature of any driver or receiver on the component is the same as `J_TEMPERATURE`.

If you use *TempCntl* for the Typical mode, the V/I curve of the corresponding driver/receiver on that component is interpolated as follows:

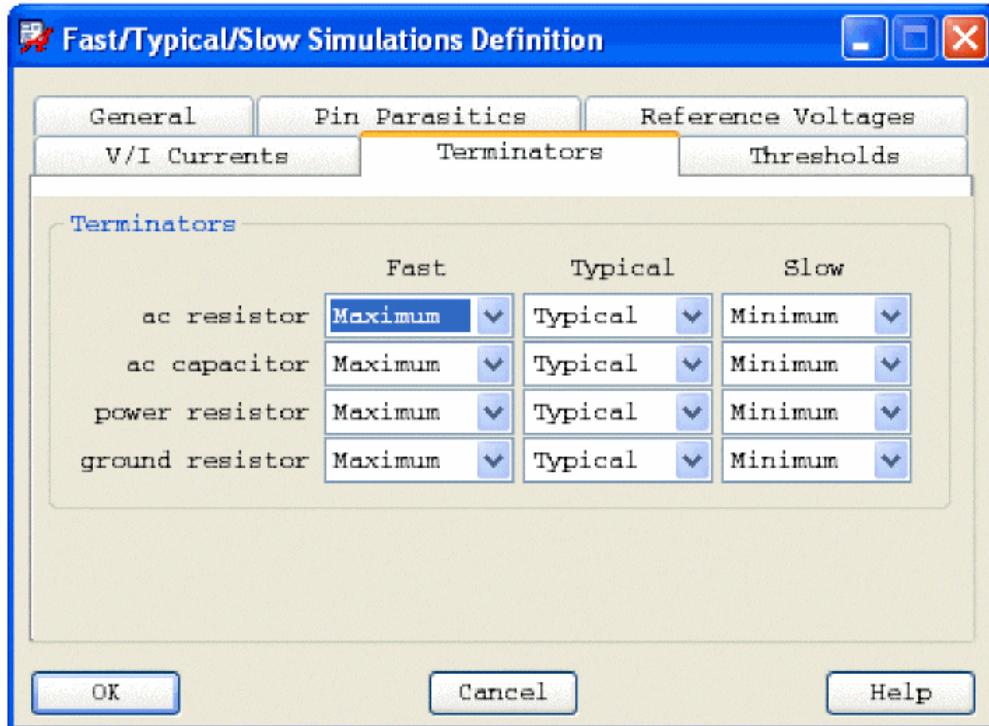
The position of `J_TEMPERATURE` is determined with respect to the V/I reference temperature section in the dml model. Given that the Reference Temperature values are: 10, 50, and 100, the value is determined as *Minimum*, *Typical*, or *Maximum*. For example, a temperature of 90 degrees falls between 50 and 100, that is between the *Typical* and *Maximum* curves.

The V/I curve is interpolated based on the *Typical* and *Maximum* curves as well as the relative position of 90 with respect to 50 and 100.

⚠ If you set the value of `J_TEMPERATURE` to below 50, the *Minimum* curve is used. If you set the value to greater than 100, the *Maximum* curve is used.

Terminators Tab

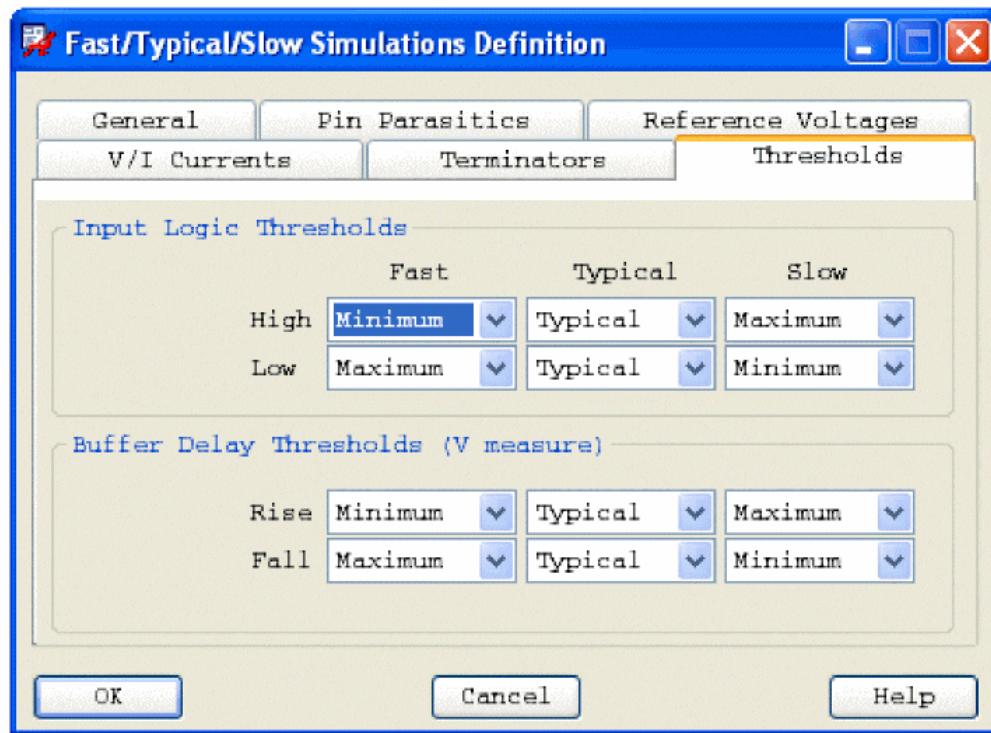
Use the Terminators tab to define fast, typical, and slow simulation speed mode for the ac resistor, ac capacitor, power resistor, and ground resistor properties.



Option	Function
ac resistor	Displays pull-down menus of ac resistor values: Minimum, Typical, and Maximum.
ac capacitor	Displays pull-down menus of ac capacitor values: Minimum, Typical, and Maximum.
power resistor	Displays pull-down menus of power resistor values: Minimum, Typical, and Maximum.
ground resistor	Displays pull-down menus of ground resistor values: Minimum, Typical, and Maximum.

Thresholds Tab

Use the Thresholds tab to define fast, typical, and slow simulation speed mode for the High Input Logic Threshold (Vih), Low Input Logic Threshold (Vil), and Buffer Delay Threshold (Vmeasure or Vmeas).



Option	Function
<i>High Input Logic Threshold (Vih)</i>	Displays pull-down menus with values: Minimum, Typical, and Maximum.
<i>Low Input Logic Threshold (Vil)</i>	Displays pull-down with values: Minimum, Typical, and Maximum.
<i>Buffer Delay Threshold (V measure)</i>	Displays pull-down menus of power resistor values: Minimum, Typical, and Maximum.

3-D Modeling Parameters Dialog Box

Use this dialog box to set 3D modeling parameters used by the 3D Field Solver.

- ⓘ Sentinel-NPE 3-D field solver is supplied and supported by a third-party vendor. You can use the Sentinel-NPE 3-D field solver in Allegro X Advanced Package Designer+. Before you begin, you must ensure that you have the Sentinel-NPE field solver installed on your operating system.

General Tab	Bond Wire Tab	Ball Tab
Bump Tab	External Ground Tab	SI Ignore Layers Tab
S-Parameters Tab		

General Tab

Option	Description	
<i>Cavity Type</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 a cavity down case.</p> <p>Options are:</p>	
	<i>Auto Detect</i>	Specifies that the cavity type be determined by the software.
	<i>Up</i>	Specifies a cavity up case.
	<i>Down</i>	Specifies a cavity down case.
<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.	
<i>CG planar boundary box</i>	<p>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>.</p> <div style="border: 1px solid #f0e68c; padding: 5px; margin-top: 10px;"> ⚠ A large box produces a more accurate model but increases model processing time. </div> <p>The default is <i>Medium</i>.</p>	
<i>CG z-directional boundary box</i>	<p>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>.</p> <div style="border: 1px solid #f0e68c; padding: 5px; margin-top: 10px;"> ⚠ A large box produces a more accurate model but increases model processing time. </div> <p>The default is <i>Medium</i>.</p>	
<i>Multiport</i>	<p>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.</p>	

Bond Wire Tab

Option	Description
<i>Diameter</i>	Specifies the diameter of all bond wires in the design. ⚠ A value of zero indicates that no bond wires are modeled.
<i>Conductivity</i>	Specifies the conductivity of all bond wires in the design. ⚠ The value specified should account for the plastic resin used to surround the bond wires.
<i>Die Component</i>	Specifies the die component for which the following <i>Die elevation</i> is defined.
<i>Die elevation</i>	Specifies the profile for the <i>Die Component</i> named previously. ⚠ Flip-chip designs do not require this parameter. Option: Info... Displays a window showing bond wire profile definitions.
<i>Wire Group</i>	Specifies the numerical ID of the loop wire group.
<i>Loop height</i>	Specifies the height between the wire peak point to the wire end point on the die (see <i>h</i> in the following diagram).
<i>Alpha</i>	Specifies the angle of the wire from the endpoint on the die. The default is 80 degrees.
<i>Beta</i>	Specifies the angle of the wire from the endpoint not on the die. The default is 80 degrees.

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.

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

Bump Tab

Option	Description
<i>Dmax</i>	Specifies the maximum diameter for the solder bumps. ⓘ Using a value that is too large risks solder bump overlap.
<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.

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

External Ground Tab

Option	Description
<i>Include PCB plane</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</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.

⚠ Having one plane, both planes, or no planes is permissible.

SI Ignore Layers Tab

Option	Description
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.

S-Parameters Tab

Use the S-Parameters tab to set S-Parameter transient simulations options

Option	Description
Transient Simulation Method	<p>The method Tlsm uses to model and simulate the S-Parameter elements in the circuit netlist. The options are:</p> <ul style="list-style-type: none"> <i>Convolution</i>: a direct-approach frequency-domain to time-domain conversion method using N2 complexity algorithm <i>Fast Convolution</i>: an approximation-approach to the direct Convolution option using NlogN complexity algorithm. This faster option is useful in cases where your simulation involves high numbers of time steps.
DC Extrapolation Method	<p>The method Tlsm uses to globally extrapolate the low-frequency points of the S-Parameters (down to 0Hz) if they are missing. The options are:</p> <ul style="list-style-type: none"> <i>Default</i>: This is the MagPhase option, described below. <i>MagPhase</i>: Extrapolates the DC values based on the magnitude and phase values. <i>Reallmag</i>: Extrapolates the DC values based on the real and imaginary values. <i>SmithChart</i>: Extrapolates the DC values based on an exact-approach method. <i>FirstPoint</i>: Extrapolates the DC values based on the DC value that is equal to the first non-zero point.
Enforce Impulse Response Causality	Instructs Tlsm to use the Hilbert transform to enforce the S-Parameters impulse response causality. This option is useful when you are simulating noisy S-Parameters that are not causal; that is, do not represent a physical system. Because enforcing causality modifies the original S-Parameter data, the default for this option is set to <i>Off</i> .

⚠ You can set environment variables at the system level to direct the behavior of Tlsm when running simulations:

SetTlsmTimeStep

Examples:

```
setenv SetTlsmTimeStep 10
setenv SetTlsmTimeStep 50
```

Description:

When set, Tlsm uses a specified time step in picoseconds for simulations

Related Topics

- [signal prefs](#)

Signal Prefs Tasks

You can use the `signal prefs` command to perform the following tasks:

- Setting Layers to be Ignored by the 3D Field Solver
- Specifying Glitch Settings
- Setting Adaptive Mesh Settings
- Setting up a Spectre/Hspice Simulation
- Running a Spectre/Hspice Simulation
- Setting Time-domain Voltage Ripple Display Settings
- Setting Frequency-domain Impedance Settings
- Setting Geometry Windows At The Drawing Level
- Setting Transient Simulation Preferences
- Setting and Detecting Coplanar Waveguides

Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)

Setting Layers to be Ignored by the 3D Field Solver

You can set which layers to be ignored by the 3D field solver by following these steps:

1. Choose *Analyze – 3-D Modeling*.
- or -
In *Allegro X Advanced Package DesignerXL*, choose *Analyze – 3-D Package Model*.
2. Click *Parameters*.
The 3-D Modeling Parameters dialog box appears.
3. Click the *SI Ignore Layers* tab.
4. In the *All Layers* list box on the left side, click on a layer that you want ignored.
The chosen layer moves to the *SI Ignore Layers* list box on the right side.
5. Repeat the previous step until all ignore layers are in the *SI Ignore Layers* list box.
6. Click *OK*.
The 3D Field Solver ignores the chosen layers in the stackup during simulation.

Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)
- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

Specifying Glitch Settings

To specify glitch settings:

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Click the *Simulations* tab.
3. Click *Advanced Measurements Settings*.
4. Enter a percentage value in the *Glitch tolerance* field.
5. Click *OK*.

Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)
- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

Setting Adaptive Mesh Settings

Follow these steps to set adaptive mesh settings:

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Click the *Power Integrity* tab.
3. Select a grid size value from the *Multinode grid size* field drop-down menu.
4. Select an adaptive mesh value from the *Adapt Level* field drop-down menu or enter a numerical value in the *Adapt Level* text field.

 Adaptive meshing does not support adaptive cell patterns higher than 256.

5. Click *OK*.
When simulation is complete, SigWave opens and displays the simulation results. By default, no trace is chosen.
6. If you select a trace in SigWave, the associated mesh cell in the design canvas of PCB SI is highlighted and zoomed into. –or– If you select a mesh cell in PCB SI, the associated trace in SigWave is highlighted.

Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)
- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

Setting up a Spectre/Hspice Simulation

Perform these steps to setup a Spectre/Hspice simulation:

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Set the standard preferences listed in the dialog box.
3. Click the *Simulation* tab.
4. From the *Simulator* drop-down menu, select either Spectre or Hspice.
The *Set Simulator Preferences* button becomes active.
5. Click *Set Simulator Preferences* to open the Advanced Simulator Preferences dialog box.
6. Set the control parameters for the conditions under which you want the simulator to run. These controls are described in the Advanced Simulator Preferences dialog box.
7. Close the dialog box.

Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)
- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

Running a Spectre/Hspice Simulation

Upon completing your setup for simulation, as described above, follow this use model to perform either Hspice or Spectre simulations.

1. Add the path to the simulator to your user \$PATH as well as to the paths for any libraries you may use.
2. Develop DML MacroModels for the IO buffer subcircuits. The basic composition of your models include:

- o Name of the MacroModel
Identical to the name of the 7-terminal subcircuit that you insert in the MacroModel section.
- o Body of the MacroModel
Identical to the body of any DML buffer MacroModel. It specifies basic IO buffer information, including (but not limited to):
Rise and fall times
Logic thresholds
Model types
Test fixtures

This is illustrated in the following example of a portion of a MacroModel body:

```
(Technology CMOS)
```

```
(Model
```

```
(ModelType IO)
```

```
(Polarity "Non-Inverting")
```

```
(Enable "Active-High"))
```

```
Logic Thresholds
```

```
(Output
```

```
(High
```

```
(typical 2.5))
```

```
....
```

- o MacroModel Subsection

Similar to an ESPICE MacroModel subsection, the difference being that you insert a simulator-specific 7 terminal wrapper subcircuit for the IO buffer rather than an ESPICE subcircuit. You must also indicate the simulator for which the MacroModel is targeted. This is illustrated in the following example of a MacroModel subsection:

```
(MacroModel
```

```
(NumberOfTerminals 7)
```

```
(language "simulator_name")
```

*The syntax of the subcircuit. If not specified, defaults to ESPICE syntax.

```
(SubCircuits "
```

* The Subcircuits section contains the 7-terminal subcircuit wrapper

* for the IO buffer.

simulator language=spice

.subckt <simulator_name>_out 1 2 3 4 5 6 7

*Calls the subcircuit containing the buffer's transistor-level model.

X_<simulator_name> 1 2 3 4 5 6 7 Any_<simulator_name>_transistor_model_subcircuit

.ends <simulator_name>_out

"))

1. From the SI Model Browser (*Analyze – Model Browser*) load the DML libraries that contain the IO buffer MacroModels, IbisDevices, and Packages.
2. Assign an IbisDevice to a component.
3. Edit the IbisDevice to assign the IO buffer models (or the DML MacroModel for a Spectre IO buffer) to the appropriate pins and, if necessary, to assign a package model to the IbisDevice.

⚠ If no IbisDevice is assigned to a component, you must use IO buffers targeted for your simulator type as defaults. Do this from the DeviceModels tab in the Analysis Preferences dialog box in PCB SI (*Analyze – Preferences*).

4. Set the simulator preferences from the controls in the Simulation tab of the Analysis Preferences dialog box and in the Advanced Simulator Preferences dialog box.
5. Perform the simulation, then generate and view the reports and waveforms.

Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)
- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

Setting Time-domain Voltage Ripple Display Settings

To set time-domain voltage ripple display settings:

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Click the *Power Integrity* tab.
3. If necessary, set a multi-node grid size and adaptive level, as described in [Setting Adaptive Mesh Settings](#).
4. In the *Simulator Preferences and Conditions* section of the dialog box, select *Time-Domain Voltage Ripple Display*.
The noise current pulse selections become enabled.
5. Follow the appropriate steps for setting up one of the pulses (you can select only one type of pulse).

Trapezoidal Noise Current Pulse

- a. Select *Trapezoidal Noise Current Pulse*.
The *Fastest Tr/Tff* field becomes enabled.
- b. Enter the smallest rise/fall time among all of the IC noise sources for establishing the trapezoidal current pulse. The default value is 500ps (0.5ns).

Gaussian Noise Current Pulse

- a. Select *GaussianNoise Current Pulse*.
The *Gaussian Width* field becomes enabled.
- b. Enter the time width for establishing the Gaussian current pulse. The default value is 1ns, which represents 1GHz of bandwidth.
6. Click *OK* to save your settings and close the Analysis Preferences dialog box.

Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)
- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

Setting Frequency-domain Impedance Settings

Follow these steps to set the frequency-domain impedance settings:

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Click the *Power Integrity* tab.
3. If necessary, set a multi-node grid size and adaptive level, as described in [Setting Adaptive Mesh Settings](#).
4. In the *Simulator Preferences and Conditions* section of the dialog box, select *Frequency-Domain Impedance Display*.
5. Click *OK* to save your settings and close the Analysis Preferences dialog box.

Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)
- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

Setting Geometry Windows At The Drawing Level

Follow these steps to set geometry windows at the drawing level:

1. Open a `.brd` or `.mcm` database file.
2. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
3. Click on the *Interconnect Models* tab, enter the desired value in the *Geometry Window* text box, then click *OK*.
4. Save the database file.
5. The file now contains a drawing-level GW value. This is the value that will be used for this package or board when a SI analysis is run for a system incorporating multiple `.mcm` and/or `.brd` files.

Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)
- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

Setting Transient Simulation Preferences

Perform these steps to set transient simulation preferences:

1. Open a .brd or .mcm database file.
2. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
3. Click on the *S-Parameters* tab, select the simulation and extrapolation methods of your choice and whether you want to enable impulse response causality.
4. Click *OK* to save your settings and close the Analysis Preferences dialog box.

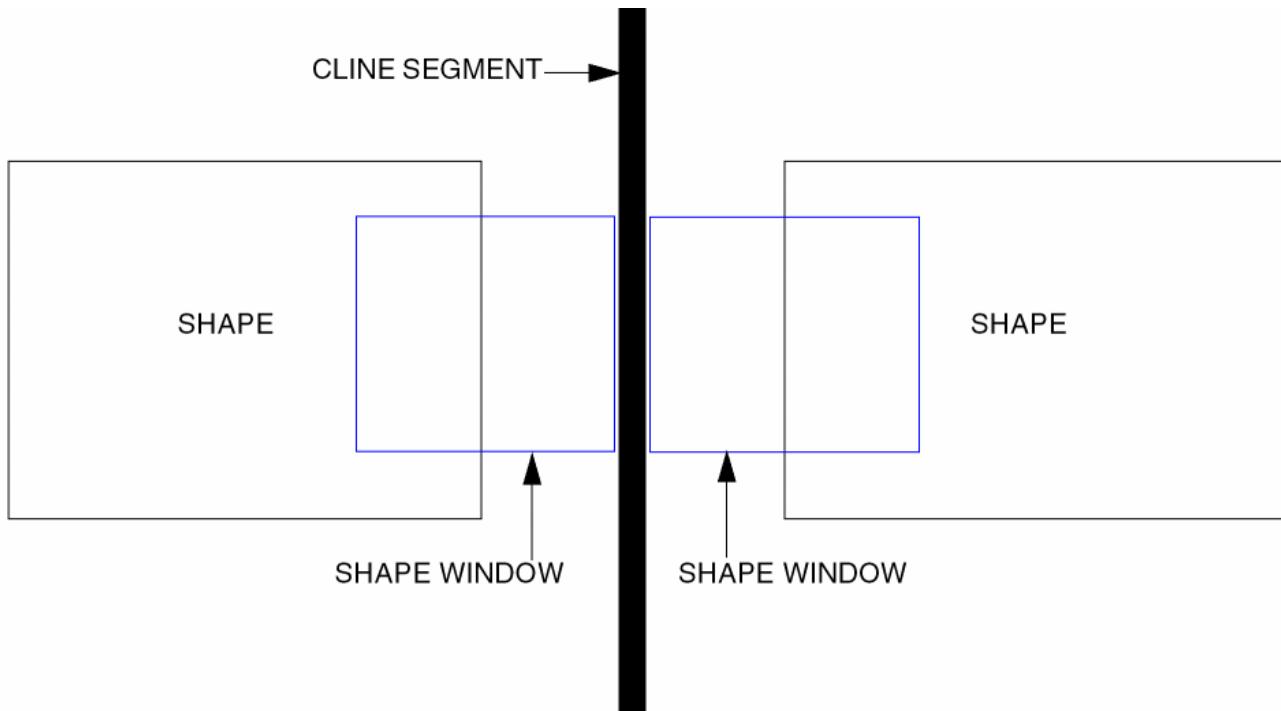
Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)
- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

Setting and Detecting Coplanar Waveguides

To set and detect coplanar waveguides, perform these steps:

1. Choose *Analyze – Preferences*.
The Analysis Preferences dialog box appears.
2. Click the InterconnectModels tab.
3. Select *Enable CPW Extraction* and *Ems2d FW* to enable coplanar waveguides in the entire design.
4. If you wish to disable CPW for specific nets, do the following for each selected net, otherwise proceed to step 5.
 - a. Right-click on the net you want to apply the *CPW_DISABLED* property to.
 - b. Choose *Property Edit* from the pop-up menu.
The Edit Property dialog box opens.
 - c. Select *Cpw_Disabled* from the Available Properties list and click *Apply*.
The selected net will now be handled during analysis as a non-CPW net. If you have selected only the *Ems2dFW* option (without *Enable CPW Extraction*), non-CPW nets will be generated with *Bem2d*.
5. Set the Geometry Window parameter to accommodate the configuration of DC shapes surrounding the cline segment, as shown in the graphic.



For each segment of the cline, Ems2d will use the dimensions set in the Geometry Window to check for shapes on either side of the cline.

6. Click the *Preferences* button to open the EMS2D Preferences dialog box.
7. Choose the frequency settings and other options appropriate for your analysis. These settings are explained in the [EMS2D Preferences Dialog Box](#) section.
8. Click *OK*.

Related Topics

- [signal prefs](#)
- [Signal Prefs Dialog Boxes](#)
- [Analysis Preferences Dialog Box](#)
- [Signal Prefs Tasks](#)

signal snrscreen

The `signal snrscreen` command is no longer in use. Use the [signal probe](#) command to launch the [Signal Analysis Dialog Box](#) dialog from where you can run the Signal Screening command to start the signal quality signal process for the selected nets.

signal report

The `signal report` command is no longer in use. Use the `signal probe` command to launch the Signal Analysis dialog box from where you can launch the *Analysis Report Generator* dialog to generate signal analysis reports.

Related Topics

- [signal probe](#)
- [Signal Analysis Dialog Box](#)
- [*Analysis Report Generator Dialog Box*](#)

signal probe

The `signal probe` command displays the Signal Analysis dialog box as the starting point for performing signal integrity and EMI emissions simulations. You use the Signal Analysis dialog box to choose nets and driver-receiver combinations for analysis. You also open the Signal Analysis [case x] and Analysis Report Generator [case x] dialog boxes from the Signal Analysis dialog box. In the Signal Analysis or Analysis Report Generator dialog box, you choose which waveforms or reports to generate. The simulator performs the necessary simulations.

You can also start the SigXplorer topology editor and the sigxsect interconnect cross-section viewer from the Signal Analysis dialog box. Use SigXplorer to perform what-if studies on different driver and receiver combinations and transmission line scenarios. Use `sigxsect` to display cross-sections of routed interconnect segments.

 This command is not available in L Series products

Related Topics

- [Signal Analysis Tasks](#)

Signal Probe Dialog Boxes

Access Using

- Toolbar Icon: 

Signal Analysis Dialog Box	Signal Select Browser Dialog Box	Analysis Report Generator Dialog Box
Analysis Waveform Generator	Stimulus Setup Dialog Box	

Signal Analysis Dialog Box

<i>Net:</i>	Specifies a net name or a net name match pattern. New names and match patterns are added to the pull down list of names.
<i>List of Nets</i>	Displays a standard file browser set to display netlist files.
<i>Net Browser</i>	Displays the Signal Select Browser dialog box..
<i>Nets</i>	Lists the chosen nets.
<i>Driver Pins</i>	Lists all driver pins on the net highlighted in the Nets list box.
<i>Load Pins</i>	Lists the receiver pins (or loads) seen by the driver pin highlighted in the Driver Pins list box.
<i>Other Pins</i>	Lists all the pins seen by the driver pin highlighted in the Driver Pins list box that are not driver or receiver pins. These pins usually have the UNSPEC pinuse.
<i>Reports</i>	Starts the Analysis Report Generator (case x) dialog box, for defining, generating, viewing, and storing text reports of simulation results.
<i>Waveforms</i>	Starts the Signal Analysis [case x] dialog box for defining, generating, viewing, and storing simulation results as waveforms.
<i>View Topology</i>	Starts the SigXplorer topology modeling interface for the chosen signal.
<i>Signal Screening</i>	Starts the signal quality signal process for the selected nets. You can perform signal quality screening on a set of nets which could either be single-ended or part of a differential pair.

Related Topics

- [Signal Select Browser Dialog Box](#)

S Commands

S Commands--signal probe

Signal Select Browser Dialog Box

<i>Net Filter</i>	Specifies a net name match pattern for choosing nets from the design.
<i>Apply</i>	Displays in the Available Nets list box a list of Xnets that match the pattern in the Net Filter field.
<i>Available Nets</i>	Lists the available Xnets.
<i>Selected Nets</i>	Lists the chosen Xnets.
<- All	Moves all chosen Xnets in the Selected Nets list box to the Available Nets list box.
All ->	Moves all chosen Xnets in the Available Nets list box to the Selected Nets list box.

Analysis Report Generator Dialog Box

Use the Analysis Report Generator to create text-formatted reports on various simulation scenarios. The current case displays in the title bar.

Use the *Standard Report* tab to generate one of eleven standard reports. Use the *Custom Report* tab to set up the order of columns and their specific contents.

[Standard Report Tab] [Custom Report Tab]

Standard Report Tab

Case Selection Area

Current Case	Choose a simulation case from a list of available cases. Choosing a case establishes it as the current case.
--------------	--

Report Types Area

Use this area to specify the report types you want to generate.

Reflection Summary	Includes Noise Margin high and low, Overshoot high and low, Switch and Settle rise and fall, and Pass-Fail for monotonic edge.
Delay	Includes Propagation Delay, Switch and Settle rise and fall, and monotonicity pass or fail.
Ringing	Includes Noise Margin high and low, Overshoot high and low, and driver and load I/O characteristics.
Single Net EMI	Shows the maximum EMI emission level for the computed frequency level.
Parasitics	Includes Parasitic impedance, capacitance.
SSN	Includes Simultaneous Switching Noise Fall Time, Rise Time, Low State and High State Logic Input and Output Threshold
SDF Wire Delay	Standard Delay Format report.
Segment Crosstalk	Presents detailed segment-based coupling information derived from crosstalk tables.
Crosstalk Summary	Delivers a shortened crosstalk report, identifying the chosen victim xnet and driver, and reporting on high and low state crosstalk for odd and even stimulus.
Crosstalk Detailed	Identifies the chosen victim xnet, drivers, and all receivers, and reports on high and low state crosstalk for odd and even stimulus. It also provides information on the devices and models used, their electrical characteristics, the default simulation settings, and a full glossary of abbreviations.

Fast/Typical/Slow Mode Area

Use this area to choose simulation speed. Fast, typical, and slow simulation mode parameters are defined in the Fast/Typical/Slow Simulations Definition dialog box.

Fast	Performs simulations in Fast mode.
Typical	Performs simulations in Typical mode.
Slow	Performs simulations in Slow mode.
Fast/Slow	Performs simulations in Fast mode for the driver and Slow mode for the receiver.
Slow/Fast	Performs simulations in Slow mode for the driver and Fast mode for the receiver.

Victim Area

Use this area to designate nets and drivers as potential victims for crosstalk checking.

Net Selection	Selects the victim nets. The nets to be monitored. All Selected Nets selects all nets shown in the Signal Analysis Dialog Box . Highlighted Net Only is the net highlighted in the Signal Analysis dialog box.
---------------	--

S Commands

S Commands--signal probe

Driver Selection	Selects the driver to stimulate when the victim net or nets are held at the high state. Choices are: Fastest Driver, Highlighted Driver Only, and All Xnet Drivers.
-------------------------	---

Aggressor Area

Use this area to designate aggressor nets and define their switching behavior.

Switch Mode	Choices are: Odd, Even, or Odd/Even switching direction for drivers on aggressor nets relative to the victim driver. Odd means aggressor drivers move to the opposite state of that in which the victim net is held. The default is Odd. Even means aggressor drivers move to the same state as that in which the victim net is held. Odd/Even means that simulations are run with both odd and even switching.
Net Selection	Choices are All/Group Neighbors or Each Neighbor. All/Group Neighbors stimulates all neighbor nets at once. Each Neighbor reports on the individual effect of each neighboring net on the victim net.
Driver Selection	Selects the driver on each aggressor xnet to be stimulated in Crosstalk simulations. Choices are: Fastest Driver and All Drivers. Net Selection must be Each Neighbor to choose All Drivers. When Net Selection is All/Group Neighbors, the fastest driver is always used.

Reflection Data Simulation Area

Type	Reflection simulates only a primary net and none of the neighboring nets. Reflection simulation does not take the parasitics of power and ground pins into account. <i>Comprehensive Odd or Even</i> specifies that SigNoise run simulations of the primary net (the one chosen) and its neighbor nets at the same time. Comprehensive simulation also takes power and ground parasitics into account. If Odd, SigNoise applies the stimulus type you choose to the primary net and the opposite to the neighbor nets. If Even, SigNoise applies the stimulus type you choose to the primary net and the neighbor nets simultaneously. <i>Comprehensive Static</i> simulates a primary net while holding the neighboring nets in a steady state (the low state). This simulation mode accounts for loading due to coupling from neighboring nets.
Measurement	<i>Pulse</i> measures the rise and fall of the cycle number specified in Preferences Pulse Cycle Count. <i>Rise - Fall</i> measures the first rise and first fall in a cycle. <i>Custom Stimulus</i> lets you define pulse parameter data (specifically; values for net and Xnet, frequency, count cycle, offset, jitter, and bit pattern) at the board level. Choosing this option actives the <i>Assign</i> button which opens the Stimulus Setup Dialog Box .

General

Use Timing Windows	If checked, timing window properties are used to refine the crosstalk simulations to account for received crosstalk that is insignificant due to the timing of signals. The default is No.
Save Circuit Files	If checked, tIsim and SPICE circuit files are retained in the case directory for each simulation performed.
Save Waveforms	If checked, waveform files are retained in the case directory for each simulation performed.
Create Report	Creates the text reports as specified.
Preferences	Opens the Analysis Preferences dialog box.

Custom Report Tab

Case Selection Area

Current Case	Choose from a list of available cases. Choosing a case establishes it as the current case.
---------------------	--

Report Area

Report name	Lists existing Custom Reports. The default is CustomRpt, the default Custom Report format.
Clone Selected Report	Makes a copy of the chosen report under a new name.

S Commands

S Commands--signal probe

Sort By	Choice of field on which to sort the table of simulation results. The sort order puts worst case values at the top of the report. The default is to sort on minimum noise margin. You can choose from 28 choices listed in the pulldown menu.
New Custom Report	Names a new, blank Custom Report.

Fast Typical Slow Mode Area

Use this area to choose simulation speed. Fast, typical, and slow simulation mode parameters are defined in the Fast/Typical/Slow Simulations Definition dialog box.

<i>Fast</i>	Reports data for simulations performed in Fast mode.
<i>Typical</i>	Reports data for simulations performed in Typical mode.
<i>Slow</i>	Reports data for simulations performed in Slow mode.
<i>Fast/Slow</i>	Reports data for simulations performed in Fast mode for the driver and Slow mode for the receiver.
<i>Slow/Fast</i>	Reports data for simulations performed in Slow mode for the driver and Fast mode for the receiver.

Victim Area

Use this area to designate nets and drivers as potential victims for crosstalk checking.

<i>Net Selection</i>	Selects the victim nets. The nets to be monitored. All Selected Nets selects all nets shown in the Signal Analysis Dialog Box . Highlighted Net Only is the net highlighted in the Signal Analysis dialog box.
<i>Driver Selection</i>	Selects the driver to stimulate when the victim net or nets are held at the high state. Choices are: Fastest Driver, Highlighted Driver Only, and All Xnet Drivers.

Simulation Data Table Area

Use this area to define the contents of the report's 8 data columns. The default content for each column is as follows:

<i>Column 1</i>	The Xnet name.
<i>Column 2</i>	The driver pin name.
<i>Column 3</i>	The receiver pin name.
<i>Column 4</i>	The high state noise margin value for a rising edge.
<i>Column 5</i>	The low state noise margin value for a falling edge.
<i>Column 6</i>	The high state overshoot value for a rising edge.
<i>Column 7</i>	The low state overshoot value for a falling edge.
<i>Column 8</i>	The high state crosstalk value measured for a rising edge in odd switching mode.

The choices in the pull down menu for each column are as follows.

<i>Blank</i>	An empty column.
<i>Xnet</i>	The Xnet name.
<i>Driver Pin</i>	The driver pin name.
<i>Receiver Pin</i>	The receiver pin name.
<i>Receiver Mode</i>	The state of the receiver, either Active or Standby.
<i>Strobe Xnet</i>	The strobe extended net name
<i>Aggressor Net</i>	The name of any neighboring aggressor net involved in the simulation.

S Commands
S Commands--signal probe

<i>Segment</i>	The victim segment used for segment crosstalk analysis.
<i>Layers</i>	The victim layer and the aggressor layer for the segment.
<i>Propagation Delay</i>	The propagation delay value measured for the net.
<i>Switch Rise Delay</i>	The first switch delay value measured for a rising edge.
<i>Switch Fall Delay</i>	The first switch delay value measured for a falling edge.
<i>Min Switch Delay</i>	The smaller of the two first switch delay values.
<i>Settle Rise Delay</i>	The final settle delay value measured for a rising edge.
<i>Settle Fall Delay</i>	The final settle delay value measured for a falling edge.
<i>Max Settle Delay</i>	The larger of the two final settle delay values.
<i>FirstIncident Rise</i>	A PASS or FAIL status indicating whether a first incident rule violation exists for the rising edge.
<i>FirstIncident Fall</i>	A PASS or FAIL status indicating whether a first incident rule violation exists for the falling edge.
<i>Monotonic Rise</i>	A PASS or FAIL status indicating whether the rising edge is non-monotonic.
<i>Monotonic Fall</i>	A PASS or FAIL status indicating whether the falling edge is non-monotonic.
<i>Monotonic</i>	A PASS or FAIL status indicating that both rise and fall edges are non-monotonic or that one or both are not.
<i>Glitch Rise</i>	The tolerance check on the rising waveform. If no glitch occurs in the rising waveform, the report denotes a PASS. If one does occur, it reports a FAIL.
<i>Glitch Fall</i>	The tolerance check on the falling waveform. If no glitch occurs in the falling waveform, the report denotes a PASS. If one does occur, it reports a FAIL.
<i>Glitch</i>	The tolerance check of the rising and falling waveform
<i>Maximum Overshoot</i>	The high state overshoot value for a rising edge.
<i>Minimum Overshoot</i>	The low state overshoot value for a falling edge.
<i>Noise Margin High</i>	The high state noise margin value for a rising edge.
<i>Noise Margin Low</i>	The low state noise margin value for a falling edge.
<i>Min Noise Margin</i>	The smaller of the two first noise margin values.
<i>HighState Odd Xtalk</i>	The high state crosstalk value measured for a rising edge in odd switching mode.
<i>LowState Odd Xtalk</i>	The low state crosstalk value measured for a falling edge in odd switching mode.
<i>HighState Even Xtalk</i>	The high state crosstalk value measured for a rising edge in even switching mode.

S Commands

S Commands--signal probe

<i>LowState Even Xtalk</i>	The low state crosstalk value measured for a falling edge in even switching mode.
<i>Maximum Crosstalk</i>	The largest of the four crosstalk values.
<i>Simulation Id</i>	The simulation identifier.

Setup Data Table Area

Use this area to define the content of the 4 columns the in the report's data table. The default content for each column is as follows:

<i>Column 1</i>	The pin name
<i>Column 2</i>	The net name
<i>Column 3</i>	The device name
<i>Column 4</i>	The IOCell model assigned to the pin

The choices in the pull down menu for each column are as follows.

<i>Blank</i>	An empty column
<i>XNet</i>	The Xnet name
<i>Pin</i>	The pin name
<i>Net</i>	The net name
<i>Pin Use</i>	The pinuse code assigned to the pin.
<i>Device</i>	The name of the device of which the pin is a part.
<i>Junction Temperature</i>	The reference temperature from the IOCell model assigned to the pin.
<i>Diff Pair Mate</i>	The name of any pin that is the differential pair mate of the pin.
<i>IOCell Model Name</i>	Name of the IOCell model assigned to the pin.
<i>LowOutput Logic Threshold</i>	The low output logic threshold for the PullDown VI curve of the IOCell model assigned to the pin.
<i>HighOutput Logic Threshold</i>	The high output logic threshold for the PullUp VI curve of the IOCell model assigned to the pin.
<i>LowInput Logic Threshold</i>	The low input logic threshold (Vin low) from the IOCell model assigned to the pin.
<i>HighInput Logic Threshold</i>	The high input logic threshold (Vin high) from the IOCell model assigned to the pin.

General

<i>Use Timing Windows</i>	If checked, timing window properties are used to refine the crosstalk simulations to account for received crosstalk that is insignificant due to the timing of signals. The default is No.
<i>Save Circuit Files</i>	If checked, t1sim and SPICE circuit files are retained in the case directory for each simulation performed.
<i>Save Waveforms</i>	If checked, waveform files are retained in the case directory for each simulation performed.
<i>Create Report</i>	Generates the report as specified in the dialog box.
<i>OK</i>	Saves the named Custom Report format and adds it to the Report name pulldown list.

Analysis Waveform Generator

Use the tabs on this dialog box to generate waveforms for the currently chosen case. The case name is shown in the title bar. The dialog box allows default setups for various simulation scenarios.

[[Common Buttons \(displayed regardless of the tab chosen\)](#)] [[Reflection Tab](#)] [[Comprehensive Tab](#)] [[Crosstalk Tab](#)] [[SSN Tab](#)] [[EMI Single Tab](#)] [[Related Topics](#)]

Common Buttons (displayed regardless of the tab chosen)

Create Waveforms	Starts simulation.
View Waveform	Starts the SigWave window and loads a waveform file chosen from the list box.
Preferences	Displays the Analysis Simulation Preferences dialog box.

Reflection Tab

Performs a Reflection simulation for a primary net and none of the neighboring nets. Reflection simulation does not take the parasitics of power and ground pins into account.

You can also access the Stimulus and Fast/Typical/Slow Mode options on the SigNoise Reflection tab from the SigXplorer Simulation – Preferences command.

Case Selection Area

Current Case	Choose from a list of available cases. Choosing a case establishes it as the current case.
--------------	--

Stimulus Area

Stimulus	Choose a type of stimulus. Choices are: <i>Rise</i> , <i>Fall</i> , <i>Rise/Fall</i> , <i>Pulse</i> , <i>Inverted Pulse</i> , or <i>Custom</i> . <i>Custom Stimulus</i> lets you define pulse parameter data (specifically; values for net and Xnet, frequency, count cycle, offset, jitter, and bit pattern) at the board level. Choosing this option activates the <i>Assign</i> button which opens the Stimulus Setup dialog box.
----------	--

Fast/Typical/Slow Mode Area

Use this area to choose simulation speed. Fast, typical, and slow simulation mode parameters are defined in the Fast/Typical/Slow Simulations Definition dialog box.

Fast	Performs simulations in Fast mode.
Typical	Performs simulations in Typical mode.
Slow	Performs simulations in Slow mode.
Fast/Slow	Performs simulations in Fast mode for the driver and Slow mode for the receiver.
Slow/Fast	Performs simulations in Slow mode for the driver and Fast mode for the receiver.

Victim Area

Use this area to designate nets and drivers as potential victims for crosstalk checking.

Net Selection	Choose the victim nets. The nets to be monitored. All Selected Nets selects all nets shown in the Signal Analysis dialog box. Highlighted Net Only is the net highlighted in the Signal Analysis dialog box.
Driver Selection	Choose the driver to stimulate when the victim net or nets are held at the high state. Choices are: Fastest Driver, Highlighted Driver Only, and All Xnet Drivers.

Other

Save Circuit Files	If checked, tisim circuit files are retained in the case directory for each simulation performed.
--------------------	---

S Commands

S Commands--signal probe

Create Waveforms Start the designated simulation or simulations.

Comprehensive Tab

Performs a Comprehensive simulation for the primary chosen net or nets, and neighbor nets at the same time. Results show glitches in the primary net produced by activity on the neighbor nets.

Case Selection Area

Current Case Choose from a list of available cases. Choosing a case establishes it as the current case.

Stimulus Area

Stimulus Choose a type of stimulus. Choices are: *Rise*, *Fall*, *Rise/Fall*, *Pulse*, *Inverted Pulse*.

Fast Typical Slow Mode Area

Use this area to choose simulation speed. Fast, typical, and slow simulation mode parameters are defined in the Fast/Typical/Slow Simulations Definition dialog box.

<i>Fast</i>	Performs simulations in Fast mode.
<i>Typical</i>	Performs simulations in Typical mode.
<i>Slow</i>	Performs simulations in Slow mode.
<i>Fast/Slow</i>	Performs simulations in Fast mode for the driver and Slow mode for the receiver.
<i>Slow/Fast</i>	Performs simulations in Slow mode for the driver and Fast mode for the receiver.

Victim Area

Use this area to designate nets and drivers as potential victims for crosstalk checking.

Net Selection	Choose the victim nets. The nets to be monitored. All Selected Nets selects all nets shown in the Signal Analysis dialog box. Highlighted Net Only is the net highlighted in the Signal Analysis dialog box.
Driver Selection	Choose the driver to stimulate when the victim net or nets are held at the high state. Choices are: Fastest Driver, Highlighted Driver Only, and All Xnet Drivers.

Aggressor Area

Use this area to designate aggressor nets and define their switching behavior.

Switch Mode	Choices are: Odd, Even, Odd/Even, or Static switching direction for drivers on aggressor nets relative to the victim driver. Odd means aggressor drivers move to the opposite state of that in which the victim net is held. The default is Odd. Even means aggressor drivers move to the same state as that in which the victim net is held. Odd/Even means that simulations are run with both odd and even switching. Static means that aggressor nets are held in a steady state during simulation. This switching mode accounts for loading due to coupling from neighboring aggressor nets.
--------------------	--

Other

Save Circuit Files If checked, tisim circuit files are retained in the case directory for each simulation performed.

Crosstalk Tab

Performs a Crosstalk simulation for the primary chosen net or nets, and neighbor nets.

Case Selection Area

Current Case Choose from a list of available cases. Choosing a case establishes it as the current case.

Stimulus Area

S Commands

S Commands--signal probe

Stimulus	Choose a type of stimulus. Choices are: Rise, Fall, Rise/Fall, Pulse, or Inverted Pulse
-----------------	---

Fast Typical Slow Mode

Use this area to choose simulation speed. Fast, typical, and slow simulation mode parameters are defined in the Fast/Typical/Slow Simulations Definition dialog box.

<i>Fast</i>	Performs simulations in Fast mode.
<i>Typical</i>	Performs simulations in Typical mode.
<i>Slow</i>	Performs simulations in Slow mode.
<i>Fast/Slow</i>	Performs simulations in Fast mode for the driver and Slow mode for the receiver.
<i>Slow/Fast</i>	Performs simulations in Slow mode for the driver and Fast mode for the receiver.

Victim Area

Use this area to designate nets and drivers as potential victims for crosstalk checking.

<i>Net Selection</i>	Choose the victim nets. The nets to be monitored. All Selected Nets selects all nets shown in the Signal Analysis dialog box. Highlighted Net Only is the net highlighted in the Signal Analysis dialog box.
<i>Driver Selection</i>	Choose the driver to stimulate when the victim net or nets are held at the high state. Choices are: Fastest Driver, Highlighted Driver Only, and All Xnet Drivers.

Aggressor Area

Use this area to designate aggressor nets and define their switching behavior.

<i>Switch Mode</i>	Choices are: Odd, Even, or Odd/Even switching direction for drivers on aggressor nets relative to the victim driver. Odd means aggressor drivers move to the opposite state of that in which the victim net is held. The default is Odd. Even means aggressor drivers move to the same state as that in which the victim net is held. Odd/Even means that simulations are run with both odd and even switching.
<i>Net Selection</i>	Choices are All/Group Neighbors or Each Neighbor. All/Group Neighbors stimulates all neighbor nets at once. Each Neighbor reports on the individual effect of each neighboring net on the victim net.
<i>Driver Selection</i>	Selects the driver on each aggressor xnet to be stimulated in Crosstalk simulations. Choices are: Fastest Driver and All Drivers. Net Selection must be Each Neighbor to choose All Drivers. When Net Selection is All/Group Neighbors, the fastest driver is always used.

Other

<i>Save Circuit Files</i>	If checked, t1sim circuit files are retained in the case directory for each simulation performed.
<i>Use Timing Windows</i>	If checked, timing window properties are used to refine the crosstalk simulations to account for received crosstalk that is insignificant due to the timing of signals. The default is No.

SSN Tab

Performs a SSN simulation for the chosen net or nets.

Case Selection Area

<i>Current Case</i>	Choose from a list of available cases. Choosing a case establishes it as the current case.
---------------------	--

Stimulus Area

<i>Stimulus</i>	Choose a type of stimulus. Choices are: Rise, Fall, Rise/Fall, Pulse, or Inverted Pulse
-----------------	---

Fast Typical Slow Mode

Use this area to choose simulation speed. Fast, typical, and slow simulation mode parameters are defined in the Fast/Typical/Slow Simulations

S Commands
S Commands--signal probe

Definition dialog box.

<i>Fast</i>	Performs simulations in Fast mode.
<i>Typical</i>	Performs simulations in Typical mode.
<i>Slow</i>	Performs simulations in Slow mode.
<i>Fast/Slow</i>	Performs simulations in Fast mode for the driver and Slow mode for the receiver.
<i>Slow/Fast</i>	Performs simulations in Slow mode for the driver and Fast mode for the receiver.

Victim Area

Use this area to designate nets and drivers as potential victims for crosstalk checking.

<i>Net Selection</i>	Choose the victim nets. The nets to be monitored. All Selected Nets selects all nets shown in the Signal Analysis dialog box. Highlighted Net Only is the net highlighted in the Signal Analysis dialog box.
<i>Driver Selection</i>	Choose the driver to stimulate when the victim net or nets are held at the high state. Choices are: Fastest Driver, Highlighted Driver Only, and All Xnet Drivers.

Other

<i>Save Circuit Files</i>	If checked, t1sim circuit files are retained in the case directory for each simulation performed.
---------------------------	---

EMI Single Tab

Performs a Reflection simulation for a single net to evaluate the differential mode radiated emissions for the net.

Case Selection Area

<i>Current Case</i>	Choose from a list of available cases. Choosing a case establishes it as the current case.
---------------------	--

Stimulus Area

<i>Stimulus</i>	Pulse simulation only.
-----------------	------------------------

Fast/Typical/Slow Mode Area

Use this area to choose simulation speed. Fast, typical, and slow simulation mode parameters are defined in the Fast/Typical/Slow Simulations Definition dialog box.

<i>Fast</i>	Performs simulations in Fast mode.
<i>Typical</i>	Performs simulations in Typical mode.
<i>Slow</i>	Performs simulations in Slow mode.
<i>Fast/Slow</i>	Performs simulations in Fast mode for the driver and Slow mode for the receiver.
<i>Slow/Fast</i>	Performs simulations in Slow mode for the driver and Fast mode for the receiver.

Victim Area

Use this area to designate nets and drivers as potential victims for crosstalk checking.

<i>Net Selection</i>	Choose the victim nets. The nets to be monitored. All Selected Nets selects all nets shown in the Signal Analysis dialog box. Highlighted Net Only is the net highlighted in the Signal Analysis dialog box.
<i>Driver Selection</i>	Choose the driver to stimulate when the victim net or nets are held at the high state. Choices are: Fastest Driver, Highlighted Driver Only, and All Xnet Drivers.

Other

<i>Save Circuit Files</i>	If checked, t1sim circuit files are retained in the case directory for each simulation performed.
---------------------------	---

Related Topics

- [Signal Analysis Dialog Box](#)

Stimulus Setup Dialog Box

Use this dialog box to assign and/or edit stimulus values for the nets/xnets listed in the Xnet column. These are the nets that you selected directly from the canvas of the active design, or from a netlist or a net browser in the Signal Analysis dialog box. The values that you set here override any that might be configured in a `.inc` custom stimulus file.

Filter fields	The filtering fields above each column let you specify an expression string or choose from a supported value. The parameters you specify determine which nets are listed in the display.
Columns	For each netlisted in the <i>Xnet</i> column, you can assign or edit values for <i>Frequency</i> , <i>Duty Cycle</i> , <i>Cycle Count</i> , <i>Offset</i> , <i>Jitter</i> , and <i>Bit Pattern</i> .
Assign fields	These fields let you assign to all displayed nets the value that you enter for the associated column. You can also select a value from the drop-down menu when more than a single value is supported. ⚠ If a value you enter is not valid for that column (for example, you cannot enter more than 100% for <i>Duty Cycle</i>), an error message displays across the bottom of the dialog box.
Bit Pattern drop-downs	These drop-downs contain a selection that lets you enter bit patterns of various lengths. Choosing <i>Random</i> displays a pop-up in which you can enter a pattern-length up to 1,024 bits.
Export	This function lets you save your <i>displayed</i> stimulus settings to an Excel spreadsheet with a <code>.csv</code> extension. Nets that you have filtered out of the columns do not get copied to the <code>.csv</code> file.
Import	This function lets you copy the contents of a <code>.csv</code> file into the Stimulus Setup dialog box. The spreadsheet file you are importing must contain only valid data. For example, non-existent nets are not added.

Related Topics

- [Signal Analysis Dialog Box](#)

Signal Analysis Tasks

Following are tasks associated with the analysis probe commands:

- [Choosing a Group of Nets for Analysis](#)
- [Choosing a Single Net or Pin Pair for Analysis](#)
- [Choosing Groups of Violation Nets for Analysis](#)
- [Choosing Xnets for Simulation](#)
- [Setting Custom Stimulus for Report Generation](#)
- [Setting Custom Stimulus for Waveform Generation](#)
- [Creating Waveforms](#)
- [Creating Standard Reports](#)
- [Creating Custom Reports](#)

S Commands

S Commands--signal probe

- Performing Signal Quality Screening
- Starting SigXplorer from the Simulator

Related Topics

- [signal probe](#)

Choosing a Group of Nets for Analysis

To choose a group of nets for analysis:

1. Run `signal probe`.
The Signal Analysis dialog box appears.
2. In the Signal Analysis dialog box, click *List of Nets*.
An Open file browser appears.
3. In the browser, choose a filter or enter a netlist file name of the type `<list_file_name>.lst`.
4. Choose a netlist file name and click *Open*.
The nets highlight in the design window and the names of the nets, driver pins, and receiver pins appear in the Nets, Driver Pins, and Load Pins list boxes in the Signal Analysis dialog box.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

Choosing a Single Net or Pin Pair for Analysis

To choose a single net or pin pair for analysis, follow these steps:

1. Run `signal probe`.
The Signal Analysis dialog box appears.
2. In the Signal Analysis dialog box, enter a net name or a net name match pattern in the Net: field or choose a net or net name match pattern from the pulldown menu.
The names of the net, and driver and receiver pins appear in the Nets, Driver Pins, and Load Pins list boxes.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

Choosing Groups of Violation Nets for Analysis

Follow these steps to choose groups of violation nets for analysis:

1. Run `signal probe`.
The Signal Analysis dialog box appears.
2. In the Signal Analysis dialog box, click *List of Nets*.
The Open file browser appears.
3. Choose a violation netlist file name of the type `<scanViolationNets>.lst` and click *Open*.
The nets highlight in the design window and the names of the nets, driver pins, and receiver pins appear in the *Nets*, *Driver Pins*, and *Load Pins* list boxes in the Signal Analysis dialog box.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

Choosing Xnets for Simulation

Perform the following steps to choose xnets for simulation:

1. Run `signal probe`.
The Signal Analysis dialog box appears.
2. In the Signal Analysis dialog box, click *Net Browser*.
The Signal Select Browser appears.
3. In the browser, edit the contents of the Net Filter field and click *Apply* to fill in the Available Nets list box.
4. In the Available Nets list box, click individual nets to move them to the Selected Nets list box. - or - Use *All ->* and *<-All* to move all nets to either list.
5. Click *OK*.
The nets highlight in the design window and the names of the nets, driver pins, and receiver pins appear in the Nets, Driver Pins, and Load Pins list boxes in the Signal Analysis dialog box.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

Setting Custom Stimulus for Report Generation

You can set custom stimulus for report generation by following these steps:

1. Run `signal probe`.

The Signal Analysis dialog box appears.

2. Select the nets to which to assign custom stimulus. You can do so in any of the following ways:

- Directly from the design canvas

The nets you have selected appear in the list windows of the Signal Analysis dialog.

3. Click the *Reports* button to display the Analysis Report Generator.

4. In the *Reflection Data Simulation:Measurement* area of the *Standard Report* tab, choose *Custom Stimulus*.

The *Assign* button is enabled.

5. Click *Assign*.

The Stimulus Setup dialog box appears with the nets previously selected displayed in the columns.

6. Assign or modify the stimuli for selected nets, as described in [Stimulus Setup Dialog Box](#).

Column cells that you make changes to are highlighted in yellow until you apply your changes.

7. To save the custom stimulus to an editable spreadsheet file, choose the *Export* button as described in the [Stimulus Setup Dialog Box](#).

8. To import custom stimulus for valid nets into the dialog box, choose the *Import* button as described in the [Stimulus Setup Dialog Box](#).

9. Upon completion, click *OK* to close the dialog box with your changes.

The custom stimulus for the selected nets are recorded in the Reflection reports.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

Setting Custom Stimulus for Waveform Generation

Follow these steps to set custom stimulus for waveform generation:

1. Run `signal probe`.
The Signal Analysis dialog box appears.
2. Select the nets to which to assign custom stimulus. You can do so in any of the following ways:
 - Directly from the design canvas
The nets you have selected appear in the list windows of the Signal Analysis dialog.
3. Click the *Waveforms* button to display the Analysis Waveform Generator.
4. In the *Stimulus* area of the *Reflection* tab, choose *Custom* from the drop-down menu.
The *Assign* button is enabled.
5. Click *Assign*.
The Stimulus Setup dialog box appears with the nets previously selected displayed in the columns.
6. Assign or modify the stimuli for selected nets, as described in [Stimulus Setup Dialog Box](#).
Column cells that you make changes to are highlighted in yellow until you apply your changes.
7. To save the custom stimulus to an editable spreadsheet file, choose the *Export* button as described in [Stimulus Setup Dialog Box](#).
8. To import custom stimulus for valid nets into the dialog box, choose the *Import* button as described in [Stimulus Setup Dialog Box](#).
9. Upon completion, click *OK* to close the dialog box with your changes.
The custom stimulus for the selected nets are displayed in the waveform.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

Creating Waveforms

To create waveforms:

1. Run `signal probe`.
The Signal Analysis dialog box appears.
2. Display net and pin names in the Nets, Driver Pins, and Load Pins list boxes.
3. Use the *Net*:field to enter a name match pattern to choose nets from the design. -or- Use *List of Nets* to browse for a netlist file. -or- Use *Net Browser* to choose Xnets from the design.
One or more net names appear in the Nets list box.
4. Choose a net to simulate.
5. In the Nets list box, click to choose a net name. -or- Directly in the design window, click on a net to choose it.
The net name highlights in the Nets list box and in the design window. The names of driver pins and load pins on the highlighted net appear in the Driver Pins and Load Pins list boxes.
6. Choose a connection to simulate.
7. In the Driver Pins and Load Pins list boxes, click to choose a connection to simulate.
The chosen pin names highlight in the Driver Pins and Load Pins list boxes and in the design window.

Entering Simulation Details

1. In the Signal Analysis dialog box, click *Waveforms*.
The Signal Analysis [case x] dialog box appears. The current case is named in the title bar.
Use the tabs in the dialog box to choose the simulation type (Reflection, Comprehensive, Crosstalk, SSN, or EMI Single).
 - For all types of simulations, choose the Stimulus, Fast/Typical/Slow Mode, and Victim net information.
 - For Comprehensive and Crosstalk simulations, additionally specify Aggressor net information.
2. Click to choose whether or not to *Use Timing Windows* for Crosstalk simulation.
3. Click to choose whether or not to *Save Circuit Files* generated during the simulations.
4. If necessary, use *Preferences* to display the Analysis Preferences dialog box where you can modify simulation parameters.

Simulating and Generating Waveforms

1. Click *Create Waveforms*. The simulator performs one or more simulations and creates the `waveform.sim` files in the case directory.

Viewing Waveforms

1. Click *View Waveform* and choose a `.sim` file in the list box. The waveform is displayed in the SigWave window.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

Creating Standard Reports

Follow these steps to create standard reports:

1. Run `signal probe`.
The Signal Analysis dialog box appears.
2. Display net and pin names in the Nets, Driver Pins, and Load Pins list boxes.
3. Use the *Net:* field to enter a name match pattern to choose nets from the design. -or- Use *List of Nets* to browse for a netlist file. -or- Use *Net Browser* to choose Xnets from the design.
One or more net names appear in the Nets list box.
4. Choose a net to simulate.
5. In the Nets list box, click to choose a net name. -or- Directly in the design window, click on a net to choose it.
The net name highlights in the Nets list box and in the design window. The names of driver pins and load pins on the highlighted net appear in the Driver Pins and Load Pins list boxes.
6. Choose a connection to simulate.
7. In the Driver Pins and Load Pins list boxes, click to choose a connection to simulate.
The chosen pin names highlight in the Driver Pins and Load Pins list boxes and in the design window.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

Creating Custom Reports

Follow these steps to generate custom reports:

1. Run signal probe.
The Signal Analysis dialog box appears.
2. Display net and pin names in the Nets, Driver Pins, and Load Pins list boxes.
3. Use the *Net:* field to enter a name match pattern to choose nets from the design. -or- Use *List of Nets* to browse for a netlist file. -or- Use *Net Browser* to choose Xnets from the design.
One or more net names appear in the Nets list box.
4. Choose a net to simulate.
5. In the Nets list box, click to choose a net name. -or- Directly in the design window, click on a net to choose it.
The net name highlights in the Nets list box and in the design window. The names of driver pins and load pins on the highlighted net appear in the Driver Pins and Load Pins list boxes.
6. Choose a connection to simulate.
7. In the Driver Pins and Load Pins list boxes, click to choose a connection to simulate.
The chosen pin names highlight in the Driver Pins and Load Pins list boxes and in the design window.

Entering Simulation Details, Defining the Custom Report Format, and Simulating and Generating the Custom Report

1. In the Signal Analysis dialog box, click *Reports*.
The Report Generator (case x) dialog box appears. The current case is named in the title bar.
2. Click to choose the Custom Report tab.

Defining the Custom Report Format

The *Report Name:* field displays the current report name. The Simulation Data Table and Setup Data Table areas reflect the established formats for this report. (CustomRpt is the default Custom Report format.)

1. Establish the name and basic format for the report.
2. Use *Clone Selected Report* to copy an existing report format, rename it and add the renamed copy to the list of available reports.
3. Choose a custom report from the *Report Name:* pulldown menu.
4. Click *Clone Selected Report*.
5. In the fill-in, enter the name for the new report.
The new report name appears in the *Report Name:* field and the pulldown menu. The report's format is reflected in the fields of the Simulation Data Table and Setup Data Table areas. - or - Use *New Custom Report* to create a empty custom report, name it, and add it to the list of available reports.
6. Click *New Custom Report*.
7. In the fill-in, enter the name for the new report.
The new report name appears in the *Report Name:* field. and the pulldown menu.
8. From the pulldown menu in the *Sort By:* field, choose the field on which to sort the simulation data.
9. Modify the contents of the Simulation Data Table.
10. For each column field in the Simulation Data Table area, from the pulldown menu, choose the type of simulation data to display in that column. For an empty column, choose Blank.
11. Modify the contents of the Setup Data Table.
12. For each column field in the Simulation Data Table area, from the pulldown menu, choose the setup data to display in that column. For an empty column, choose Blank.

Entering Simulation Details

1. To change to a different case, in the Current Case: field, click to display a list of available cases. Click to choose one. The title bars and Current Case: fields in both the Report Generator (casex) and Signal Analysis [casex] dialog boxes change to reflect the new case.
2. In the Fast/Typical/Slow Mode area, click to choose one or more simulation modes.
3. In the Victim area, choose nets and drivers for simulation.
4. Click to choose whether or not to Use Timing Windows for Crosstalk simulation.
5. Click to choose whether or not to Save Circuit Files generated during the simulations.
6. Click to choose whether or not to Save Waveforms generated during the simulations.
7. If necessary, use Preferences to display the Analysis Preferences dialog box where you can modify simulation parameters.
8. You can click OK to save the Custom Report format as you have defined it without simulating or generating a report.

Simulating and Generating the Report

1. Use *Create Report* to perform the simulations and generate the reports.
The report is created and shown in a text viewer.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

Performing Signal Quality Screening

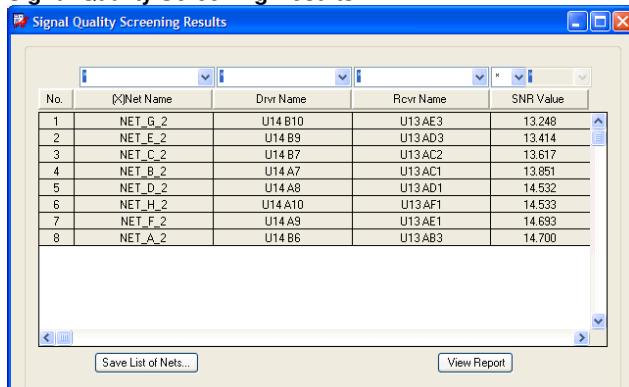
You can perform signal quality screening from the Signal Analysis dialog box in PCB SI by following these steps:

1. Choose *Analyze – Probe* to launch the Signal Analysis dialog box from PCB SI.
2. Select the nets on which signal quality screening is to be performed using one of the following three ways:
 - Select nets in the layout window.
 - Click *List of Nets* to specify an input *.lst* file, which contains a list of nets to be evaluated.
 - Click *Net Browser* to display the Signal Select Browser and select the required nets as shown in
3. Select the required nets and click *OK*.
4. Click the *Waveforms* button and then the *Preferences* button to display the Analysis Preferences dialog box.
5. Adjust the *Pulse Clock Frequency* value, if required.
For Signal quality screening, the buffer is ignored as only an impulse is needed to obtain the frequency response. However, you can choose *Pin or Die* to include package parasitics in the simulation.

 The *Bit Period* for signal quality screening is calculated as follows: $1/\text{Pulse Clock Frequency}$

6. Specify the following on this form:
 - Driver Pin Measurement Location
 - Receiver Pin Measurement Location
7. Click the *InterconnectModels* tab.
You can specify the frequency range for frequency domain simulations here. The *Default Cutoff Frequency* is used for performing signal quality screening.
8. Click *OK*.
9. In the Signal Analysis dialog box, click *Signal Screening*.
The signal quality screening engine kicks off with the selected nets as the input. When the process completes, a results table with the net name and their corresponding SNR values is displayed.

Signal Quality Screening Results



No.	Net Name	Drvr Name	Rcvr Name	SNR Value
1	NET_G_2	U14B10	U13AE3	13.248
2	NET_E_2	U14B9	U13AD3	13.414
3	NET_C_2	U14B7	U13AC2	13.617
4	NET_B_2	U14A7	U13AC1	13.851
5	NET_D_2	U14A8	U13AD1	14.532
6	NET_H_2	U14A10	U13AF1	14.533
7	NET_F_2	U14A9	U13AE1	14.693
8	NET_A_2	U14B6	U13AB3	14.700

The SNR values are sorted from the lowest to the highest. The Signal Quality Screening Results form includes filters over each column, which can help you isolate the violating nets, save the list of nets, modify design, and re-run the signal quality screening process to check if they meet the SNR constraint.

UI Element	Description
Filters (Net Name, Drvr Name, Rcvr Name, SNR Value)	Let you display specific list of nets based on the net name, driver or receiver name, or the SNR values (based on an absolute number up to 2 places of decimal).
Save List of Nets	Lets you save the list of nets for future loading of the netlist into the probe dialog for signal quality screening.

S Commands

S Commands--signal probe

View Report	Lets you view and save a text version of the report.
-------------	--

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

Starting SigXplorer from the Simulator

You can start SigXplorer from the simulator's Signal Analysis dialog box and use the SigXplorer topology canvas to experiment with termination, alternate topology and what-if transmission line values.

1. Run `signal probe`.
The Signal Analysis dialog box appears.
2. Display net and pin names in the Nets, Driver Pins, and Load Pins list boxes.
3. Use the *Net:* field to enter a name match pattern to choose nets from the design. -or- Use *List of Nets* to browse for a netlist file. -or- Use *Net Browser* to choose Xnets from the design.
One or more net names appear in the Nets list box.
4. Choose a net to simulate.
5. In the Nets list box, click to choose a net name. -or- Directly in the design window, click on a net to choose it.
The net name highlights in the Nets list box and in the design window. The names of driver pins and load pins on the highlighted net appear in the Driver Pins and Load Pins list boxes.
6. Choose a connection to simulate.
7. In the Driver Pins and Load Pins list boxes, click to choose a connection to simulate.
The chosen pin names highlight in the Driver Pins and Load Pins list boxes and in the design window.
8. Click View Circuit.
The SigXplorer topology canvas appears. In some cases you are prompted to choose which version of SigXplorer to start:
 - Allegro PCB SI GXL
 - Allegro PCB SI XL

Simulating from SigXplorer

1. With your topology displayed in the topology canvas, use *Analyze – Preferences* to specify how the simulation will perform.
The Analysis Preferences dialog box appears.
2. In the Analysis Preferences dialog box, specify parameter values for Pulse Stimulus, Simulation, FTS modes, Measurement modes, Buffer Delay Selection, and EMI simulation.
3. Use *Analyze – Simulate* to start the simulation.
Simulation begins and messages display in the Command tab of the spreadsheet. Simulation results display in the Results spreadsheet tab and in the SigWave window.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

signal init

The `signal init` command displays the Signal Analysis Initialization dialog box for managing the system setup and managing simulation cases.

 This command is not available in L Series products.

Related Topics

- [System Configuration Editor](#)
- [Connect By Component Dialog Box](#)
- [Creating a New Case](#)
- [Editing a Case Description](#)
- [Removing a Case](#)
- [Choosing a System Configuration](#)
- [Choosing the Current Case](#)

Signal Analysis Initialization Dialog Boxes

Access Using

- Menu Path: *Analyze – Initialize*

Use this dialog box to perform the following setup tasks:

- Create a new system configuration.
- Modify an existing system configuration.
- Add, edit, or delete cases

System Configuration Setup Area

System Configuration	Displays the existing system configurations that have been defined for the current design. A system configuration file (.scf) is a database representation of all the participating designs (including interconnecting cables and connectors) that comprise the system. Note: The system configuration also includes the Xnets and pin-pairs that traverse a system as well as their assigned constraint values.
New Design Link	Displays the System Configuration Editor dialog box to create a new system configuration.
Edit Design Link	Displays the System Configuration Editor to modify the currently active system configuration.  Refer to the "multi-board designs" chapter of your product documentation for information on working with system configurations (design links).
Browse	Displays a File Browser to search for system configuration (.scf) files.

Case Setup Area

Use this area to manage the Signal Analysis cases created in the analysis directory, and to edit their text descriptions.

Current Case	Displays the name of the current case under analysis.
Case list box	Lists available cases with descriptions. The current case is highlighted.
Ask about case updates...	Indicates whether you want to be notified about case updates whenever the project changes.
New Case	Creates a new case.
Set Desc	Adds or edits a case description.
Remove Case	Deletes a chosen case.

 By default, *Always ask me about case updates when the project changes* is unchecked, and the *Keep the current case, clearing simulation data* executes in the background. To change this behavior, choose *Analyze – Initialize* and check *Always ask me about case updates when the project changes*. Subsequent parameter changes and simulations will then invoke the *Case Update* dialog box, where you can change the case management settings.

 You can also access the *Case Update* dialog box by choosing *Analyze – Probe* and clicking *Reports* or *Waveforms*.

Related Topics

- [Connect By Component Dialog Box](#)
- [Creating a New Case](#)
- [Editing a Case Description](#)
- [Removing a Case](#)
- [Choosing a System Configuration](#)
- [Choosing the Current Case](#)

System Configuration Editor

Use this dialog box to create a new system configuration or to modify the active system configuration.

Drawings	
<i>Add File</i>	Add a board file (design) to the system configuration
<i>Add BoardModel</i>	Add a BoardModel to the system configuration (a design can be represented in the abstract as an electrical BoardModel)
<i>Remove</i>	Remove a design from the system configuration
<i>Set Design Name</i>	Add a label to the chosen design name
<i>Set Drawing Path</i>	Specify a path to the chosen design name. You can, in effect, instantiate the same design many times with reference to a single board file (.brd). You give each derivative a unique label. For example, a memory module of the same design may be instantiated in a system.
Connections Field	
<i>Add</i>	Name a connection between participating designs in the system configuration.
<i>Remove</i>	Remove a connection between participating designs in the system configuration.
<i>Copy</i>	Clone an existing connection between participating designs in the system configuration.
<i>Set Length</i>	Specify a cable length in meters. Use zero for plug-in boards.
<i>Set Cable Model</i>	Specify a cable model from the library.
<i>System Xnets Name From Design</i>	Choose from the drop-down list a design in the system configuration from which the system Xnet will be named. See <i>How to Control Xnet Naming</i> in your product documentation for details on naming board and system-level Xnets.
Connections PinMap Fields	
<i>Add Wires</i>	Add new pin connections to the chosen cable connection. Presents a series of dialog boxes to collect information on the first wire number, the number of wires, and the starting pin (at either end of the connections).
<i>Remove Wires</i>	Remove the chosen pin connections from the chosen cable connection.
<i>Connect by Component</i>	Select two components on specific designs to form the pin-to-pin connections. A pin-to-pin connection is established between each pin on one component to each pin on the other component with the same pin number.
<i>TextEdit PinMap</i>	Opens the entire pin connection map in a text editor for further manipulation.
<i>Set From Pins</i>	Presents a dialog box for you to select the starting pin at one end of the cable connection
<i>Set to Pins</i>	Presents a dialog box for you to edit the starting pin at the other end of cable connection

Related Topics

- [signal init](#)
- [Creating a New Case](#)
- [Editing a Case Description](#)
- [Removing a Case](#)
- [Choosing a System Configuration](#)
- [Choosing the Current Case](#)

Connect By Component Dialog Box

You access this dialog box from the System Configuration Editor (*Connect by Component* button) to form the pin-to-pin connections across design links. A pin-to-pin connection is established between each pin on one component to each pin on the other component with the same pin number.

From/To	Displays in drop-down menus the design link names and components in the drawing.
----------------	--

Related Topics

- [signal init](#)
- [Signal Analysis Initialization Dialog Boxes](#)
- [Editing a Case Description](#)
- [Removing a Case](#)
- [Choosing a System Configuration](#)
- [Choosing the Current Case](#)

Creating a New Case

To create a new case, follow these steps:

1. Display the Signal Analysis Initialization dialog box in Allegro SI or the layout editor by choosing *Analyze – Initialize*.
- or -
SigXplorer by choosing *SigNoise – Initialize*.
2. Check *Always ask me about case updates when the project changes*.
3. Change a parameter or simulate.
4. In the Case list box, click to choose a case to use as the basis for the new case.
The chosen case is highlighted in the list box.
5. Click *New Case*.
A new case is created with the next available case number and added to the list box. The new case becomes the current case. Its name is highlighted in the list box and listed in the Current Case: field.
6. Click Set Desc to edit the text description associated with the new case.
7. Click *OK*.

The simulator creates a case directory for the new case. Setup data for the new case duplicates the data for the case upon which the new case was based. The new case becomes the current case and other simulation forms are changed to reflect the new current case.

Related Topics

- [signal init](#)
- [Signal Analysis Initialization Dialog Boxes](#)
- [System Configuration Editor](#)
- [Removing a Case](#)
- [Choosing a System Configuration](#)
- [Choosing the Current Case](#)

Editing a Case Description

Perform these steps to edit a case description:

1. Display the Signal Analysis Initialization dialog box in Allegro SI or the layout editor by choosing *Analyze – Initialize*.
- or -
SigXplorer by choosing *SigNoise – Initialize*.
2. Click *Set Desc* and enter the text description.
3. Click *OK*.

The text description appears with the case name in the case list box and on simulation reports.

Related Topics

- [signal init](#)
- [Signal Analysis Initialization Dialog Boxes](#)
- [System Configuration Editor](#)
- [Connect By Component Dialog Box](#)
- [Choosing a System Configuration](#)
- [Choosing the Current Case](#)

Removing a Case

You can remove a case by following these steps:

1. Display the Signal Analysis Initialization dialog box in Allegro SI or the layout editor by choosing *Analyze – Initialize*.
- or -
SigXplorer by choosing *SigNoise – Initialize*.
2. Check *Always ask me about case updates when the project changes*.
3. Change a parameter or simulate.
4. In the Case list box, click to choose the case to remove.
The chosen case is highlighted in the list box.
5. Click *Remove Case*.
The case name and description are deleted from the list box.
6. Click *OK*.
All analysis directory files associated with the case are deleted. The most recent case becomes the current case.

Related Topics

- [signal init](#)
- [Signal Analysis Initialization Dialog Boxes](#)
- [System Configuration Editor](#)
- [Connect By Component Dialog Box](#)
- [Creating a New Case](#)
- [Choosing the Current Case](#)

Choosing a System Configuration

Follow these steps to choose a system configuration:

1. Choose *Analyze – Initialize*.
The Signal Analysis Initialization dialog box displays.
2. In the System Configuration field, choose an appropriate System Configuration from the drop-down menu.
3. Click *OK*.

Related Topics

- [signal init](#)
- [Signal Analysis Initialization Dialog Boxes](#)
- [System Configuration Editor](#)
- [Connect By Component Dialog Box](#)
- [Creating a New Case](#)
- [Editing a Case Description](#)

Choosing the Current Case

Perform these steps to choose a current case:

1. Display the Signal Analysis Initialization dialog box in Allegro SI or the layout editor by choosing *Analyze – Initialize*.
- or -
SigXplorer by choosing *SigNoise – Initialize*.
2. In the Case list box, click to choose a case.
The chosen case is highlighted in the list box and its name appears above the list box in the Current Case field.
3. Click *OK*.
Other simulation forms are changed to reflect the new current case.

Related Topics

- [signal init](#)
- [Signal Analysis Initialization Dialog Boxes](#)
- [System Configuration Editor](#)
- [Connect By Component Dialog Box](#)
- [Creating a New Case](#)
- [Editing a Case Description](#)
- [Removing a Case](#)

signal lib audit

The `signal lib audit` command opens a file browser from which you can access a design model library file. When you choose a file, the `dmlcheck` utility verifies its formatting.

 This command is not available in L Series products.

signal library

The `signal library` command displays the Signal Analysis Library Browser.

Use the Signal Analysis Library Browser for specifying the device and interconnect libraries used by the simulator during signal analysis. These libraries contain the device and interconnect models used by the simulator to build circuit simulations.

Other associated dialog boxes launched via the Signal Analysis Library Browser enable you to create and edit the device and interconnect models contained in these libraries.

 This command is not available in L Series products.

Related Topics

- [Signal Library Tasks](#)

Signal Analysis Library Dialog Boxes

Access Using

- Menu Path: *Analyze – Model Browser*



- Toolbar Icon:

SI Model Browser	DML Library Management Dialog Box	Set Model Search Path Dialog Box
Analog Output Model Editor Dialog Box	IBIS Device Model Editor Dialog Box	IBIS Device Pin Data Dialog Box
Buffer Delays Dialog Box	IOCell Editor Dialog Box	V/I Curve Editor Dialog Box
V/T Curve Editor Dialog Box	Set V/I Curve Point Dialog Box	

SI Model Browser

Using SI Model Browser (and its associated dialog boxes) you can perform the following basic model development tasks:

- List the models in a library.
- Create a device model with default values or clone an existing device model and add the newly created model to the working library.
- Delete a model from the working library.
- Translate a model.

The SI Model Browser's tabbed interface accommodates the model type that you want to translate, be it IBIS, Spectre, Spice, IML, DML, or HSPICE. You need to select the appropriate tab, click the model, and click the *Translate* button to translate it. From these tabs, you can also edit a model directly in its native format. Once translated, these models also appear under the DML tab.

Each tab contains a field for filtering the listed models, as well as a button to set the model's library search path and to set its associated file extensions (Set Model Search Path dialog box).

You can filter fields at the top of the SI Model Browser control which models are displayed in the Model Browser list box. You can specify which models are listed in the model search list by library, by model type, or by characters in the model name.

Displaying a List of Models

Model List Options

Option	Display Shows ...	Function
<i>Library Filter</i>	Currently selected device or model library.	Changes the current device or library (click arrow).
<i>Model Type Filter</i>	Current model filter setting.	Changes the model filter to display only models of a particular type (click arrow).
<i>Model Name Pattern</i>	Current model name pattern setting.	Changes the model name pattern string to display only models whose name is included in the specified character string (edit type-in box).  Use * for wildcard selection.

Creating Models and Adding them to a Working Library

You can add a device or interconnect model to the working device or interconnect model library in either of two ways:

- By copying (or cloning) an existing model.
- By creating a new model with default values.

You must first create a device model and add it to the working library before you can edit it to characterize a particular device.

Create / Add Model Buttons

Button	Function
<i>Add-></i>	<p>Displays the <i>Add Model</i> pop-up menu and enables you to choose a device and interconnect model type to add to your working device or interconnect library.</p> <p>⚠ Menu options vary according to the library type selected.</p> <p>The following menu option is common when either a device or interconnect library is selected.</p>
<i>CloneSelection</i>	Copies or clones the model that you select in the SI Model Browser list, prompts you to name the copy, and adds the renamed copy to the working library.
<i>Delete</i>	Deletes the selected model.
<i>Edit</i>	Displays a text editor or a model editor, depending on the type of model you select in the SI Model Browser search list.
<i>Select</i>	Selects a model.
<i>Model Editor</i>	Opens the selected model in Model Editor. Model Editor assists in reviewing and validating models that you create or edit. For more information on Model Editor, see the <i>Working with Model Editor</i> chapter in <i>Allegro SI SigXplorer User Guide</i> .
<i>Set Search Path</i>	Launches the Set Model Search Path Dialog Box dialog box.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

DML Library Management Dialog Box

You use the [DML Library Management](#) dialog box to create and manage your libraries of device and interconnect models, and launch Model Editor. You can also use it to specify which device and interconnect libraries you want SigXplorer to access, as well as the order of library access (in the Set Model Search Path dialog box).

Libraries are searched starting at the top of the list. If a model is included in two or more libraries, you can use the search order (in the Set Model Search Path dialog box) to determine which library the simulator searches first. The simulator uses the first model found.

You can also set a particular library as the working library. A working library is the only library to which the simulator can add models. If you want to add to a library that is not the working library, you must make it the working library before you start the process of adding the model. You can have at most two working libraries: one working device model library and one working interconnect model library.

Option	Function
<i>Working Library</i>	Sets the working (or active) library for device models. (Before you edit or add new models, make the target library the working library.)
<i>Ignore Library</i>	Ignores the library during search.
<i>Select for Merge/Index</i>	Select the libraries for merging or indexing.
<i>Create New Lib</i>	Displays a file browser where you can specify the device/interconnect library (.dml or .iml) to be created and added to the device library search list.
<i>Check Lib</i>	Runs the <code>dmlcheck</code> utility on the selected model and displays the result in a log file.
<i>Merge Libs</i>	Merges all .dml files present in the library list into one .dml file.
<i>Make Lib Index</i>	Creates an index file for all of the .dml files present in the library list. (The working library is excluded.)
<i>Set Search Path</i>	Opens the Set Model Search Path dialog box.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Set Model Search Path Dialog Box

Use the [Set Model Search Path](#) dialog box to specify the directories in which to search for signal models, and their search order.

Option	Function
<i>Add Directory</i>	Adds a directory containing device library or device index to the device library search list.
<i>Move To Top</i>	Raises the selected device library to the topmost position in the device library search list.
<i>Move Up</i>	Raises the selected device library one position up in the device library search list.
<i>Move Down</i>	Lowers the selected device library one position down in the device library search list.
<i>Move To Bottom</i>	Lowers the selected device library to the bottom position in the device library search list.
<i>Remove Library</i>	Removes selected device libraries from the device library search list.
<i>Reset To Default</i>	Resets the library list to default as specified by the <code>SI_MODEL_PATH</code> directive in the <code>cds.cpm</code> file.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Analog Output Model Editor Dialog Box

Option	Function
<i>Model</i>	Displays the name of the Analog Output model.
<i>Series Resistance</i>	Displays the resistance value for a series resistor.
<i>Rise</i>	Displays the path to an Analog Workbench file or displays a file browser.
<i>Fall</i>	Displays the path to an Analog Workbench file or displays a file browser.
<i>Pulse</i>	Displays the path to an Analog Workbench file or displays a file browser.
<i>Inv Pulse</i>	Displays the path to an Analog Workbench file or displays a file browser.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

IBIS Device Model Editor Dialog Box

The IBIS Device Model Editor dialog box contains three tabs that you can use to perform the following tasks.

- Edit information for the pins associated with the IBIS device model.
- Group power and ground pins and assign them to power and ground buses.
- Group signal pins and assign IOCell models and IOCell supply buses.

Edit Pins Tab

Model Info Area

Option	Function
<i>Model Name</i>	Name of the IBIS device model.
<i>Manufacturer</i>	Name of the model manufacturer (not used by SigNoise).
<i>Package Model</i>	Name of a package model associated with the IBIS device model.

Estimated Pin Parasitics Area

Option	Function
<i>Resistance</i>	Minimum, typical, and maximum values for resistance.
<i>Capacitance</i>	Minimum, typical, and maximum values for capacitance.
<i>Inductance</i>	Minimum, typical, and maximum values for inductance.

IBIS Pin Data Area.

Option	Function
<i>Pin</i>	The pin number.
<i>Signal</i>	The signal associated with the pin.
<i>IOCell</i>	The associated IOCell model.
<i>Resistance</i>	The resistance, if you are using individual pin parasitics.
<i>Capacitance</i>	The capacitance, if you are using individual pin parasitics.
<i>Inductance</i>	The inductance, if you are using individual pin parasitics.
<i>DiffPair Mate</i>	The inverse pin, if the pin is part of a differential pair.
<i>Wire</i>	The wire number, which determines which wire of the PackageModel is used for this pin.

Edit Pins Buttons

Button	Function
Add Pin Data	Prompts for the name of a new pin to add, and displays the IBIS Device Pin Data dialog box to add or modify data including buffer delays for a new pin.
Measure Delays	Measures buffer delays by simulating each pin with the proper test load. On pins with a Model Selector assigned, buffer delays are simulated for each selectable IOCELL. If you use a Package Model, you must perform simulations for each driver pin. Otherwise, pins with identical parasitics and IOCELL assignments will share simulation data. A progress meter displays the status of the process for buffer delay simulations, especially for complex parts. You can click the Stop button to cancel the simulation. Options: <i>Unmeasured Drivers</i> - creates data for drivers not previously processed. <i>All Drivers</i> - creates data for all drivers, refreshes previously processed data. <i>Clear All Delays</i> - deletes all buffer and differential pair delays from the model.
Set WireNumbers	Sets the wire number for each pin based on a sort criteria. Options: <i>Order by Pin Name</i> - Pins are sorted by pin name. Wire numbers are assigned numerically starting at one. Alphabetic and numeric portions of names are separately considered so that, for example, <i>A2</i> appears before <i>A10</i> and <i>B6</i> . <i>Order by IOCell Name</i> - Pins are sorted first by IOCell model name. Second, pins with the same IOCell model assigned are sorted by pin name. Wire numbers are then assigned numerically starting at one. ⚠ After the wire numbers have been set, the pin list is displayed in wire number order.
DML Check	Runs the <code>dmlcheck</code> utility on the model being edited and displays the result in a text window.
OK	Runs <code>dmlcheck</code> if changes to the model are made. Otherwise, choosing OK closes the window.

Assign Power/Ground Pins Tab

All Pins Area

Option	Function
Pin #	The pin number.
IOCell	Any IOCell model currently assigned to the pin.
Pwr Bus	Any power bus currently assigned to the pin. This column is blank for a pin that is not currently assigned to a power bus.
Gnd Bus	Any ground bus currently assigned to the pin. This column is blank for a pin that is not currently assigned to a ground bus.
Pinuse	The pin use code. UNSPEC indicates that no pin use is assigned.
Net Name	The name of any net connected to the pin. This column is blank when you are editing a model selected from a library. This column displays a net name when you are editing a model associated with a device that exists in the active design.
Nets Shown for Component field	The RefDes for the device associated with the model being edited. This occurs when you invoke the IBIS Device Model Editor for a specific instance of a device selected in the Signal Model Assignment dialog box.
Sort By (column buttons)	Selects one of the columns on which to sort the data.
Filters (dropdown menus)	Filters the information displayed in the column. Initially the field contains an asterisk (*) so that all data is displayed. The <i>Power Bus</i> menu lists existing power buses. The <i>Ground Bus</i> menu lists existing ground buses. The <i>Pinuse</i> menu lists existing pin use codes.

All Pins Area Buttons

Button	Function
Select All	Selects all pins currently displayed in the <i>All Pins</i> list box and re-displays them in the <i>Selected Pins</i> list box.
Deselect All	Deselects all pins currently selected and clears the <i>All Pins</i> list box.
Deselect One	Deselects one pin in the <i>All Pins</i> list box.

Select Pins Area

Option	Function
Assign/De-assign buttons	Assigns or de-assigns the group of pins listed in the <i>Selected Pins</i> list box to the IOCell model, power bus, or ground bus named in the associated field.
Pin #	The pin number for pins selected from the <i>All Pins</i> list box.
IOCell	The IOCell model assigned for pins selected from the <i>All Pins</i> list box.
Pwr Bus	The power bus name for pins selected from the <i>All Pins</i> list box.
Gnd Bus	The ground bus name for pins selected from the <i>All Pins</i> list box.
IOCell Browse (field and button)	Enter the IOCell model name to be assigned to the group of pin or click Browse to display the Model Browser and select an IOCell model there.
Pwr Bus (field and menu)	Select an existing power bus name from the menu.
Gnd Bus (field and menu)	Select an existing ground bus name from the pull down menu.

Select Pins Area Buttons

Button	Function
DML Check	Runs the <code>dmlcheck</code> utility on the model being edited and displays the result in a text window.
OK	Closes the window without running <code>dmlcheck</code> .

Assign Signal Pins Tab

All Pins Area

Option	Function
<i>Pin #</i>	The pin number.
<i>IOCell</i>	Any IOCell model currently assigned to the pin.
<i>Pwr Bus</i>	Any power bus currently assigned to the pin. This column is blank for a pin that is not currently assigned to a power bus.
<i>Gnd Bus</i>	Any ground bus currently assigned to the pin. This column is blank for a pin that is not currently assigned to a ground bus.
<i>Pinuse</i>	The pin use code. UNSPEC indicates that no pin use is assigned.
<i>Net Name</i>	The name of any net connected to the pin. This column is blank when you are editing a model selected from a library. This column displays a net name when you are editing a model associated with a device that exists in the active design.
<i>Nets Shown for Component field</i>	The RefDes for the device associated with the model being edited. This occurs when you invoke the IBIS Device Model Editor for a specific instance of a device selected in the Signal Model Assignment dialog box.
<i>Sort By (column buttons)</i>	Selects one of the columns on which to sort the data.
<i>Filters (dropdown menus)</i>	Filters the information displayed in the column. Initially the field contains an asterisk (*) so that all data is displayed. The <i>Power Bus</i> menu lists existing power buses. The <i>Ground Bus</i> menu lists existing ground buses. The <i>Pinuse</i> menu lists existing pin use codes.

All Pins Area Buttons

Button	Function
<i>Select All</i>	Selects all pins currently displayed in the <i>All Pins</i> list box and re-displays them in the <i>Selected Pins</i> list box.
<i>Deselect All</i>	Deselects all pins currently selected and clears the <i>All Pins</i> list box.
<i>Deselect One</i>	Deselects one pin in the <i>All Pins</i> list box.

Select Pins Area

Option	Function
Assign/De-assign buttons	Assigns or de-assigns the group of pins listed in the <i>Selected Pins</i> list box to the IOCell model, power bus, or ground bus named in the associated field.
Pin #	The pin number for pins selected from the <i>All Pins</i> list box.
IOCell	The IOCell model assigned for pins selected from the <i>All Pins</i> list box.
Pwr Bus	The power bus name for pins selected from the <i>All Pins</i> list box.
Gnd Bus	The ground bus name for pins selected from the <i>All Pins</i> list box.
IOCell Browse (field and button)	Enter the IOCell model name to be assigned to the group of pin or click Browse to display the Model Browser and select an IOCell model there.
Pwr Bus (field and menu)	Select an existing power bus name from the menu.
Gnd Bus (field and menu)	Select an existing ground bus name from the pull down menu.

Select Pins Area Buttons

Button	Function
DML Check	Runs the <code>dmlcheck</code> utility on the model being edited and displays the result in a text window.
OK	Closes the window without running <code>dmlcheck</code> .

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

IBIS Device Pin Data Dialog Box

From the IBIS Device Model Editor, you can display the IBIS Device Pin Data dialog box to:

- Add or edit data (including individual pin parasitics) for the pins in the IBIS device model.
- Add or edit buffer delay information for the pins in the IBIS device model.

IBIS Pin Map Area

Option	Function
<i>Pin</i>	The pin whose data is displayed.
<i>Signal</i>	The signal associated with the pin. Pins with an NC signal are not connected. You can ignore these pins in the IBIS Device Model Editor.
<i>Resistance Capacitance Inductance</i>	The Individual Pin Parasitic values for the pin (if you are not using a package model).
<i>Wire Number</i>	The wire number for the pin. (This can be the same as the pin number if numeric.) Wire numbers specify the wire numbers for the package model and are used only for IBIS device models that have a package model.
<i>IOCell</i>	The IOCell model associated with the pin. Pins with an NC model are not connected. You can ignore these pins in the IBIS Device Model Editor. If you want to view the voltage versus current (V/I) curves for an IOCell model before you assign it to a pin as part of the IBIS device model, open the model in the IOCell Editor and use the View VI button. <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;">  Whenever you have made changes to the IOCell models for a device, regenerate the buffer delay values for the device using All Drivers mode. </div>
<i>Power Bus</i>	Name of the power bus.
<i>Power Clamp Bus</i>	Name of the power clamp bus.
<i>Ground Bus</i>	Name of the ground bus
<i>Ground Clamp Bus</i>	Name of the ground clamp bus.

Diff Pair Data Area

Option	Function
<i>Type</i>	Identifies the pin listed in the <i>Pin</i> field as the Inverting or Non-inverting pin of the differential pair. When the <i>Type</i> field displays <i>None</i> , the pin identified in the <i>Pin</i> field is not part of a differential pair.
<i>Mate Pin</i>	The name of the differential pair mate pin to the pin identified in the <i>Pin</i> field.
<i>Launch Delay</i>	Minimum, typical and maximum launch delay values for the pin, if it is an Output or IO pin.
<i>Input High Input Low Output High Output Low</i>	Minimum, typical and maximum differential logic threshold values.

IBIS Device Pin Data Buttons

Button	Function
<i>Buffer Delays</i>	Displays the Buffer Delays dialog box that enables you to change the buffer delay information for a pin. This dialog box contains the data that SigNoise uses to calculate buffer delay values for rising and falling drivers (output buffers).

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Buffer Delays Dialog Box

Option	Function
<i>IOCell Test Fixture: Resistor Capacitor Term Voltage Ref Voltage</i>	Displays values entered in the Delay Measurement tab of the IOCell Editor for the associated IOCell model.
<i>Rise Delay</i>	Fast, Typical, and Slow values for rise delay measured from IOCell delay measurement information. Use these fields to edit output buffer delay values directly.
<i>Fall Delay</i>	Fast, Typical, and Slow values for fall delay measured from IOCell delay measurement information. Use these fields to edit output buffer delay values directly.

Button	Function
<i>Edit IOCell</i>	Starts IOCell Editor for the associated IOCell model where you can edit test fixture values (resistor, capacitor, term voltage, and ref voltage) and other IOCell model information. Test fixture values specify the loading conditions under which SigNoise measures buffer delays. Test fixture values are associated with the IOCell model for a pin rather than with the pin itself.
<i>Measure Delays</i>	Refreshes the delay values if the IOCell model information has changed: slow, typical, and fast buffer delay values for rising and falling drivers. SigNoise performs the buffer delay measurements in All Drivers mode. (You can also measure buffer delays from the IBIS Device Model Editor.) Whenever any IOCell model data changes, it is important to recalculate buffer delay values in All Drivers mode using either the <i>Measure Delays</i> button or the <i>Measure Delays–All Drivers</i> button in the IBIS Device Model Editor

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

IOCell Editor Dialog Box

Common Buttons

Button	Function
<i>View VI</i>	Displays the minimum, typical, or maximum VI curve.

General Tab

Option	Function
<i>Name</i>	Displays the name of the model.
<i>Type</i>	Displays the type of model.
<i>Technology</i>	Displays a pop-up menu of technologies. Choices are CMOS, TTL, and ECL.
<i>Die Capacitance</i>	Displays minimum, typical, and maximum values for die capacitance.
<i>Reference Temperature</i>	Displays minimum, typical, and maximum reference temperatures.

Button	Function
<i>Power Clamp</i>	Starts the V/I Curve Editor for PowerClamp
<i>Ground Clamp</i>	Starts the V/I Curve Editor for GroundClamp

Input Section Tab

Option	Function
<i>Logic Thresholds</i>	Displays minimum, typical, and maximum values for high and low input thresholds.
<i>View VI</i>	Displays the minimum, typical, or maximum VI curve.

Output Section Tab

Option	Function
<i>Ramp (20%/80%)</i>	Displays minimum, typical, and maximum dV and dT values for rising and falling slew rates.

Button	Function
<i>PullUp</i>	Invokes the VI curve editor to examine PullUp VI curves.
<i>PullDown</i>	Invokes the VI curve editor to examine PullUp VI curves.
<i>Rise Wave</i>	Invokes the VT curve editor to examine Rise Wave VT curves
<i>Fall Wave</i>	Invokes the VT curve editor to examine Fall Wave VT curves

Delay Measurement Tab

Option	Function
<i>Test Fixture -- Resistor, Capacitor, and Termination Voltage</i>	Displays test fixture values for resistance, capacitance, and termination voltage.
<i>V Measure</i>	Displays the reference voltage.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

V/I Curve Editor Dialog Box

Option	Function
<i>Reference Voltage</i>	Displays the minimum, typical, and maximum reference voltages.
<i>V/I Convention</i>	Specifies either IBIS or Databook format.
<i>Voltage</i>	The voltage of the curve point.
<i>Min I</i>	The minimum tolerance of the curve point in mA.
<i>Typ I</i>	The typical tolerance of the curve point in mA.
<i>Max I</i>	The maximum tolerance of the curve point in mA.

Button	Function
<i>Add</i>	Adds, modifies, or deletes a curve point. (Displays the Set V/I Curve Point dialog box.)
<i>View</i>	Displays the minimum, typical, and maximum curves in the SigWave window.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

V/T Curve Editor Dialog Box

Option	Function
<i>Test Package (R, L, C)</i>	Resistance, delay, and capacitance for the test package.
<i>V/T Curve Test Fixture (R, L, C) and (Vmin, Vtyp, Vmax)</i>	Resistance, delay, and capacitance for the test fixture.

Button	Function
<i>View</i>	Display the curve in the SigWave window.
<i>Import</i>	Imports the .wave file.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Set V/I Curve Point Dialog Box

Option	Function
<i>Voltage</i>	The voltage of the curve point.
<i>Min I</i>	The minimum tolerance of the curve point in mA.
<i>Typ I</i>	The typical tolerance of the curve point in mA.
<i>Max I</i>	The maximum tolerance of the curve point in mA.
<i>Delete</i>	Deletes the selected line from the VI Curve Editor.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Signal Library Tasks

Working with Models and Libraries	Specifying a Working Device Model Library/Interconnect Model Library	Adding a Device Library or Index
Adding a Standard Cadence Library	Deleting a Library From The Search List	Creating a Device Model Index
Creating a Device Model Index from the Operating System Command Line	Reordering the Libraries in the Search List	Merging Device Model Libraries
Translating Other Device Model Library Formats to DML	Working with Device Models	Editing an Analog Output Model
Creating a Cable Model	Editing a Cable Model	Creating a DesignLink Model
Editing a DesignLink Model	Creating an ESpice Device Model	Editing an ESpice Device Model
Creating an IBIS Device Model	Editing an IBIS Device Model	Adding a Pin to an IBIS Device Model
Editing the Pin Data for an Existing Pin on an IBIS Device Model	Creating an IOCell Model	Editing an IOCell Model
Creating a Package Model by Copying and Editing an Existing Model	Editing a Package Model	Adding a Package Model to an IBIS Device Model
Working with Interconnect Models	Editing a Trace, MultiTrace, Pin or Shape Model	

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)

Working with Models and Libraries

You can perform the following tasks with the `signal library` command working with models and libraries:

- Specifying a Working Device Model Library/Interconnect Model Library
- Adding a Device Library or Index
- Adding a Standard Cadence Library
- Deleting a Library From The Search List
- Creating a Device Model Index
- Creating a Device Model Index from the Operating System Command Line
- Reordering the Libraries in the Search List
- Merging Device Model Libraries

- [Translating Other Device Model Library Formats to DML](#)

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Specifying a Working Device Model Library/interconnect Model Library

To specify a working device model library, follow these steps:

1. Choose Analyze – Model Browser.
The SI Model Browser dialog box appears.
2. Click the *Library Management* button.
The DML Library Management dialog opens.
3. In the *DML Libraries* list, click the *Working Library* check box next to the library, which you want to designate as the working device model library.

 For IML Libraries, select the library in the *IML Libraries* list (bottom pane of the window) instead of DML Libraries.

4. Click the *Set Search Path* button.
The Set Model Search Path dialog appears. The library file name you designated as the working library appears in the *Directories To Be Searched for Model Files* list. You can change the search order of libraries in this dialog box.
5. Click *OK*.
6. Click *OK*.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Adding a Device Library or Index

To add a device library or an index, follow these steps:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Click *Set Search Path*.
3. In the Set Model Search Path dialog box, click *Add Directory* and browse to the location where the desired library or index files are present.
4. Click *OK*.
5. Use the *Move To Top*, *Move Up*, *Move Down*, or *Move To Bottom* buttons to set the search priority.
6. Click *OK*.

The directory containing libraries or index files is added to the *Directories To Be Searched for Model Files* search list. If the library is not in the working directory, the full path to the library is displayed in the list box.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Adding a Standard Cadence Library

You can add a standard Cadence library by following these steps:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Click *Set Search Path*.
3. In the Set Model Search Path dialog box, click *Add Directory* and browse to the location where one of the following libraries is present:
 - `cds_models.ndx` (A small sample model library.)
 - `cds_partlib.ndx` (The standard Cadence parts library.)
4. Click *OK*.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Deleting a Library From The Search List

To delete a library from the search list, perform these steps:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. In either the *Device Library Files* list or the *Interconnect Library Files* list, select the library you want to delete.
3. Click *Remove Library*.
The selected library is deleted from the search list.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Creating a Device Model Index

To create a device model index, follow these steps:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Click *Library Management*.
3. Click the *Select for Merge/Index* check boxes next to the .dml files for which you want to create an index.
4. Click *Make Lib Index*.
The Save As dialog box appears.
5. Enter a name for the new index file and click *Save*.
6. Click *Yes* in the message box stating that the selected files be included in the index.
7. Click *OK*.

⚠ Index files (.ndx) are read-only. For this reason, you cannot include a .dml file that is designated as the working library, since the simulator automatically saves edits to the working library file.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Creating a Device Model Index from the Operating System Command Line

1. Use the `mkdeviceindex` utility from the operating system command line to create a library index for one or more device model library files.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Reordering the Libraries in the Search List

You can reorder the libraries in the search list by following these steps:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Click *Set Search Path*.
3. In the Set Model Search Path dialog box, select a library and use the *Move To Top*, *Move Up*, *Move Down*, or *Move To Bottom* buttons to reorder the libraries in the search list.
4. Click *OK*.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Merging Device Model Libraries

Follow these steps to merge device model libraries:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Click *Library Management*.
3. Click the *Select for Merge/Index* check boxes next to the .dml files which you want to merge.
4. Click *Merge Libs*.
The Save As dialog box appears.

 All of the .dml files shown in the *Device Library Files* search list will be merged together. Files with extensions other than .dml are ignored.

1. Enter a name for the new merged file and click *Save*.
2. Click *Yes* in the message box stating that the selected files be merged.
3. Click *OK*.

The new merged file replaces all of the .dml files previously listed in the search list.

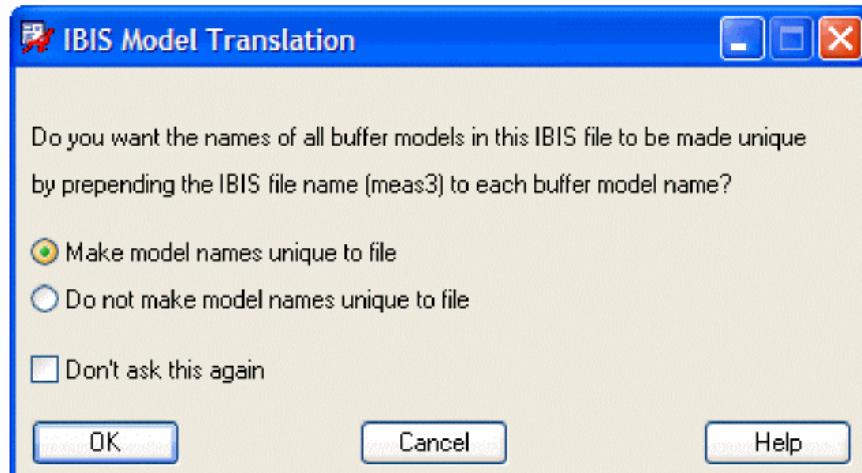
Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Translating Other Device Model Library Formats to DML

The SI Model Browser's tabbed interface accommodates the model type that you want to translate to a .dml format, be it IBIS, Spectre, Spice, IML, DML, or HSPICE. You need to select the appropriate tab, click the model, and click the *Translate* button to translate it. From these tabs, you can also edit a model directly in its native format. Once translated, these models also appear under the DML tab.

1. Choose Analyze – Model Browser.
The SI Model Browser dialog box appears.
2. Select the model to be translated.
3. Click *Translate*.



4. Choose whether you want to make model names unique to the file.
5. Click *OK*.
The selected file is translated into the specified .dml file. The new .dml file is added to the search list.
Any warnings or error messages that are generated during the translation process are displayed in a corresponding text window.

 For information on translating IBIS and Spice models, refer to *Translating IBIS and Spice Models* section of the *Working with Signal Models and Libraries* chapter of Allegro SI SigXplorer User Guide.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Working with Device Models

 See the Cadence Sample Device Model Library for commented device model examples and sample formats. The Cadence sample device model library is located in your installation hierarchy in the following directory:

/install_dir/share pcb/signal/cds_iocells.ndx

You can perform the following tasks with the `signal library` command working with device models:

- [Editing an Analog Output Model](#)
- [Creating a Cable Model](#)
- [Editing a Cable Model](#)
- [Creating a DesignLink Model](#)
- [Editing a DesignLink Model](#)
- [Creating an ESpice Device Model](#)
- [Editing an ESpice Device Model](#)
- [Creating an IBIS Device Model](#)
- [Editing an IBIS Device Model](#)
- [Adding a Pin to an IBIS Device Model](#)
- [Editing the Pin Data for an Existing Pin on an IBIS Device Model](#)
- [Creating an IOCell Model](#)
- [Editing an IOCell Model](#)
- [Creating a Package Model by Copying and Editing an Existing Model](#)
- [Editing a Package Model](#)
- [Adding a Package Model to an IBIS Device Model](#)

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Editing an Analog Output Model

An Analog Output model characterizes a driver pin on an analog device. In Analog Output models, you specify Cadence Analog Workbench (AWB) wave files for rising and falling edges, pulses, and inverted pulses to describe the behavior of the driver pin.

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Select *AnalogOutput* in the Model Type Filter list.
3. Select an *AnalogOutput* model and click *Edit*.
The Analog Output Model Editor appears with the current data for the selected model.

 SigWave also launches, so you can view the waveform files that you are loading.

4. In the Analog Output Model Editor, specify a resistance value for a series resistor in the *Series Resistance* text box.
5. Specify the paths to one or more AWB wave files in the *Rise*, *Fall*, *Pulse*, or *Inv Pulse* text boxes.
- or -
Click on the *Rise*, *Fall*, *Pulse*, and *Inv Pulse* buttons with the text boxes empty to display a File browser that enables you to select AWB wave files to load.
6. When the paths to the wave files are displayed, click on the *Rise*, *Fall*, *Pulse*, and *Inv Pulse* buttons to load the specified AWB files.
The SigWave window shows you the waveforms for the AWB wave files in the model.
7. Click *OK*.
The Analog Output model is updated with the specified changes.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Creating a Cable Model

A Cable model is similar to a PackageModel. Both contain RLGC matrices. However, you insert a Cable model into a DesignLink model and you insert a PackageModel into an IBIS Device model.

 Before you create a new model, be sure that the library you want to add it to is designated as the working library.

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Click the *Add->* button and then select *Cable*.
A dialog box appears.
3. Enter the name of the model in the *New Cable model name* text box, then click *OK*.
The Cable model is created and added to the SI Model Browser list box.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Editing a Cable Model

To edit a cable model:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Select *Cable* in the Model Type Filter list.
3. Select a Cable model and click *Edit*.
Your default text editor appears displaying the model syntax.
4. Edit the syntax to modify the model, then choose *File – Save* in the text editor to save the changes.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Creating a DesignLink Model

 Before you create a new model, be sure that the library you want to add it to is designated as the working library.

To create a DesignLink model:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Click the *Add-> button and then select DesignLink*.
A dialog box appears.
3. Enter the name of the model in the *Model Name* text box, then click *OK*.
The DesignLink model is created and added to the Model Browser list box.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Editing a DesignLink Model

To edit a DesignLink model, follow these steps:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Select *DesignLink* in the Model Type Filter list.
3. Select a *DesignLink* model to edit in the SI Model Browser list box, then click *Edit*.
The System Configuration Editor appears with the current data for the selected model.
4. Modify the DesignLink parameters as desired, then click *OK*.
The model is updated with the specified changes.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Creating an ESpice Device Model

ESpice device models are models of discrete devices, which are written in a `.subckt` SPICE declaration.

 Before you create a new model, be sure that the library you want to add it to is designated as the working library.

Follow these steps to create an ESpice device model:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Click the *Add->* button and then select *ESpiceDevice*.
The Create ESpice Device Model dialog box appears.
3. Enter a name in the *Model Name* text box.
4. Click to display a menu of discrete device types in the *Circuit Type* field.
A menu appears.
5. Select one of the circuit type options; *Resistor*, *Capacitor*, or *Inductor*.
6. Specify an appropriate value in the *Value* text box. For example, specify a resistance value for a resistor.
7. Enter pin names in the *Single Pins* text box. Single pins have only one connection inside the package. The other type of pin (a common pin) has more than one connection inside the package.

 Be careful not to include a space in the pin name. Otherwise, a model with a double pin count is created.

8. Enter a pin name in the *Common Pin* text box. Common pins are typically the pins in a package that connect to power or ground. For example, a `SIP8` resistor pack can have seven resistors in its IC that can be designed to be pullups or pulldowns. In the resistor pack's model there is one common pin through which all seven resistors in the IC connect to power or ground and seven single pins that connect the interconnect in the design to the resistors in the IC.
9. In the *PinCount* text box, enter the number of physical pins in the package.
10. Click *OK*.
The ESpice device model is created and added to the Model Browser list box.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Editing an ESpice Device Model

Follow these steps to edit an ESpice device model:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Select *ESpiceDevice* in the Model Type Filter list.
3. Select an *ESpiceDevice* model and click *Edit*.
Your default text editor appears displaying the model syntax.
4. Edit the syntax to modify the model, then choose *File – Save* in the text editor to save the changes.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Creating an IBIS Device Model

IBIS Device models are assigned to ICs and connectors with the `SIGNAL_MODEL` property. An `IbisDevice` model for a connector has package parasitics but no IOCell models.

 Before you create a new model, be sure that the library you want to add it to is designated as the working library.

Follow these steps to create an IBIS device model:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Click the *Add->* button and then select *IbisDevice*.
The Create IBIS Device Model dialog box appears.
3. Enter a name in the *Model Name* text box.
4. Enter the number of pins in the model in the *Pin Count* text box.
5. Enter package pin parasitic values in the *Pin Parasitics R, L, and C* text boxes.
The values that you enter here apply to all pins in the model. If you need different parasitic values for some pins, you can change them by editing the model in the IBIS Device Model Editor.
6. Enter IOCell models in the *IOCell Model* text boxes.
The simulator fills these fields with the default IOCell models you specified in the Signal Analysis Parameters dialog box. If you want the model to use IOCells other than your default IOCell models, enter these IOCell models here.
7. Enter in the *Pins* text boxes (to the right of the IOCell Model fields) the names (pin numbers) of the pins that use these models and enter the names of the power and ground pins.
8. Click *OK*.
The model is created and added to the Model Browser list box.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Editing an IBIS Device Model

Follow these steps to edit an IBIS device model:

1. Choose *Analyze – Model Browser*.

The SI Model Browser dialog box appears.

2. Select an `IbisDevice` model to edit in the Model Browser list box, then click *Edit*.

The IBIS Device Model Editor appears with the current data for the selected model.

3. Use the three tabs of the IBIS Device Model Editor to edit the model as desired.

Use the *Edit Pins* tab to modify information about the pins associated with the IBIS Device model.

Use the *Assign Power/Ground Pins* tab to group power and ground pins and assign them to power and ground buses or to auto-assign buses to individual pins.

Use the *Assign Signal Pins* tab to group signal pins and assign them to IOCell models and buses.

4. When your edits are complete, click *OK*.

The device model is updated with the specified changes.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Adding a Pin to an IBIS Device Model

To add a pin to an IBIS device model:

1. Choose Analyze – Model Browser.
The SI Model Browser dialog box appears.
2. Select the `IbisDevice` model to edit in the Model Browser, then click *Edit*.
The IBIS Device Model Editor appears with the current data for the selected model.
3. In the *IBIS Device Pin Data* area, click *Add Pin Data*.
A prompt appears.
4. Enter a name for the new pin, then click *OK*.
The new pin is added to the *IBIS Pin Data* list box.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Editing the Pin Data for an Existing Pin on an IBIS Device Model

You can edit the pin data for an existing pin on an IBIS device model by following these steps:

1. Choose Analyze – Model Browser.
The SI Model Browser dialog box appears.
2. Select the `IbisDevice` model to edit in the Model Browser, then click *Edit*.
The IBIS Device Model Editor appears with the current data for the selected model.
3. In the *IBIS Pin Data* list box, click to select the individual pin you want to edit.
The IBIS Device Pin Data dialog box appears with the current data for the specified pin.

 The IBIS Device Model Editor remains open in the background. If you click a different pin, the IBIS Device Pin Data dialog box changes to display data for that pin.

4. Modify the pin data as desired, then click *OK*.
The pin is updated with the specified changes.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Creating an IOCell Model

 Before you create a new model, be sure that the library you want to add it to is designated as the working library.

Follow these steps to create an IOCell model:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Click *Add->*, then select one of the following model types from the pop-up menu.

- IbisIO
- IbisIO_OpenPullUp
- IbisIO_OpenPullDown
- IbisOutput
- IbisOutput_OpenPullUp
- IbisOutput_OpenPullDown
- IbisInput
- IbisTerminator

A dialog box appears.

3. Enter a name for the model, then click *OK*.

A new IOCell model of the type you selected is created using default values and its name is added to the Model Browser list box.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Editing an IOCell Model

Follow these steps to edit an IOCell model:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Select one of the following device model types to edit in the Model Browser list box, then click *Edit*.)
 - IbisIO
 - IbisIO_OpenPullUp
 - IbisIO_OpenPullDown
 - IbisOutput
 - IbisOutput_OpenPullUp
 - IbisOutput_OpenPullDown
 - IbisInput
 - IbisTerminator

The IOCell Editor appears with the current data for the selected model.

3. Use the four tabs of the IOCell Editor to modify the model data.
Use the *General* tab to describe the model. (You can invoke the VI Curve editor for PowerClamp and GroundClamp VI curves from this tab.)
Use the *Input Section* tab to describe the high and low logic thresholds for an input buffer.
Use the *OutputSection* tab to describe the rise and fall times for an output buffer. (You can invoke the VI Curve editor for PullUp and PullDown VI curves from this tab. You can invoke the VT Curve editor for RisingWave and FallingWave VT curves from this tab.)
Use the *Delay Measurement* tab to describe the test fixture and the measurement threshold (*Vmeasure*) used for buffer delay measurement.
4. When your edits are complete, click *OK*.
The IO Cell model is updated with the specified changes.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Creating a Package Model by Copying and Editing an Existing Model

A Package model is similar to a Cable model. Both contain RLGC matrices. However, you insert a Cable model into a DesignLink and you insert a Package model into an IBIS Device model. Cadence recommends that you create new Package models by cloning an existing Package model from the sample library and editing that copy to characterize the device you are modeling.

 Before you create a new model, be sure that the library you want to add it to is designated as the working library.

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. In the Model Browser list box, highlight the `PackageModel` you want to copy, then click *Edit*.
Your default text editor opens with the contents of the Package model.
3. Edit the syntax to modify the model, then choose *File – Save As* in the text editor to save the file as a new Package model.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Editing a Package Model

To edit a package model, follow these steps:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Select an `PackageModel` to edit in the Model Browser list box, then click *Edit*.
Your default text editor opens displaying the model syntax.
3. Edit the syntax to modify the model, then choose *File – Save* in the text editor to save the changes.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Adding a Package Model to an IBIS Device Model

To add a package model to an IBIS device model, follow these steps:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Select an `IbisDevice` model to edit in the Model Browser list box, then click *Edit*.
The IBIS Device Model Editor appears with the current data for the selected model.
3. In the Model Browser (still open in the background), use the browser to find and click the `PackageModel` you want to assign to the IBIS device model. The model appears in the *Package Model* field of the IBIS Device Model Editor dialog box.
4. Click *OK* in the IBIS Device Model Editor.
The `PackageModel` is added to the `IbisDevice` model.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Working with Interconnect Models

 See the [Allegro PCB SI User Guide](#) for information regarding the Interconnect Description Language (IDL) and the formats used for interconnect models.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

Editing a Trace, MultiTrace, Pin or Shape Model

 Before you create a new model, be sure that the library you want to add it to is designated as the working library.

To edit a trace, MultiTrace, pin, or a shape model, perform the following steps:

1. Choose *Analyze – Model Browser*.
The SI Model Browser dialog box appears.
2. Select a model to edit from the Model Browser list box, then click *Edit*.
Your default text editor opens displaying the model syntax.

 Edit the syntax to modify the model, then choose *File – Save* in the text editor to save the changes.

Related Topics

- [signal library](#)
- [Signal Analysis Library Dialog Boxes](#)
- [Signal Library Tasks](#)

signal libs audit

The `signal libs audit` command opens a file browser from which you can access a list file containing design model library file names. When you choose a file, the `dmlcheck` utility verifies the formatting of the files in the list.

 This command is not available in L Series products.

signal model

The `signal model` command displays the Signal Model Assignment dialog box for assigning models to devices, pins, and bondwires. You can also remove model assignments.

The simulator uses device models to create complete circuit simulation models for nets in your design, which means that you can assign a device model to each component in the design. You can assign a device model either to an individual component or to all components having the same device file.

Related Topics

- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Signal Model Assignment Dialog Box

Access Using

- Menu Path: *Analyze – Model Assignment*



- Toolbar Icon:

Use this dialog box to assign models to design elements.

- Use the Devices tab to assign device models to components, automatically or manually. You can browse for device models, modify existing models before assigning them, and create new models. You can also load and save the Assignment Mapping file for the design.
- Use the BondWires tab to locate and assign trace models to bondwire connections. You can also modify trace models.
- Use the RefDesPins tab to assign IOCell models to specific components or component pins. You can also assign models to pins that have a selection of programmable buffer models.
- Use the Connectors tab to assign coupled connector models to components such as male/female connectors, PCI slots, and other components that connect one design to another.

When you finish edits to model assignments, a report appears listing the changes.

Devices Tab

Use the Devices tab to assign device models to components in the design or to create new models. When the Model Browser is open along with the Signal Model Assignment dialog box, the name of a model chosen in the Model Browser also displays in the Signal Model: field.

<i>DevType/Refdes</i>	Displays, in tree form, device types for components. Expands to display reference designators for components of that type.
<i>Signal Model list</i>	Displays the model name currently assigned to a device type or reference designator.
<i>Signal Model field</i>	Displays the model name currently chosen for assignment.
<i>Create Model</i>	Displays the Create Device Model dialog box for the chosen component. Prompts you to create either an Espice Device model or an IBISDevice Model for the component.
<i>Find Model</i>	Displays the Model Browser configured to find device models matching: The assigned SIGNAL_MODEL (if one exists). The Allegro PCB Design Entry HDL or System Connectivity Manager DEFAULT_SIGNAL_MODEL (if one exists). The device type.
<i>Edit Model</i>	Invokes an appropriate editor so you can modify the model.
<i>Save (Assignment Map File:)</i>	Saves model assignments to a model assignment data file which maps device types to appropriate models.
<i>Load (Assignment Map File:)</i>	Loads a model assignment data file.
<i>Auto Setup</i>	Start automatic model assignment for all discrete components with a value attribute.
<i>Preferences</i>	Displays the Analysis Preferences dialog box.

Bondwires Tab

This tab is used only for package designs. Use this to assign trace models to individual bondwire connections in the design or to modify trace models. Bondwires are connect lines (clines) on wire bond layers. When the Model Browser is open along with the Signal Model Assignment dialog box, the name of a trace model chosen in the Model Browser also displays in the Signal Model: field.

<i>Pkg Pin, and Net</i>	Displays, in tree form, the die pad, package pin, and net for a bondwire.
-------------------------	---

S Commands

S Commands--signal model

<i>Signal Model</i> list	Displays the trace model currently assigned to a bondwire.
<i>Signal Model</i> field	Displays the model name chosen for assignment.
<i>Find Model</i>	Displays the Model Browser configured to show trace models.
<i>Edit Model</i>	Invokes a text editor so you can modify a trace model.
<i>Preferences</i>	Displays the Analysis Preferences dialog box.
<i>Assign Current Signal Model to BondWire Picks</i>	Click to assign the signal model named in the <i>Signal Model:</i> field to the chosen BondWires.
<i>All (Assign Current Signal Model to all BondWires)</i>	Click All to assign the signal model named in the <i>Signal Model:</i> field to all BondWires.

RefDesPins Tab

Use the RefDesPins tab to assign IOCell models and programmable buffer models to individual pins identified by reference designator. When the Model Browser is open along with the Signal Model Assignment dialog box, the name of a model chosen in the Model Browser also displays in the *Signal Model:* field.

<i>Refdes</i>	Displays, in tree form, the reference designators for components in the design. Expands to display pin numbers and pinuse.
<i>Signal Model</i> list	Displays the IOCell models assigned to the pins.
<i>Device Class Filter</i>	Displays a list of available device classes to limit the display of reference designators in the list box. Choices include: IC, DISCRETE, IO, Other, *.
<i>Signal Model</i> field	Displays the model name chosen for assignment.
<i>Prog Buffers-></i>	Displays a list of the Programmable Buffer models available in the library for the chosen pin. The number to the right of the button indicates the number of available models for the chosen pin.
<i>Create Model</i>	Inactive.
<i>Find Model</i>	Displays the Model Browser configured to show IOCell or programmable buffer models.
<i>Edit Model</i>	Invokes an editor so you can modify the chosen model.
<i>Display Programmable Buffer Pins Only/Display All</i>	Toggles between display of all pins in the design (by component) or only pins which have programmable buffers.
<i>Preferences</i>	Displays the Analysis Preferences dialog box.

Connectors Tab

Use the Connectors tab to assign coupled connector models to components such as male/female connectors, PCI slots, and other components that connect one design to another.

<i>DevType Value Refdes</i>	Displays, in tree form, device types for components. Expands to display reference designators for components of that type.
<i>Connector Model</i> list	Displays the model name chosen for assignment to the component.
<i>Source Library</i>	Displays the source library in which the selected connector model resides.
<i>Connector Model</i> field:	Lets you type in an existing connector model name for specified components. Connector model names previously entered in the field appear as selection entries in the drop-down. You can clear individual model assignments by choosing the <i>No Model</i> entry.
<i>Find Model</i>	Displays the Model Browser configured to Device Library models.

<i>Edit Model</i>	The functionality for editing connector models is not available in the 16.01 release.
<i>Clear All Connector Model Assignments</i>	Clears all connector models assigned to components in the design.

Related Topics

- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Create Device Model Dialog Box

Button	Function
<i>Create IbisDevice model</i>	Selects the Create IBIS Device model editor.
<i>Create ESpiceDevice model</i>	Selects the Create ESpice Device model editor.
<i>OK</i>	Displays the chosen model editor.

Related Topics

- [signal model](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Create IBIS Device Model Editor Dialog Box

Option	Function	
<i>ModelName</i>	Specifies the name of the model to create.	
<i>PinCount</i>	Specifies the number of pins in the model.	
<i>Pin Parasitics</i>	Specifies the parasitic values for the package: Options are:	
	<i>R</i>	Sets the pin resistance value
	<i>L</i>	Sets the pin inductance value
	<i>C</i>	Sets the pin capacitance value
<i>IOCell Model fields</i>	Displays the default IOCell models for each PINUSE type. These default values are defined in the Signal Analysis Preferences dialog box (General Tab). To use IOCell models different from the defaults, enter a different IOCell model name for the PINUSE value. Options are:	
	<i>IN</i>	Specifies an IOCell model to use for pins with a PINUSE value of IN.
	<i>OUT</i>	Specifies an IOCell model to use for pins with a PINUSE value of OUT.
	<i>BI</i>	Specifies an IOCell model to use for pins with a PINUSE value of BI (bidirectional).
	<i>TRI</i>	Specifies an IOCell model to use for pins with a PINUSE value of TRI (tristate).
	<i>OCL</i>	Specifies an IOCell model to use for pins with a PINUSE value of OCL (open drain).
	<i>OCA</i>	Specifies an IOCell model to use for pins with a PINUSE value of OCA (open source).
<i>Pins fields</i>	Displays the pin name associated with each PINUSE type. Specify the pin name corresponding to each PINUSE type: IN, OUT, BI, TRI, OCL, OCA, Power, and Ground.	

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Create ESpice Device Model Editor Dialog Box

Option	Function
<i>Model Name</i>	Displays the name of the model to create.
<i>Circuit Type</i>	Displays a pop-up menu of discrete device types: Resistor, Capacitor, or Inductor.
<i>Value</i>	Displays a numeric value for the device type.
<i>Single Pins</i>	Enter the names of single pins that have only one connection in the package. Do not include spaces in the pin names.
<i>Common Pin</i>	Enter the name of a common pin (if one exists) that typically connects to power or ground. Otherwise, leave the field blank.
<i>Pin Count</i>	Enter the number of physical pins in the package.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)

Signal Model Assignment Tasks

Following tasks are associated with the `signal model` command:

- [Assigning Device Models to Components](#)
- [Assigning IOCell Models to Pins](#)
- [Assigning Programmable Buffer Models to Pins](#)
- [Assigning Trace Models to Bondwires](#)
- [Automatically Assigning Device Models to Discrete Components](#)
- [Creating a New Device Model in Model Assignment](#)
- [Editing a Device Model in Model Assignment](#)
- [Removing a Device Model Assignment](#)
- [Setting Up Models in Allegro PCB Design Entry HDL, System Connectivity Manager, or Third-Party Libraries](#)

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)

Assigning Device Models to Components

If you know the reference designator for a specific component to which you want to assign a model, use the RefDesPins tab to assign the model.

To assign an existing device model to a component:

1. Run `signal model`.
The Signal Model Assignment dialog box appears.
2. Click to choose the Devices tab.
3. In the *DevType/Refdes* column, either Click to choose the device type (to assign a device model to all components with this device type). --or — Click to expand the device type. Then click to choose the reference designator (to assign a device model to a single component).
4. Click *Find Model*.
The SI Model Browser dialog box opens.
5. Use the SI Model Browser dialog box to find and choose the device model you want to assign. Click the model name in the SI Model Browser dialog box.
The name appears in the Model Name: field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the device type or reference designator that you chose.
6. Click *Save* to save the assignments to the Model Assignment Mapping file.
7. Click *OK*.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Assigning IOCell Models to Pins

To assign IOCell models to pins:

1. Run [signal model](#).
The Signal Model Assignment dialog box appears.
2. Click to choose the RefDesPins tab.
3. In the Device Class Filter field, click to display a pulldown menu of filter choices (IC, DISCRETE, IO, Other, and *).
4. Click to choose a filter to restrict display of components in the list box.
5. In the Refdes column, click to expand a reference designator into its pin list. Each pin description can include the pin number, pinuse, and any assigned IOCell model.
The components displayed in the list box are restricted by any choice made in the Device Class Filter field.
6. In the Refdes column, click to choose a pin.
7. Click *Find Model*.
The SI Model Browser dialog box opens.
8. Use the SI Model Browser dialog box to find and choose the IOCell model you want to assign. Click the model name in the SI Model Browser dialog box.
The name appears in the Model Name: field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the pin that you chose.
9. To modify the IOCell model, click *Edit Model*.
The IOCell model editor appears with the data for the model in place. Edit the model data and save your edits.

The modified IOCell model is saved to the working library and its name appears in the *Model Name:* field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the pin that you chose.

1. Click *Save* to save the assignments to the Model Assignment Mapping file.
2. Click *OK*.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Assigning Programmable Buffer Models to Pins

Perform the following steps to assign programmable buffer modes to pins:

1. Run signal model.
The Signal Model Assignment dialog box appears.
2. Click to choose the RefDesPins tab.
3. In the Refdes column, click to expand a reference designator into its pin list. Each pin description can include the pin number, pinuse, and any assigned IOCell model.
4. In the Refdes column, click to choose a pin.
5. When Programmable Buffer models are selectable, the Prog Buffers -> button is active. The number to the right of the button indicates the number of available models.
6. Click *Prog Buffers ->* to display a list of available models.
7. Click to choose a model.
The name appears in the *Model Name:* field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the pin that you chose.
8. When many pins appear and the minority are programmable buffers, click *Display Programmable Buffers Pins Only* to refresh the display and show only programmable buffer pins. The button text changes to *Display All*.
9. Click *OK*.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Assigning Trace Models to Bondwires

You can assign trace models to bond wires (and remove trace model assignments) by using any of the following methods:

- [Assigning a Trace Model to a Single Bond Wire](#)
- [Assigning a Trace Model to Multiple Bond Wires](#)
- [Assigning a Trace Model to All Bond Wires](#)
- [Removing All Trace Model Assignments](#)

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Assigning a Trace Model to a Single Bond Wire

Follow these steps to assign a trace model to a single bond wire:

1. Run `signal model`.
The Signal Model Assignment dialog box appears.
2. Click to choose the Bondwires tab.
3. In the Select Model Assignment Options area, verify that Assign current signal model to BondWire picks is inactive. By default, this option is inactive.
4. In the Die Pad Pkg Pin Net column, choose a bond wire.
Any model assigned to the BondWire appears in the Model Name: field.
5. Click *Find Model*.
The SI Model Browser dialog box appears.
6. Use the SI Model Browser dialog box to find and choose the trace model you want to assign. Click the model name in the SI Model Browser dialog box.
The name appears in the Model Name: field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the bondwire that you chose.
7. To modify the trace model, click *Edit Model*.
A text editor appears with the data for the model in place. Edit the model data and save your edits.
The modified model is saved to the working library and its name appears in the *Model Name*: field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the bondwire that you chose.
8. Click *Save* to save the assignments to the Model Assignment Mapping file.
9. Click *OK*.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Assigning a Trace Model to Multiple Bond Wires

Follow these steps to assign a trace model to multiple bondwires:

1. Run [signal model](#).
The Signal Model Assignment dialog box appears.
2. Click to choose the Bondwires tab.
3. In the Select Model Assignment Options area, click to choose Assign current signal model to BondWire picks. By default, this option is inactive.
4. Click Find Model.
The SI Model Browser dialog box appears.
5. Use the SI Model Browser dialog box to find and choose the trace model you want to assign. Click the model name in the SI Model Browser dialog box.
The trace model name appears in the Model Name: field.
6. In the Die Pad Pkg Pin Net column, choose a bondwire.
The signal model named in the *Model Name:* field is assigned to the bondwire that you chose. The model name is added in the Signal Model column in the Signal Model Assignment dialog box, on the line for the bondwire that you chose.
7. Optionally, you can continue to choose bondwires in the Die Pad Pkg Pin Net column.
Each time you choose a bondwire, the signal model named in the *Model Name:* field is assigned to the bondwire that you chose. The model name is added in the Signal Model column in the Signal Model Assignment dialog box, on the line for the bondwire that you chose.
8. To modify the trace model, click Edit Model.
A text editor appears with the data for the model in place. Edit the model data and save your edits.
The modified model is saved to the working library and its name appears in the *Model Name:* field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the bondwire that you chose.
9. Click Save to save the assignments to the Model Assignment Mapping file.
10. Click *OK*.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Assigning a Trace Model to All Bond Wires

To assign a trace model to all bond wires in the design:

1. Run `signal model`.
The Signal Model Assignment dialog box appears.
2. Click to choose the Bondwires tab.
3. In the Select Model Assignment Options area, verify that Assign current signal model to BondWire picks is inactive. By default, this option is inactive.
4. Click Find Model.
The SI Model Browser dialog box appears.
5. Use the SI Model Browser dialog box to find and choose the trace model you want to assign. Click the model name in the SI Model Browser dialog box.
The trace model name appears in the *Model Name*: field.
6. In the Select Model Assignment Options area, click *All* to assign current signal model to all bondwires
The signal model named in the *Model Name*: field is assigned to all bondwires. The model name is added in the Signal Model column in the Signal Model Assignment dialog box, on the line for each bondwire.
7. To modify the trace model, click *Edit Model*.
A text editor appears with the data for the model in place. Edit the model data and save your edits.
The modified model is saved to the working library and its name appears in the *Model Name*: field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the bondwire that you chose.
8. Click *Save* to save the assignments to the Model Assignment Mapping file.
9. Click *OK*.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Removing All Trace Model Assignments

To remove all trace model assignments, follow these steps:

1. Run [signal model](#).
The Signal Model Assignment dialog box appears.
2. Click to choose the Bondwires tab.
3. In the Select Model Assignment Options area, verify that Assign current signal model to BondWire picks is inactive. By default, this option is inactive.
4. Delete any trace model name that appears in the Model Name: field.
The *Model Name:* field is empty.
5. In the Select Model Assignment Options area, click All to Assign current signal model to all BondWires
Since the Model Name: field is empty, all trace model name assignments in the Signal Model column in the Signal Model Assignment dialog box are removed.
6. Click Save to save the assignments to the Model Assignment Mapping file.
7. Click OK.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Automatically Assigning Device Models to Discrete Components

During automatic model assignment, the simulator attempts to assign models to all components with two pins which have a non-zero VALUE property and no previous model assignment.

To automatically assign device models to discrete components like resistors and capacitors:

1. Run `signal model`.
The Signal Model Assignment dialog box appears.
2. Click to choose the Devices tab.
3. Click *Auto Setup*.
4. Optionally, click *Save* to save the assignments to the Model Assignment Mapping file.
5. Click *OK*.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Creating a New Device Model in Model Assignment

You can create a new device model in model assignment by following these steps:

1. Run `signal model`.
The Signal Model Assignment dialog box appears.
2. Click to choose the Devices tab.
3. In the DevType/Refdes column, either Click to choose the device type (to assign a device model to all components with this device type). --or — Click to expand the device type. Then click to choose the reference designator (to assign a device model to a single component).
4. Click *Create Model*.
The Create Device Model dialog box appears.
5. Choose *Create E-Spice Device* or *Create IBIS Device* and click *OK*.
The respective create model dialog box appears.
6. Use the dialog box to enter new parameters for the model and click *OK*.
The new model is saved to the working library and its name appears in the Model Name: field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the device type or reference designator that you chose.
7. Optionally, click *Save* to save the assignments to the Model Assignment Mapping file.
8. Click *OK*.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Editing a Device Model in Model Assignment

To edit a device model in model assignment, follow these steps:

1. Run `signal model`.
The Signal Model Assignment dialog box appears.
2. Click to choose the Devices tab.
3. In the DevType/Refdes column, either Click to choose the device type (to assign a device model to all components with this device type). --or — Click to expand the device type. Then click to choose the reference designator (to assign a device model to a single component).
4. Click *Find Model*.
The SI Model Browser dialog box opens.
5. Use the Model Browser to find and choose a device model similar to the one you want to assign. Click the model name in the SI Model Browser dialog box.
The name appears in the *Model Name*: field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the device type or reference designator that you chose.
6. Click *Edit Model*.
The appropriate device model editor appears with the data for the model in place.
7. Edit the model data and click *OK* to save your edits.
The modified model is saved to the working library and its name appears in the *Model Name*: field and in the Signal Model column in the Signal Model Assignment dialog box, on the line for the device type or reference designator that you chose.
8. Assign the edited model following the appropriate model assignment task.
9. Click *Save* to save the assignments to the Model Assignment Mapping file.
10. Click *OK*.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Removing a Device Model Assignment

Perform the following steps to remove a device model assignment:

1. Run `signal model`.
The Signal Model Assignment dialog box appears.
2. Click to choose the Devices tab.
3. In the DevType/Refdes column, click to choose either a general device type or the specific reference designator from which you want to remove a model.
4. Click in the *Signal Model:* field and enter a backspace to erase the name.
5. Click *OK*.

The device model assignment to that component is removed, and the Signal Model Assignment Changes Report appears.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

Setting Up Models in Allegro PCB Design Entry HDL, System Connectivity Manager, or Third-Party Libraries

The model setup for components can be specified in Allegro PCB Design Entry HDL or System Connectivity Manager libraries. When the simulator finds that a component does not have a SIGNAL_MODEL property, it checks to see if there is a SIGNAL_MODEL property on the device definition. You can attach SIGNAL_MODEL properties to device definitions by setting the SIGNAL_MODEL property on one of the following:

- On the `chips_prt` file for the device
- On the `phys_prt.dat` file for your schematic
- On the layout editor device file (if you are using netin)

The value of the SIGNAL_MODEL property must be the name of an IBISDevice or ESpiceDevice model. Furthermore, the simulator validates all model assignments based on the PINUSE property.

The SIGNAL_MODEL property assigned to components using the Signal Model Assignment dialog box (instances) overrides those in the device definition, if they exist.

The simulator uses the following precedence to determine which model gets assigned to a device:

1. An instance-specific SIGNAL_MODEL assignment made in the Signal Model Assignment dialog box (stored in the `.brd` file)
2. A SIGNAL_MODEL property on the component definition (Allegro PCB Design Entry HDL or System Connectivity Manager's PPT file)
3. A VOLT_TEMP_MODEL property on the component definition (Allegro PCB Design Entry HDL or System Connectivity Manager's PPT file)
4. A DEFAULT_SIGNAL_MODEL property on the component definition (Allegro PCB Design Entry HDL or System Connectivity Manager's PPT file)

A common use of the DEFAULT_SIGNAL_MODEL property is to establish a model name for the device before the actual model is developed. The simulator warns you when a part with a SIGNAL_MODEL property does not have an associated model; however, if a default model name is attached to a part, as directed by having checked the *Use Defaults For Missing Components Models* in the DeviceModels tab of the Analysis Preferences dialog box, the simulator does not report an error when a model is not yet available.

You can use a default model name pattern as a placeholder for a to-be-procured library of models or for implementing model names based on your internal model naming conventions.

Related Topics

- [signal model](#)
- [Signal Model Assignment Dialog Box](#)
- [Create Device Model Dialog Box](#)
- [Create IBIS Device Model Editor Dialog Box](#)
- [Create ESpice Device Model Editor Dialog Box](#)
- [Signal Model Assignment Tasks](#)

signal modeedit

The `signal modeedit` command is no longer supported. To access Model Editor:

- Use the [`model editor`](#) command.
- Choose [*Analyze – Model Browser*](#) and launch Model Editor from [*SI Model Browser*](#).

For more information on Model Editor, see the *Working with Model Editor* chapter in *Allegro SI SigXplorer User Guide*

signal model refresh

The `signal model refresh` command lets you perform verification and source management operations on the device models in a chosen design or library.

Upon display of the Model Dump/Refresh dialog box, models resident in the current design are checked against their original source while a meter is displayed showing the progress of the task. Upon completion of the check, a window within the dialog box displays a list of all models in the design including related source and status information for each model.

 This command is not available in L Series products.

Related Topics

- [Generating Device Model Reports](#)

Model Dump/Refresh Dialog Box

Access Using

- Menu Path: *Analyze – Model Dump/Refresh*

Use this dialog box to perform verification and source management operations on the device models in a chosen design or library.

<i>Source list</i>	A list of all available designs in the System Configuration including chosen and used device libraries in the design. The Source menu can also be used to change the original source of a model. To do so, first choose a Signal Model from the model window in the dialog box, then use the Source menu to change the source. The Status field is updated by a quick model check if the chosen source contains the model. Otherwise, the original source remains the same and a <i>Model_Not_Found</i> message is displayed.
<i>Signal Model</i>	A file structure showing all models stored in the chosen design or the chosen library. The models are grouped according to category. Categories can be expanded to see model names.
<i>Source</i>	The original source (a device library or a design in the system) for all models.
<i>Status</i>	A current status message for the models based on an automatic source check. For details, see <i>About Model Status Messages</i> .
<i>Browse</i>	Enables the user to add a library to the source list. Can also be used to change the original source of the chosen model. If that model is found in the chosen library, the Status field is updated as a result of a quick model check. If the model is not found, the original source of the chosen model remains the same and a <i>Model_Not_Found</i> message is displayed.
<i>View Differences</i>	Invokes the generation of a <i>Model Differences</i> (line-by-line comparison) report for the chosen model in a pop-up window. For details, see <i>About Generating Model Refresh Reports</i>
<i>Refresh</i>	Overwrites the chosen model in the current design with the model data contained in its source when its Status field contains a number greater than zero. This takes effect upon clicking either <i>Apply</i> or <i>OK</i> . The Refresh button is grayed out if a model is not chosen or if the model status is listed as <i>Same</i> .
<i>Refresh All</i>	Overwrites all the models in the current design that are different with the model data contained in their corresponding source. This takes effect upon clicking either <i>Apply</i> or <i>OK</i> . Note: The Refresh button is grayed out if no models in the current design are different.
<i>Dump</i>	Dumps (writes) all the device models in the current design to: <i>boardname.dml/boardname_encrypted.dml</i> . When the dump is complete, a confirmation pop-up window is displayed with a message which reads: <i>Model dump finished. For log messages, see dump_libraries.log</i> .
<i>OK</i>	Applies all the changes made and closes the form.
<i>Apply</i>	Applies all the changes made in the form without closing the form. After applying the changes, a <i>Model Refresh</i> summary report is displayed to verify all the changes.

Generating Device Model Reports

There are two device model reports that can be generated from the Model Dump/Refresh dialog box.

Model Refresh Summary Report

When you use of the Refresh and Apply buttons in the Model Dump/Refresh dialog box, the models in the current design are refreshed individually and changes applied (without having to close the form). When the Apply button is chosen, a Model Refresh report is displayed providing verification on the models refreshed thus far.

View Differences Report

When the status of a device model is listed as an integer, there are differences between the model code in the current design and its source. These differences can be easily checked by first choosing the model and then clicking View Differences on the Model Dump/Refresh dialog box.

This report shows a line-by-line comparison of the differences between the chosen model's data within the current design and its source. If no differences are detected, a message stating this condition is displayed.

Related Topics

- [signal model refresh](#)

signal demiprefs

See [signal prefs](#).

signal demiprobe

See [signal probe](#).

signal xtalktable

The `signal xtalktable` command displays the Signal Analysis Crosstalk Table dialog box for managing crosstalk tables. (This command can also be run in batch mode using `signoise`.)

 This command is not available in L Series products.

Related Topics

- [Creating a Crosstalk Table](#)
- [Selecting a Crosstalk Table](#)
- [Exporting a Crosstalk Table](#)
- [Importing a Crosstalk Table](#)
- [Deleting a Crosstalk Table](#)

Signal Analysis Crosstalk Table Dialog Box

Access Using

- Menu Path: *Analysis – Xtalk Table*

This dialog box allows you to read data into the design database from an existing crosstalk table file (.xtb), create a new crosstalk table, and export a crosstalk table to an external file.

The simulator uses crosstalk tables in crosstalk estimation and crosstalk DRC checking. The Allegro PCB Router XL Interface (SPIF) uses these tables to provide crosstalk constraint values to the Allegro PCB Router XL router.

To generate a crosstalk table, the layout editor or Allegro PCB SI invokes the SigNoise simulator, which performs these actions:

- Extracts IOCell models for drivers, spacing constraint sets, and layer stackup information from the design.
- Performs a crosstalk simulation for the fastest drivers on each net in the design, all trace spacing (line-to-line spacing) rules defined in the design, and all possible routing layer combinations. The simulator increments the trace spacings and repeats the process until it reaches the maximum trace spacing as specified by the Crosstalk Geometry Window value on the InterconnectModels tab of the Analysis Preferences dialog box.

Select Table Section

<i>Selected Table</i>	Lets you select from the drop-down menu a crosstalk table file stored in the design database. The file you select is used for simulation.
<i>Table Simulation Use Mode</i>	Allows you to select the driver speed. You must define the speed used in the table.
<i>Export Table</i>	Lets you store the selected table file in the design database to an external file. The file is saved in the .xtb text format.
<i>Import Table</i>	Lets you import a crosstalk table into the design database which you can then select from the drop-down menu.
<i>Delete Table</i>	Lets you delete the selected crosstalk table from the design database.

Create Table Section

Table Name	Use to enter the name of a new crosstalk table file.
Transmission Line Impedances	Enter the transmission line impedances (in ohms). The simulator uses these values for transmission line impedance and termination in creating pseudo-circuits used for crosstalk table generation. If your design contains impedance rules, four of the impedance values specified in the rules will be displayed as the default values in these fields. If no impedance rules are defined in the design, the impedance value that has been set in the <i>Default Impedance</i> field of the Analysis Preferences dialog box is the default selection.
Table Simulation Create Mode	Specifies the speeds of the driver models that you want to include in the table. You may choose any or all of the speeds.
Line Separation Values	If the difference between the above two values is very large, intermediate values are automatically added. If a line separation value is larger than the geometry window, the program generates an error message.
Include Plane Layers	Adds rows to the table that will define the crosstalk between lines on power and ground planes and other lines on either the same layer or on adjacent conductor layers. Note: If the board contains only a single plane layer, no plane layer data will be generated. Plane layer information is generated only on boards containing two or more planes in the stackup.
Create Table	Runs the simulation, generates the crosstalk table, and stores it in the design database.
Close	Click to close the dialog box.
Preferences	Click to display the Analysis Preferences dialog box, where you can set the parameters for simulation.

Related Topics

- [Selecting a Crosstalk Table](#)
- [Exporting a Crosstalk Table](#)
- [Importing a Crosstalk Table](#)
- [Deleting a Crosstalk Table](#)

Creating a Crosstalk Table

To create a crosstalk table, follow these steps:

1. Run `signal xtalktable`.
2. Enter a new table name in the *Table Name* field.
3. Enter up to four transmission line impedance values (in Ohms). If your design contains impedance rules, four of the impedance values specified in the rules will be displayed as the default values in these fields. If no impedance rules are defined in the design, a single impedance value of 60 ohms is the default selection.
4. Choose one or more simulation modes for Table Simulation Create Mode.
5. Enter up to eight line separation values. Default values for three conditions are displayed, but you can change them.
6. Check *Include Plane Layers* if you want to add rows to the crosstalk table that define crosstalk between lines on the plane layers of your design.
7. Click *Create Table*.

SigNoise creates a crosstalk table, based on simulation, in each simulation speed mode you choose. A message giving you the total number of simulations that will be run allows you to continue or to make edits to the creation of the table before creating it. Once you have created the table, it is stored in your design database and becomes available for selection.

Related Topics

- [signal xtalktable](#)
- [Exporting a Crosstalk Table](#)
- [Importing a Crosstalk Table](#)
- [Deleting a Crosstalk Table](#)

Selecting a Crosstalk Table

To select a crosstalk table, follow these steps:

1. Run `signal xtalktable`.
2. Choose a crosstalk table from the *Selected Table* drop-down menu.
3. For *Table Simulation Use Mode*, choose one of the simulation modes associated with the selected table.
4. The table you selected will be the one used to compute estimated crosstalk.

Related Topics

- [signal xtalktable](#)
- [Signal Analysis Crosstalk Table Dialog Box](#)
- [Importing a Crosstalk Table](#)
- [Deleting a Crosstalk Table](#)

Exporting a Crosstalk Table

To export a crosstalk table, follow these steps:

1. Run `signal xtalktable`.
2. Choose a crosstalk table from the *Selected Table* drop-down menu.
3. Click *Export Table* to store the selected table file in the design database to an external file. The file is saved in the `.xtb` text format.

Related Topics

- [signal xtalktable](#)
- [Signal Analysis Crosstalk Table Dialog Box](#)
- [Creating a Crosstalk Table](#)
- [Deleting a Crosstalk Table](#)

Importing a Crosstalk Table

1. Click *Import Table* to import a crosstalk table into the design database which you can then select from the drop-down menu.

Related Topics

- [signal xtalktable](#)
- [Signal Analysis Crosstalk Table Dialog Box](#)
- [Creating a Crosstalk Table](#)
- [Selecting a Crosstalk Table](#)

Deleting a Crosstalk Table

1. Click *Delete Table* to delete the selected crosstalk table from the design database.

Related Topics

- [signal xtalktable](#)
- [Signal Analysis Crosstalk Table Dialog Box](#)
- [Creating a Crosstalk Table](#)
- [Selecting a Crosstalk Table](#)
- [Exporting a Crosstalk Table](#)

signalintegrity

The `signalintegrity` command activates the *Signal Integrity* application mode. The application mode functionality in Allegro X PCB Editor provides an intuitive environment in which frequently-used commands are readily accessible from right mouse button pop-up menus, based on the selected set of design elements. In addition to the pop-up menus accessible from right-mouse click, an application mode in PCB Editor provides the following functionality:

- Highlighting objects when mouse pointer hovers over the elements.
- Auto-execution of default commands on double-clicking or dragging an element.
- A Super filter which lets you limit the find criteria to a single object type. You can choose the object type you want to find from the menu and select the object in the design.

PCB Editor supports the following application modes:

- General Edit
- Placement Edit
- Flow Planning
- Etch Edit
- Signal Integrity

The Signal Integrity (SI) application mode provides quick and easy access to frequently-used SI commands. The SI application mode configures the tool for a specific task by populating the right mouse button pop-up menu only with commands that operate on the currently selected element(s).

Access Using

- Menu Path: *Setup – Application Mode – Signal Integrity*

Commands Automatically Run in Application Mode

In addition to the pop-up menu, the application mode sets up commands that are automatically executed when you drag or double click on certain elements. The following table lists what happens to a design element when a specific action is performed on it in any application mode. The italicized items are specific to the SI application mode.

Auto Execution of Commands

Element	On Drag	On Shift-Drag	On Ctrl-Drag	On Double-Click
Net	Move	Move	Copy	<i>Launch SI Design Audit wizard</i>
Symbol	Move	Spin	Copy	<i>Launch Signal Model Assignment</i>
Pin	None	None	None	Add Connect
DRC_ERROR	None	None	None	<i>Launch Show Element</i>
Cline_Seg	Slide	None	Delay tune	<i>Launch Signal Analysis</i>
Ratsnest	None	None	None	Add Connect
Via	Slide	Move	Copy	None

Related Topics

- [signalintegrity](#)
- [Signal Integrity Command Tasks](#)

Signal Integrity Command Tasks

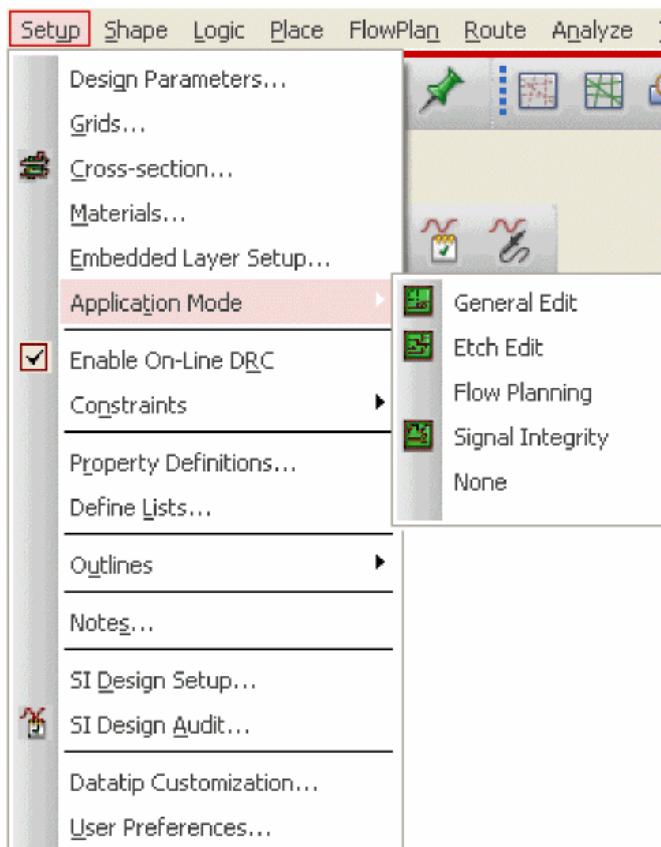
- Activating Application Mode
- Controlling Application Mode Status Icon and Tip
- Deactivating Application Mode
- Accessing Frequently-Used SI Commands
- Highlighting Objects on Mouse Over
- Using Super Filter
- Customizing Layout Editor Functions

Activating Application Mode

Signal Integrity mode is the default application mode when you initially launch the tool. You can also manually activate the SI application mode using one of the five methods described here:

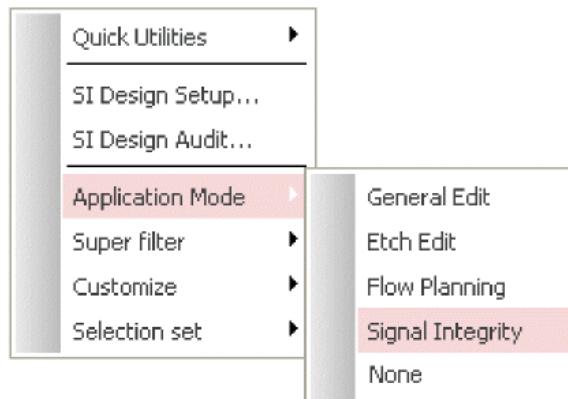
1. From the Main application menu.

Choose *Setup – Application Mode – Signal Integrity*.



2. From the pop-up menu on right-mouse click.

Right-click the canvas and choose *Application Mode – Signal Integrity* from the pop-up menu.



3. From the toolbar icon:

Click the Signal Integrity toolbar icon:

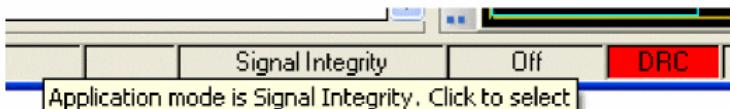


4. From the Console command

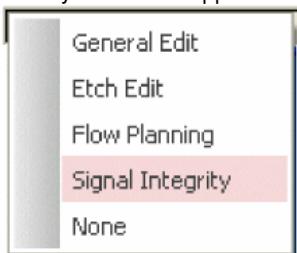
Type `signalintegrity` in the Console window and press Enter.

5. From the Status Bar

You can quickly check to see which application mode is active by moving your mouse cursor over the application box in the status bar



When you click the application mode name, a shortcut menu appears, from where you can change the current application mode.



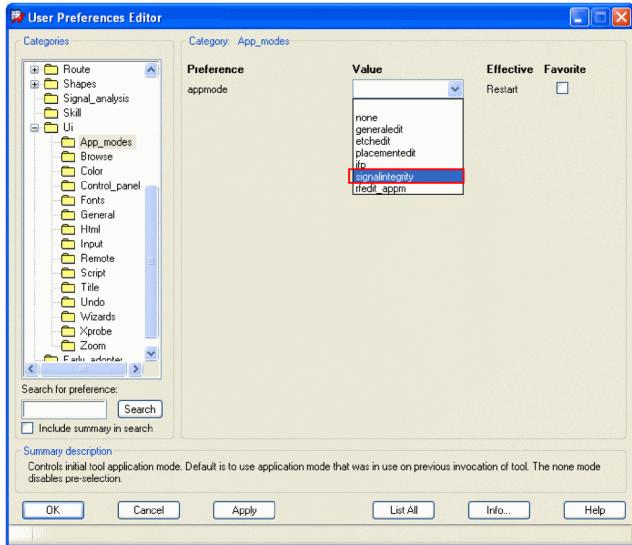
Related Topics

- [signalintegrity](#)
- [Signal Integrity Command Tasks](#)

Controlling Application Mode Status Icon and Tip

You can also use the `appmode` environment variable to control which application mode launches on startup, which defaults to the application mode used on previous invocation of the tool.

1. Choose *Setup – User preferences*.
2. In the User Preferences Editor, choose *Ui* and then choose *App_modes*.
3. Choose *signalintegrity* in the Values field for the `appmode` preference and click *OK* or *Apply*.



Related Topics

- [signalintegrity](#)
- [Signal Integrity Command Tasks](#)

Deactivating Application Mode

1. Use one of the three methods to exit from the current application mode and return to a menu-driven editing mode:
 - Choose *Setup – Application Mode – None*.
OR
 - Right-click on the canvas and choose *Application Mode – None*.
OR
 - At the command console, type `noappmode` and press Enter.
OR
 - Click the application mode in the status bar, and choose *None* from the pop-up menu.

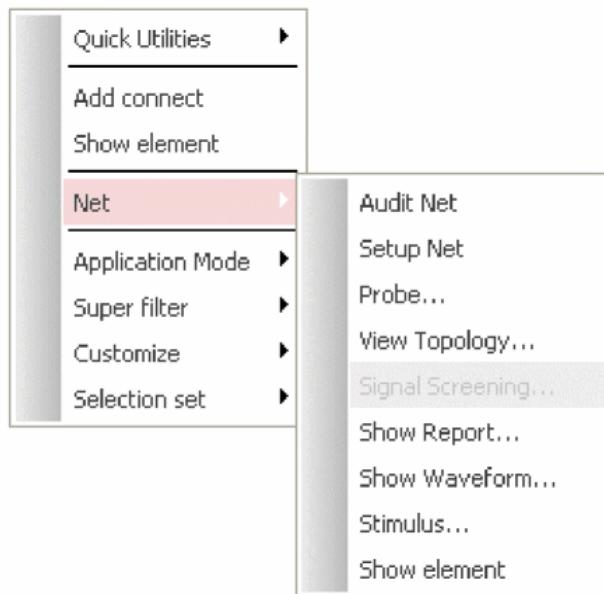
Related Topics

- [signalintegrity](#)
- [Signal Integrity Command Tasks](#)

Accessing Frequently-Used SI Commands

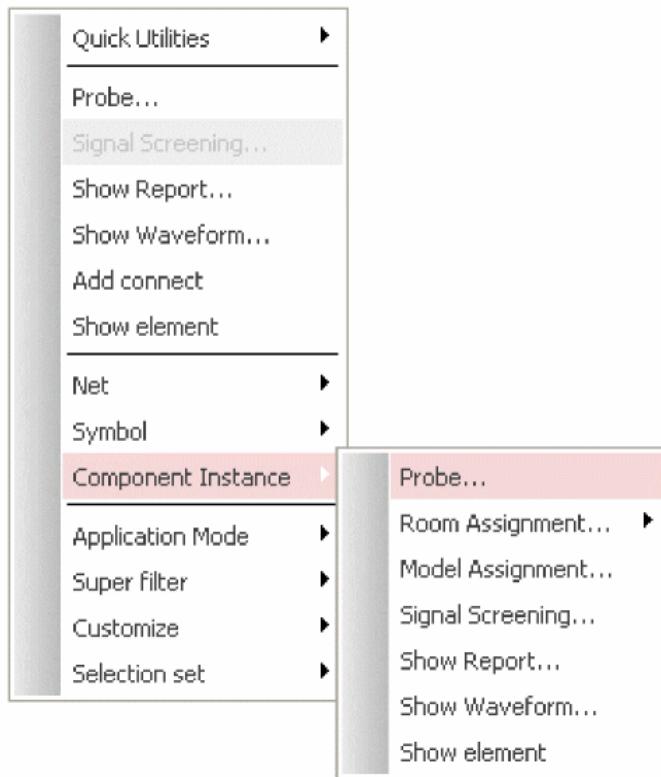
In the SI Application mode, when you right-click on an element, the content of the menu are adapted to the element to display the command associated with the element.

You can perform common tasks on nets, such as auditing and viewing topology. The following context menu commands are displayed for a net or an Xnet:



⚠ You need to select more than one net or Xnet to enable the Signal Screening command. It is grayed out when a single net is selected.

Similarly, the following context menu is displayed for a component:



⚠ The *Signal Screening* and *Show Waveform* commands act on all the nets attached to a component when launched from the Component Instance command.

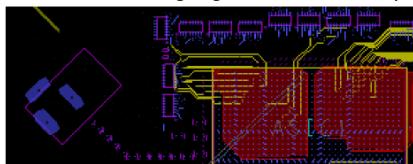
Related Topics

- [signalintegrity](#)
- [Signal Integrity Command Tasks](#)

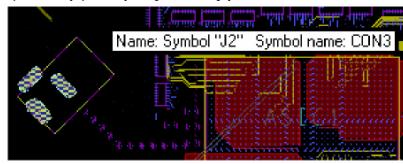
Highlighting Objects on Mouse Over

1. In the SI application mode, move the cursor over an element on the canvas.

The element is highlighted and a descriptor label (tool tip) displays its type and name as shown in the following figure:



SI Application Mode is Deactivated



SI Application Mode is Activated

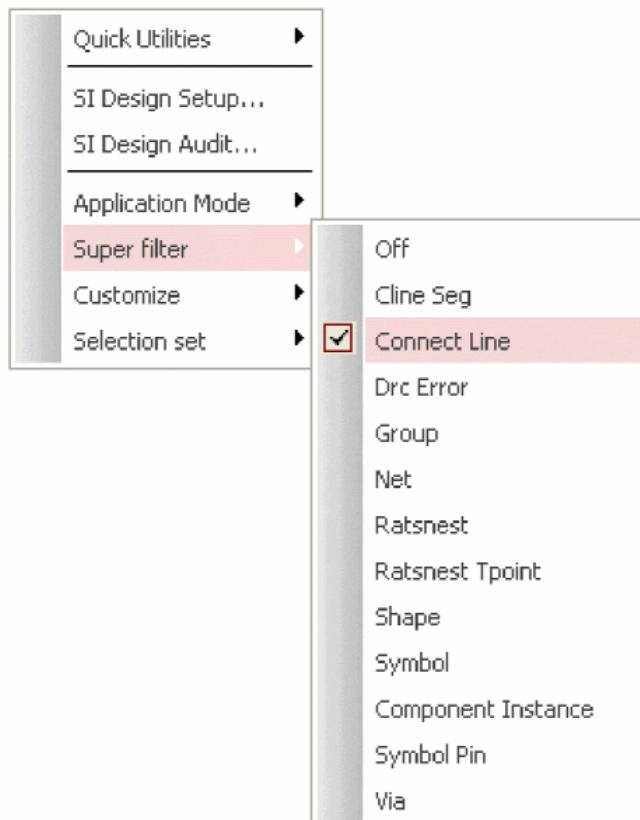
Related Topics

- [signalintegrity](#)
- [Signal Integrity Command Tasks](#)

Using Super Filter

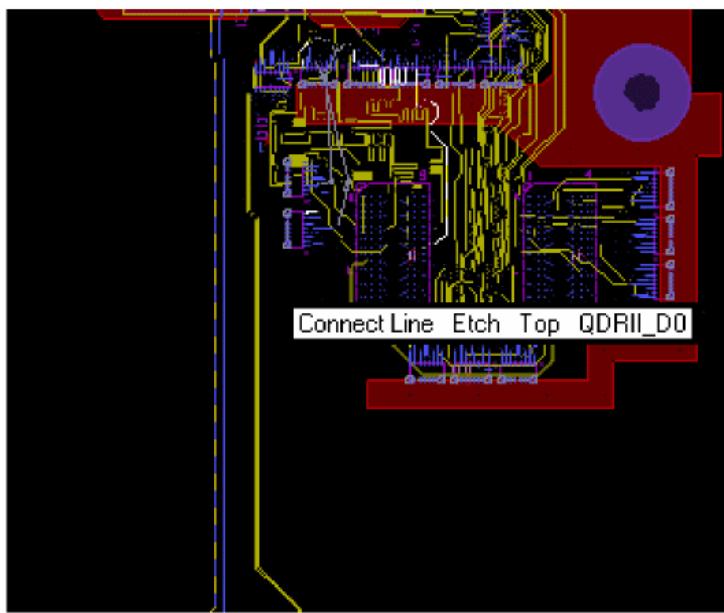
The *Super Filter* lets you quickly limit the find criteria to a single object type.

1. To access the super filter, right-click the canvas and choose *Super filter* from the pop-up menu.
2. Choose the object type you want to find in the menu.



3. Move the mouse cursor over the design.

Only the elements of the selected object type are highlighted along with a tool tip for each element.



Related Topics

- [signalintegrity](#)
- [Signal Integrity Command Tasks](#)

Customizing Layout Editor Functions

The application mode enables you to execute commands on double-clicking or dragging an element.

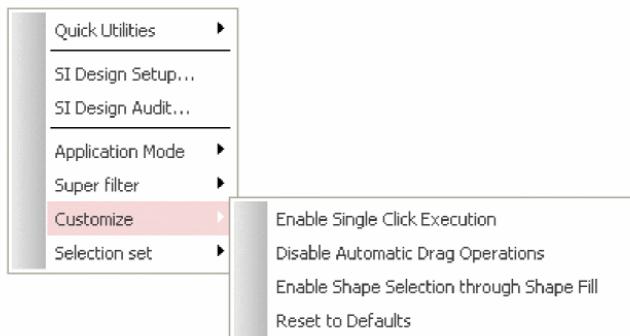
1. Change this behavior by customizing the tool.

You can customize the tool for the following:

Enabling single-click execution

Disabling automatic drag operations

Enabling shape selection through the Shape Fill command



Related Topics

- [signalintegrity](#)
- [Signal Integrity Command Tasks](#)

signal stimulus

The `signal stimulus` command is no longer in use. Use the [signal probe](#) command to launch the [Signal Analysis Dialog Box](#) dialog from where you can launch the [Analysis Waveform Generator](#) dialog to assign and/or edit stimulus values.

signal topology

The `signal topology` command is no longer in use. Use the [signal probe](#) command to launch the [Signal Analysis Dialog Box](#) dialog from where you can run the View topology command to extract the circuit topology from the board.

signal waveform

The `signal waveform` command is no longer in use. Use the [signal probe](#) command to launch the [Signal Analysis Dialog Box](#) dialog from where you can launch the [Analysis Waveform Generator](#) dialog to create and view waveforms.

signoise

The `signoise` batch command generates a crosstalk table from a batch mode simulation.

Syntax

```
signoise -x xtalktable -z impedance -m ftmode -version design-in design-out
```

-a aggressors	Use the specified aggressors for crosstalk simulations. The aggressors argument can be one of the following: Each, All.
-b meas_loc	The measurement location of the driver/receiver in a model, pin, or die. The meas_loc argument is a combination of the keywords listed below. Model: This is the default setting. The driver/receiver pin measurement location is defined by the content in the related component DML model. Pin: The pin measurement location at the external pin node. Die: The pin measurement location at the internal die node, if present. <driver>/<receiver> : combination of keywords to specify different driver/receiver pin locations. Use the one-word style (for example, Pin) if the same measurement mode will apply to both driver and receiver. Use the two-word style (for example, Pin/Die) to apply different measurement modes to driver and receiver, respectively. Note: The reports affected by receiver pin pad or die measurement locations are Reflection Summary, Delay, and Ringing. related report types are Custom and Comprehensive. Results are listed for either pin or die in the Preferred Measurement Location section at the top of the report. If taken at the pin pad, the pin pad measurement name is identical to the pin name (for example, PIN5). If taken at the die pad location, the pin name is displayed with an i appended to it (for example, Pin5i).
-c case	Use the specified case as the current case directory.
-C	Save Spice circuit files from simulations.
-d drvs	Use either the fastest driver or all drivers on the victim nets. The drvs argument is one of the following: Fastest or All.
-D ndrvs	Use either the fastest driver or all drivers on the neighboring aggressor nets. The ndrvs argument is one of the following: Fastest or All.
-e	Update the crosstalk estimates and save the design file.
-E estmode	Uses specified Fast, Typical, or Slow mode for estimating crosstalk.
-f netsfile	Restrict operation to the nets listed in the specified netsfile.
-F first_arg	The first argument for an alternate simulator. If you do not use the -S option to specify the executable, the name of the Perl script is used.
-g geomwin	Use the specified geomwin (in mils) for determining aggressor nets.
-h	Print help information.
-l length	Use the specified length (in mils) as the minimum coupled length for determining aggressor nets.
-L designlink	Use the specified designlink model for multi board analysis.
-m ftmode	Use the specified Fast, Typical, and Slow modes when simulating. The ftmode argument can be any comma-separated combination of Fast, Typical, Slow, Fast/Slow, or Slow/Fast. The default is '-m Typical.'
-M reflmeas	Use Pulse simulation or Rise/Fall simulation measurements for reflection data. The reflmeas argument is one of the following: Pulse or Rise/Fall.
-n swmode	Use the specified neighbor switching mode for crosstalk and comprehensive simulations. The swmode argument can be any comma-separated combination of Odd, Even, and Static.
-N	Produce a system connectivity file, sysconn.dat.

-o filenames	Saves reports in specified filenames. Enter filenames separated by commas. This option is valid only when you use the -r option.
-p probefile	Record SSN bounce data at the X,Y name pairs listed in probefile.
-P	Perform Plane Modeling for SSN simulations.
-r reports	Generate the specified reports. The reports argument can be any comma-separated list containing any of the following: Delay, Ringing, ReflectionSummary, Parasitics, SegmentXtalk, Xtalk, XtalkSummary, SSN, SingleNETEMISummary. The default is ‘-r ReflectionSummary’.
-s reflsims	Use the specified simulation for reflection data. The reflsims argument is one of the following: Reflection or Comprehensive. - The default is ‘-s Reflection’.
-S exec	Invokes an alternate simulator. Omit this option if the simulation is run with a Perl script specified with the -F first_arg option. The default is ‘-S Pulse.’
-t	Use timing windows for crosstalk simulations.
-v	Specify the Maximum Frequency for the driving waveform for SSN simulations when generating reports.
-w	Save waveforms for all simulations while generating reports.
-version	Prints the version.
-x xtalktable	Names the file (.xtb) to contain the crosstalk table.
-X args	Extra arguments for invoking alternate simulators. The default selection is \$CDS_ckt_file \$CDS_log_file \$CDS_wave_file. Options include: \$CDS_Proj_dir \$CDS_inst_dir \$CDS_delay_file \$CDS_cycle_file
-z impedance	Use the specified transmission line impedance value for series termination of coupled traces and line width calculations.
design_in	A .brd file or an .mcm file, to analyze for signal integrity.
design_out	If you do not specify design_out, sigNoise overwrites design_in.

Related Topics

- [signal xtalktable](#)

Generating a Crosstalk Table from a Batch Mode Simulation

1. At the operating-system prompt, enter the `signoise` command in the following format:

```
signoise [options] design_in [design_out]
```

For example, to create a ReflectionSummary report for a list of nets, type:

```
signoise -f my_nets.lst my.brd
```

Similarly, to create a CrosstalkSummary report for a list of nets, type:

```
signoise -f my_nets.lst -r XtalkSummary my.brd
```

sigxp

The `sigxp` command in Allegro PCB SI L, XL, or GXL opens the SigXplorer user interface. You use SigXplorer to create, modify, simulate, and save virtual prototypes of net topologies. You explore these topologies by modifying circuit parameters, simulating, and examining reports and waveforms. You repeat this process to tweak circuit parameters for optimum results.

You can create topologies from scratch or you can extract (derive) them from existing placed layouts for package or PCB modules. SigXplorer also supports extraction from a system of modules (multi-board topologies) interconnected through backplane or cabled connectors.

Topology files and their attributes (constraints, configuration, and so on) can be mapped (assigned) to Allegro SI nets for guiding downstream interconnect routing. You can also save Topology files and reuse them later.

You modify topology element parameters through an intuitive graphical user interface. Common circuit models for topology elements are available within the standard simulation model libraries and may be added to a topology with the Parts Browser in SigXplorer.

See the SigXplorer documentation for detailed information on SigXplorer and its associated commands.

Access Using

- Menu Path: *Tools – Topology Editor*

silkscreen audit

The `silkscreen audit` command performs the same action as the *Audit* button in the Auto Silkscreen dialog box ([silkscreen param](#)). Audit results are generated in *autosilk.log* files that contain the following information:

- Start date and time
- Design name
- Record of the silkscreen parameter values
- Messages that describe error and warning conditions

The following conditions are recorded as errors and warnings in the log file:

- If a text string cannot be moved to avoid a violation of a pad
- If silkscreen lines are not clearing pads or are too short in length

Warnings contain the coordinates and contents of text strings as well as the side of the design on which violations occur. If any database error is detected during execution of Auto Silkscreen, an error message that contains the database error code is recorded in the log file and execution stops.

Generating Audit Results

To generate audit results, follow these steps:

1. Configure the Auto Silkscreen dialog box, as described in [silkscreen param](#).
2. Type `silkscreen audit` at the command window prompt.

The `autosilk.log` file is generated.

silkscreen execute

The `silkscreen execute` command performs the same action as the *Silkscreen* button in the Auto Silkscreen dialog box ([silkscreen param](#)).

Creating Silkscreen Data

To create silkscreen data, follow these steps:

1. Configure the Auto Silkscreen dialog box, as described in [silkscreen param](#).
2. Type `silkscreen execute` at the command window prompt.

The automatic silkscreen process runs and the `autosilk.log` file is generated.

silkscreen param

The `silkscreen param` command defines the operating characteristics of the silkscreen program, letting you create composite silkscreen data on class `MANUFACTURING`, subclasses `AUTOSILK_TOP` and `AUTOSILK_BOTTOM` in your layout. It creates data for subclass `AUTOSILK_TOP` or `AUTOSILK_BOTTOM` or both, depending on how you set the Auto Silkscreen parameters.

The auto silkscreen process clears all existing data from the chosen subclasses; then, it automatically adds the silkscreen data according to the parameters that you set in the Auto Silkscreen dialog box. This process writes warnings and errors to the `autosilk.log` file.

Running automatic silkscreen on a layer enables the automatic incremental updating process that attaches silkscreen information to symbol instances.

Related Topics

- [Creating Silkscreen Data Automatically](#)

Auto Silkscreen Dialog Box

Access Using

- Menu Path: *Manufacture – Silkscreen*
- Toolbar Icon:



Use this dialog box to set the parameters for the silkscreen program.

The *Mfg Applications* tab of the Design Parameter Editor is also available for setting the parameters you define in the Auto Silkscreen dialog box. Choose *Setup – Design Parameters* ([prmed](#) command).

Layer	Specify the side of the design on which to generate the silkscreen. The options are: <i>Top</i> processes all silkscreen graphics in the subclass SILKSCREEN_TOP and creates the subclass AUTOSILK_TOP. This is the default. <i>Bottom</i> processes all silkscreen graphics in the subclass SILKSCREEN_BOTTOM and creates the subclass AUTOSILK_BOTTOM. <i>Both</i> processes all silkscreen graphics in the subclass. SILKSCREEN_TOP and SILKSCREEN_BOTTOM, and creates both AUTOSILK_TOP and AUTOSILK_BOTTOM.
Elements	Specify the type of graphics to process. The options are: <i>Lines</i> indicates to process only lines and arcs. <i>Text</i> indicates to process only text strings. <i>Both</i> indicates to process lines, arcs, and text. Only elements of the chosen type are erased. Choose the specified AUTOSILK subclass and regenerate. Unselected elements remain untouched. The default is Both.
Classes and Subclasses	Define the layout editor classes where the autosilkscreen process looks for silkscreen graphics. For each of the classes listed on the parameter dialog box the options are: <i>Any</i> uses the SILKSCREEN subclass first. <i>Silk</i> copies only graphics from the SILKSCREEN subclass. <i>None</i> takes nothing from the class. ⚠ If no silkscreen graphics exist, the program uses the ASSEMBLY subclass. If it still finds nothing, it uses the DISPLAY subclass. The default value for each of the parameters listed is <i>Silk</i>.
Text	
Rotation:	Sets the text rotation to 0, 90, 180, or 270, which define the legal rotations for any text string on an AUTOSILK subclass. The default values for each of these field options is checked. If the autosilkscreen process cannot find a location for a text string that avoids a hole at any of the allowed rotations, the text is placed on the AUTOSILK subclass at its original location and rotation, and a message is written to the log file.
Allow Under Component	Specifies whether silkscreen text may be positioned under a component that exists on the same side of the design as the one being processed. Text is considered to be under a component if it falls within the extents of the component's graphics on the PACKAGE GEOMETRY/PART GEOMETRY class, ASSEMBLY subclass. Enabled by default.
Lock autosilk text for incremental updates	During incremental autosilk updating, the AUTOSILK subclass text will not be moved. You can still edit the text, including moving it. Incremental updates automatically occur when you make changes (such as moving a symbol). This lock option disables dynamic silkscreen from moving the text when you invoke autosilk via this dialog box. This setting is preserved in the database and always respected when an automation command is used. For example, if you do not change the settings and it is not selected (default) and later run Update Symbol it will be automatically updated irrespective of the setting of the Update Symbol command's <i>Reset symbol text location and size</i> setting.
Detailed Text Checking	Enabled by default. Considers each stroke for each character as a line segment, where the line segment itself is checked for potential obstacles. For instance, if the character 'O' is large enough, a pad may potentially lie in its interior, or it may nestle in the crook of the character 'L'. Otherwise, silkscreen text is checked using the bounding box for the text. The box expands to accommodate the descenders of lower case characters, whether the string actually has lower case characters or not.
Maximum Displacement	Specifies in user units the maximum distance in any direction that silkscreen text strings can be moved to avoid intersecting with a pad. The default value is 100 mils.

<i>Displacement Increment</i>	Specifies in user units the distance increment to use when looking for a location to which a silkscreen text string is moved. This is bounded by the area defined by <i>Maximum Displacement</i> . The default value is 35 mils. The combination of a smaller displacement increment with a much larger maximum displacement can severely impact performance when searching for a location to which to move text, because the maximum number of points that may be tested increases. For example, failure to find a location using the defaults of 100 and 35 does not result in that many tests for a text string. If you change the increment to 1 mil, and a location still was not found, the number of tests that failed would be at least an order of magnitude more.
<i>Minimum Line Length</i>	Specifies the minimum length of any line or arc segment that is allowed on an AUTOSILK subclass. The process of trimming lines and arcs around pads can often produce a number of very small segments. If any of these segments is shorter than the value specified as minimum line length, they are discarded. The default value is 0, which means that no segment is discarded.
<i>Element to Pad Clearance</i>	Specifies in user units the amount of space that is to be left between silkscreen elements and the edges of pads on that side of the design. The default value is 0, which allows silkscreen elements to touch the pad edge. Using negative values allows silkscreen elements to intersect pads by the specified value.
<i>Clear solder mask pad</i>	Specifies that silkscreen elements do not contact pad areas defined for masking and conductive pad geometries are ignored.
<i>Silkscreen</i>	Runs the process, based on the parameters you set.
<i>Close</i>	Saves the settings and closes the dialog box.
<i>Audit</i>	Audits the board based on the parameters you set and generates the <i>autosilk.log</i> file (see silkscreen audit)

Creating Silkscreen Data Automatically

To automatically create silkscreen data, perform these steps:

1. Run [silkscreen param](#).
2. Set the parameters on the Auto Silkscreen dialog box, as described above.
3. Click *Silkscreen*.

The Auto Silkscreen dialog box closes. While the process runs, each silkscreen element highlights after processing. When the process finishes, the following message appears in the console command window:

Auto Silkscreen Finished

Related Topics

- [silkscreen param](#)

skill

The `skill` command lets you put your user interface into SKILL mode. When you run this command, the prompt in the command window is reconfigured from `Command:` to `skill:`

AXL-SKILL is a language processor contained in the layout editor. It contains and is an extension of the core Cadence SKILL language. You use AXL-SKILL functions to access the design database and its display and user interfaces. Once you have accessed the database, you can process the data using the core SKILL functions. See the core SKILL user guide and reference documents included in Cadence documentation and on Cadence Online Support (support.cadence.com) for details.

You access AXL-SKILL by entering the command `skill` on the command line.

 AXL-SKILL is only available in tiers of Allegro PCB Performance option L and higher.

Executing any AXL-SKILL Function on the Command Line

There are two ways to enter into SKILL mode:

1. Type `skill` in the command window prompt.

The `skill` command passes the rest of the command line arguments to the AXL-SKILL processor, which executes that single function.

If you enter the `skill` command with no arguments, the AXL-SKILL interpreter replaces the shell interpreter (that is, enters SKILL mode) and displays the prompt `skill>` on the console command line. AXL-SKILL then interprets any subsequent entries you make as AXL-SKILL functions.

2. Enter `exit` to end skill mode.

Or

- Type `skill <skill command>` in the command prompt window.

The `skill` command lets you enter the SKILL functions as arguments to the `skill` command.

slide

The `slide` command moves cline segments on fully or partially routed nets. It can be used on single nets, differential pairs, or a group of routed connections. Push and shove controls can be used to aggressively shift adjacent traces and vias. Heads-up feedback is provided when nets with electrical rules are chosen.

- In the pre-selection use model, dragging an eligible element automatically launches the command if you enabled automatic drag operations from the *Customize* right mouse button menu.

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 are:

- Cline Segments
- Vias
- Rat Ts

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

- *Design Parameters* to access the *Design Parameter Editor* when you need to change several common parameters that apply to etch edit mode. Changing a parameter here automatically updates its value on the *Options* panel as well. Choose *Setup – Design Parameters* (`prmed` command).
- *Options* to display all parameters relevant to the command when you need to quickly change one parameter. Changing a parameter here automatically updates its value on the *Route* tab of the *Design Parameter Editor* as well.

Related Topics

- [Sliding Connect Line Segments](#)
- [Sliding Vias or Rat Ts](#)

Slide Command: Options Panel

Access Using

- Menu Path: *Route – Slide*



- Toolbar Icon:
- When you edit or move connections using `slide`, the following fields appear in the *Options* panel of the Control Panel to determine how connections slide around obstacles and whether to move vias with line segments.

Active Etch Subclass	Controls selection and visibility of the active etch/conductor subclass. If you change the active subclass, its color moves to the top of the layer priority: the layout editor draws items of that color on top of other colors. Layer priority, in turn, affects selection priority. If your pick is equally close to two segments of different colors, the program chooses the segment drawn on the active segment over the other segment. On startup, the color of the active subclass moves to the top of the layer priority. If the active class is not ETCH, the active class/subclass is first changed to ETCH/TOP.
Min Corner Size	Specifies the minimum 45 degrees corner size allowed between two non-parallel clines. The default value is <code>1 x width</code> .
Min Arc Radius	Specifies the minimum arc size allowed between two clines. The default value is <code>1 x width</code> .
Vertex Action	Controls the vertex selection between two clines. The choices are: Line Corner : Add corner segment between the two segments that meet at the vertex. The corner size will vary with cursor position and with <i>Min Corner Size</i> . Arc Corner : Add corner segment between the two segments that meet at the vertex. The corner size will vary with cursor position and with <i>Min Arc Size</i> . Move : Vertex is moved with the selected segments. Edit : Edits the vertex similar to <code>vertex</code> (<i>Edit – Vertex</i>) command. None : Prevents any special action when a vertex is selected.
New Seg Angle	Specifies the angle when a cline changes direction or moves around an obstacle. By default, this option is not visible in the Options panel. You can enable by choosing <i>Setup – Design Parameters</i> (<code>prmed</code> command). The choices are: Default : Creates a new segment at 45 degree with the original segment. For octilinear routing, the new segment will be diagonal or orthogonal. For non-octilinear routing the new segment will be non-octilinear. 45 : Creates new segments that will be octilinear with the original segment. 90 : Creates new segments that will be orthogonal with the original segment. If this option is selected <i>Min Corner size</i> and <i>Min Arc size</i> options are disabled.
Bubble	Controls automatic bubbling (moving of existing connections) to resolve DRC errors. Enabling either of the hug modes or shove-preferred bubble mode sets the <i>Line lock</i> field to <i>Line</i> to prevent you from adding arcs while in shove- or hug-preferred mode. Bubble mode does not support arcs. The choices are: Off : Flags all clearance violations with error markers. Hug Only : Where possible, the routed cline contours other etch/conductor elements to avoid spacing DCRs. Other etch/conductor remains unchanged. Hug Preferred : Where possible, the routed cline contours other etch/conductor elements to avoid spacing DRCs. If not possible, the layout editor tries shoving other etch/conductor elements to open routing paths. Note : This method is more aggressive than <i>Hug Only</i> . Shove Preferred: Where possible, the routed cline pushes and shoves other etch/conductor elements to avoid spacing DRCs. If not possible, the layout editor tries hugging other etch/conductor elements.
Shove vias	Allows the bubble functionality in shove mode to move vias when you are editing etch. It is only active when the bubble functionality is enabled. Choices are: Full : Vias are shoved in a <i>shove-preferred</i> manner. Any new or edited etch always shoves vias out of the way. Minimal : Vias are shoved in a <i>hug-preferred</i> manner. Vias are not moved unless there is no way to draw a connect line around them. Off : Vias are not shoved.
Clip dangling clines	This option clips dangling clines that are too close (violate spacing constraints) to any line segments you are editing. It is active only when bubble functionality is enabled in shove mode.
Smooth	Controls whether smoothing occurs on the cline to minimize segments between the start and finish points. Smoothing occurs dynamically as you move the mouse on cline segments close to the segment you chose. Only segments changed by sliding or bubble options are smoothed. Performance with the Smooth option active may be somewhat slower than when it is inactive. The choices are: Minimal : Executes dynamic smoothing to minimize unnecessary segments. Full : Executes more extensive smoothing to remove any unnecessary jogs. An additional segment on each end of the changed segments can be included. Off : Choose to disable smoothing.

<i>Allow DRCs</i>	Specifies that the layout editor can violate design rules to make the etch edit. The violations are flagged with DRC markers; you must resolve the violations for a successful design. If <i>Allow DRCs</i> is disabled and DRCs already exist on the trace or in a group of traces that you have chosen for routing, or if the layout editor determines that DRCs are introduced to the design, the layout editor does not slide the connection.
<i>Gridless</i>	Specifies whether the connect line or via you slide has to adhere to the routing grid. When you enable gridless routing, the layout editor can slide connections at maximum density while accommodating varying design rules and line widths. This affects your connections only if bubble is active or if <i>Allow DRCs</i> is disabled.
<i>Auto Join (hold Ctrl to toggle)</i>	Controls the behavior when parallel cline meets. When enable this option joins parallel clines. When disable this option creates new cline segments to connect parallel clines. By default this option is <i>On</i> . Hold the <code>Ctrl</code> key to get the alternate behavior of <i>Auto Join</i> during a single edit, without changing the settings in the <i>Option</i> form.
<i>Extend Selection (hold Shift to toggle)</i>	Choose to enable the extended selection to include two cline segments adjacent to the original selection. By default this option is <i>Off</i> . Hold the <code>Shift</code> key to get the alternate behavior of <i>Extend Selection</i> during a single edit, without changing the settings in the <i>Option</i> form. The choices are: Segments: Extends selection to adjacent segments. This is the default option. Vias: Extends selection to adjacent vias. Segments and Vias: Extends selection to both segments and vias.

Related Topics

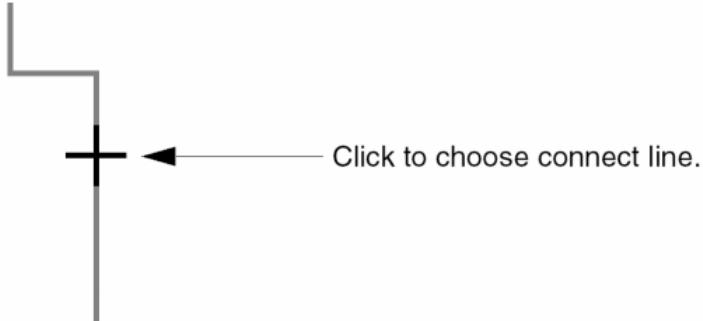
- [Sliding Vias or Rat Ts](#)

Sliding Connect Line Segments

Perform the following steps to slide connect line segments:

1. Set the *Active* layer in the *Options* panel or right-click and choose *Change Active Layer*.
2. Hover your cursor over a cline segment you want to move or window select a group of them. The tool highlights it and a datatip identifies its name.
3. Right-click and choose *Slide* from the pop-up menu or begin dragging the element to automatically launch the command. The tool identifies the cline segment name in the *Options* panel, and the net name and active subclass also appear in the two panes of the status bar, to the left of the current mouse coordinates.

 The point at which the connection begins is from the location at which you right-click.



4. Move the cline segment to its new location.
5. Click to secure the segment in the new position.
6. Continue moving any additional segments. The command terminates once you click and secure the final segment in its new position.

Related Topics

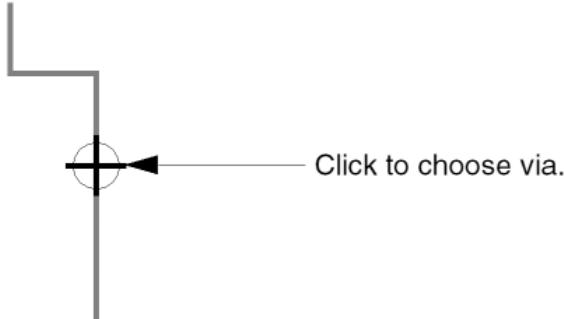
- [silkscreen param](#)

Sliding Vias or Rat Ts

Follow these steps to slide vias or rat Ts:

1. Hover your cursor on the via or Rat T that you want to move. The tool highlights it and a datatip identifies its name.
2. Right-click and choose *Slide* from the pop-up menu or begin dragging the element to automatically launch the command. The tool identifies the element name in the *Options* panel.

 To move more than one via or Rat T, window select them.



3. Move the cursor to the location where you want to move the via or Rat T.
4. Click to secure the via or Rat T in the new position.
Based on the selections you have in the *Options* panel (checking *Vias with segments* and/or *Ts with segments*), the connections route to the new position.
5. Continue moving any additional vias or Rat Ts. The command terminates once you click and secure the final element in its new position.

Related Topics

- [silkscreen param](#)
- [Slide Command: Options Panel](#)

smi message detail

The `smi message detail` command displays the extended description for the meaning of an error message.

Help Message Detail Dialog Box

Access Using

- Menu Path: *Help – Message Detail*

<i>Message Identifier</i>	Enter the <module name>-<message number from the message window.
<i>View Message Detail</i>	Displays the extended description of the message will appear in a log file viewer utility window
<i>Close</i>	Closes the dialog box.

snap_rat_t

The `snap_rat_t` command performs the same function as the *Snap Rat T* option in the right button pop-up menu. You use `snap_rat_t` when you want to route a connection on a net scheduled with T points (see [net schedule](#) for details on inserting T points). The command/option allows you move an unconnectedTpoint to another location in order to complete a connection.

Routing a Connection on a Net Scheduled with T Points

This procedure describes the use of `snap_rat_t` in the `add connect` command.

1. Run `add connect`.
2. Choose the active etch/conductor subclass in the *Options* panel.
3. Choose a pin or rat T as the starting point for the trace.

The element is highlighted. If the element is a connect line that the command determines is connected to a pin or to a rat T, ratsnests from the pin or rat T are used to find a destination element that is not already connected. (If the element is neither a pin or rat T, a destination element is not chosen.) The chosen destination element is normally the one closest to the starting element. If all connections from the starting element are already complete, no destination element is chosen.

If the net has a timing constraint, you are provided with feedback on the current etch/conductor length. If you do not have a timing constraint attached to the net but you have etch/conductor length enabled, simple etch/conductor length feedback is provided.

4. Click on each element that you want to route.

If your pick completes the connection to the destination:

- The rubber band lines and ratsnest line disappears
- Add connect reverts to its initial state

If your pick is a rat T that does not complete the connection, you can choose *Snap Rat T* from the right-button pop-up to move the rat T to your last pick location, completing the connection to the destination (or type `snap_rat_t` at the command window prompt).

 As is the case with any routing process, you can use `show element` to get information on elements that are currently highlighted. Running `show element` does not terminate `add connect`.

5. When the connection is complete, click *Done* from the right-button pop-up to terminate the command.

soft net

The `soft net` command lets the master designer assign certain nets in a partition as *soft*. Then the specified 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.

Related Topics

- [Assigning Soft Nets to a Specified Partition](#)

Soft Net Partition Assignment Dialog Box

Access Using

- Menu Path: *Place – Design Partition – Soft Net Assignment*

The dialog box is read only for exported partitions and partition designers.

<i>Soft Nets' Owner</i>	Lists the existing partition names to which the master designer may assign soft nets.
<i>Available Nets</i>	Displays a list of nets in the design.
<i>Name Filter</i>	Lets you narrow or filter the list of net names by typing in names, parts of names, and using wild cards. Use the * wildcard to enter a partial string; for example, Signal_2*.
<i>Net Filter</i>	<p>Limits the nets displayed:</p> <ul style="list-style-type: none"> • <i>All Pins Inside Partition</i> – Lists the net names whose pins are all within the XY boundaries of the specified partition; however, clines may be outside the specified partition. • <i>All Pins Outside Partition</i> – Lists the net names whose pins are all outside the XY boundaries of the specified partition; however, clines may be inside the specified partition. • <i>Pins Across Partition</i> (default filter) – Lists the net names of which at least one pin per net is inside the XY boundaries of the specified partition; however, other pins of the net are outside the XY boundaries of the specified partition. • <i>All Nets</i> – Lists all the nets in the design.
<i>All -></i>	Lets you move all the <i>Available Nets</i> into the <i>Selected Nets</i> list.
<i><-All</i>	Lets you move all the <i>Selected Nets</i> into the <i>Available Nets</i> list.
<i>Selected Nets</i>	Lists the soft nets assigned to a particular partition owner.
<i>OK</i>	Saves the settings and closes the dialog box.
<i>Cancel</i>	Closes the dialog box.
<i>Apply</i>	Saves the settings and leaves the dialog box open.
<i>Help</i>	Displays help for this command.

Assigning Soft Nets to a Specified Partition

Perform the following steps to assign soft nets to a specified partition:

1. Run the *soft net* command after you create the partitions, but before you export them to the partition designers.
The Soft Net Partition Assignment dialog box appears.
2. In the *Soft Nets' Owner* field, click the drop-down menu and choose a partition.
3. In the *Name filter* field, filter the list of net names by typing in names, parts of names, and using wild cards. Use the * wildcard to enter a partial string; for example, Signal_7* or leave the asterisk (*) in place to display the total list of nets.
4. In the *Net filter* field, click the drop-down menu to choose one of these options for filtering:
 - *All Nets*
 - *All Pins in Partition*
 - *All Pins Outside Partition*
 - *Across Partition*
5. Once you are satisfied with the filtered list, click *All ->* to move the specified net names to the *Selected Nets* section.
You can also double click a net in the *Available Nets* pane to move it to the *Selected Nets* section, or you can pick a net in the Design Window and it appears in the *Selected Nets* pane.
6. Click *Apply* and then *OK*.
The Soft Net Partition Assignment dialog box closes.
7. Run the *workflow* command to export the specified partition to the partition designer.
The soft nets are highlighted in the owner's specified partition, but are dimmed in all other partitions.

Related Topics

- [soft net](#)

source

The `source` command reads a file. This command is typically used by your local environment file to read the global environment file. The `source` command can be nested up to four levels.

Syntax

```
source <filename>
```

Example

```
source enved
```

spdif clarity3dlayout

The `spdif clarity3dlayout` command runs Clarity3DLayout.

Access Using

- Menu Path: *Analyze – Clarity3DLayout*

Using the SPDIF Clarity3DLayout Command

To run Clarity3DLayout:

1. Choose *Analyze – Clarity3DLayout*
2. Select the XNets to analyze in the XNet Selection window.
You can choose to analyze all XNets in the design (*Entire Design*) or analyze only a selected set of XNets (*All Selected*).
3. Click *Apply*.
4. Click *OK* to close the window.

specctra

The `specctra` command generates routing files from your database and launches Allegro PCB Router from your user interface to autoroute your design. For detailed information on Allegro PCB Router, see your product documentation.

Access Using

- Menu Path: *Route – Route Editor*
- Menu Path for APD L: *Route – Router*

Generating Routing Files

You can generate routing files by performing these steps:

1. Choose *Route – Route Editor*.

When you run this command, the following actions occur:

- The layout editor writes Design, Rules, and Forget files from the current database.
- The Allegro PCB Router user interface opens.
- The generated design and rules file are read into Allegro PCB Router.

 If you plan to use only the original generated .do files, skip to step 4.

2. Open a text editor to review and edit the `rules.do` file. Cadence recommends that you do the following when editing any .do files:
 - a. Copy the generated `rules.do` file to a different file name.
 - b. Edit the renamed file.
3. Load the forget file and the renamed .do file(s) into Allegro PCB Router and perform an initial route of the design.
4. If the initial route is completed to satisfaction, load the forget file and the original .do file(s).
5. Issue the check command to verify any design rule violations.
6. If you are satisfied with the results, load the original files back into the layout editor.

Automatic Router Parameters Dialog Box

The Automatic Router Parameters dialog box lets you set parameters for various routing actions. Based on the menu path you used to access this dialog box, the correct tab in the Automatic Router Parameters dialog box is displayed when the box opens. The default is the *Fanout* tab.

The individual tabs in the Automatic Router Parameters dialog box are similar to the dialog boxes of the same names in Allegro PCB Router XL. For more details about Allegro PCB Router XL, see your product documentation.

Access Using

Menu Path: Available when you right-click to display a pop-up menu and choose Setup after choosing any of these menu items:

- *Route – Fanout By Pick*
- *Route – Net(s) By Pick*
- *Route – Elongation By Pick*
- *Route – Miter By Pick*
- *Route – UnMiter By Pick*

You can also access the Automatic Router Parameters dialog box through the Automatic Router dialog box when you choose *Route – Route Automatic*.

Fanout Tab

Routes short pin escape wires from pins to vias. Lets you control pin and via sharing, specify the layer depth, control the escape direction, and set a temporary grid.

<i>Direction</i>	Specifies the fanout routing direction from the pins. Directs the autorouter to escape wires and vias inward from the component pins (In), outward from the component pins (Out), or either way (Either). The default is <i>Either</i> .
<i>Via Location</i>	Directs fanout to escape wires and vias inside the component outline, outside the component outline, or anyway relative to the component outline. The default is <i>Anywhere</i> .
	You can use this option with the direction option to locate vias relative to both the component pins and the component outline. It may be most useful when the component outline extends far beyond the pins.
<i>Maximum Fanout Length</i>	Specifies a maximum length entered in the field at the right.
<i>Enable Radial Wires</i>	Allows fanouts on radial wires (Allegro Package products).
<i>Fanout Grid</i>	<i>Allows the autorouter to automatically calculate initial via grids that permit one wire or two wires between adjacent vias. This depends on the grid choice specified in this section.</i>
<i>Current Via Grid</i>	Uses the current via grid defined in the <i>Router Setup</i> tab of the Automatic Router dialog box.
<i>1 Wire Between Vias</i>	Sets a via grid that allows one wire between adjacent vias.
<i>2 Wire Between Vias</i>	Sets a via grid that allows two wires between adjacent vias.
<i>Specified Grid</i>	Specifies the uniform X, Y fanout via grid.

Blind/BuriedVia Depth

<i>Fanout Blind/Buried Vias To</i>	When enabled (checked), controls both direction and depth of the routing for blind and buried vias according to the following options
<i>Top</i>	Sets the fanout direction toward the front or top side. This is the default.
<i>Bottom</i>	Sets the fanout direction toward the back or bottom side.
<i>Opposite Side</i>	Sets the fanout direction toward the opposite side of the design.
<i>Max. Layer Span</i>	Sets the fanout depth to span of the maximum number of signal layers indicated. The default is 2.

Pin Types

<i>All</i>	Specifies all pin types.
<i>Specified</i>	Indicates specific pin types. Choose the pin types you want. Power Nets and Signal Nets are chosen by default. Unused Pins lets you specify All or Exclude Thru-Pins. Power Nets—selects all pins that have power nets assigned Signal Nets—selects all pins that have signal nets assigned and interconnect with one or more other pins. Single Pin Nets—selects all single pin signal nets. Unused—selects all pins, including SMD pads and through-pins that have none assigned. Unused pins are collected into a net called <i>UNUSED_PINS</i> .

Sharing

<i>Share Within Distance</i>	Sets the maximum distance between sharing.
<i>Share Pins</i>	Allows fanouts to escape to through-pins on the same net.
<i>Max Share Count</i>	Specifies a value to limit the number of connections that can attach to shared pins. Available when you choose <i>Share Pins</i> .
<i>Share SMD's on Way to Via</i>	Allows fanouts to connect SMD's on the same net before escaping to a via.
<i>Max Share Count</i>	Specifies a value to limit the number of SMDs that can be connected before escaping to a via. Available when you choose <i>Share SMD's on Way to Via</i> .
<i>Share Vias</i>	Allows fanouts to share vias between SMDs on the same net.
<i>Max Share Count</i>	Specifies a value to limit the number of connections that can attach to shared pins. Available when you choose <i>Share Vias</i> .

Bus Routing Tab

Routes component pins that share the same, or nearly the same, X or Y coordinate.

This command uses a special algorithm that routes regular arrays of pins that share the same, or nearly the same, X or Y location. The autorouter determines which connections are candidates for bus routing and routes them. Clearance rules must permit sufficient space to allow bus routing without conflicts. In cases where pins on the same net are slightly offset from one another in the X or Y direction, the autorouter creates non-orthogonal connections (slanted routes).

<i>Diagonal Routing</i>	Routes with a diagonal line. This option provides the highest routing density.
<i>Orthogonal Routing</i>	Routes buses orthogonally.

Seed Vias Tab

Breaks a single connection into two shorter connections by adding a via.

Before using this command, define at least one through-via that extends through all signal layers. This command adds a single via at a corner of the bounding rectangle for each connection that satisfies the length criteria.

<i>Break-up Connections Longer Than</i>	Accepts a value. The router adds a via for each connection that exceeds the specified length in both the X and Y directions. The value is in design units.
<i>Place Vias Under SMD Components</i>	Allows the router to place seed vias under SMD components on designs with only two signal layers.

Testpoint Tab

Assigns test points to signal nets.

When using this tab, it improves PCB testability by adding test points to routed signal nets. The perimeter of each component image is used as a boundary to restrict vias to locations outside the component bodies. Test points are through-pins, vias, or single layer shapes.

Testable vias are always exposed on the probing layer. Exposed means that the via is not covered by a component body. The probing layer can be front, back, or both.

This tab is not available in Allegro X Advanced Package DesignerL.

<i>Testpoint Side</i>	Determines the side on which you want the test points. The default is Both.
-----------------------	---

Testpoint Position

<i>Center to Center Spacing</i>	Specifies the minimum center-to-center distance between test points.
<i>Center To Component Spacing</i>	Specifies the minimum distance from the test point to the edge (boundary) of the component.
<i>Component Outline Clearance</i>	Specifies the minimum distance between the edge of the test point and the edge of the component.
<i>Testpoint X Grid</i>	Specifies the X grid.
<i>Testpoint Y Grid</i>	Specifies the Y grid.
<i>Maximum Length</i>	Specifies the maximum length between a test point and the center of a pad.

Pin Use

<i>Allow Pin Use</i>	Lets you turn on By Component. Specify a filter if necessary. Highlight any components and click the right horizontal arrow to move them into the Selected Components box. To deselect a component, highlight the component in the Selected Components box and click the left horizontal arrow.
<i>Via Padstacks</i>	Lets you turn on Specify Testpoint Vias. Specify a filter if necessary. Highlight any padstacks and click the right horizontal arrow to move them into the Selected Padstacks box. To deselect a padstack, highlight the padstack in the Selected Padstacks box and click the left horizontal arrow.

Custom Smooth Tab

This tab is only available in Allegro X Advanced Package DesignerL. For a description of these settings, see *Route – Custom Smooth* (*custom smooth* command).

Spread Wires Tab

Adds extra space between wires, and between wires and pins. the layout editor adds extra wire-to-wire, wire-to-SMD pad, and wire-to-pin clearances to improve PCB manufacturability. Extra clearances are created by moving wires without moving or adding vias.

<i>General</i>	Repositions wires to increase the clearance between wires and pins, SMDs and adjacent wires. Enter the starting and ending extra clearance values. This does not move vias.
----------------	---

Specified	Defines values for the specific spread wire options you want to use. Enter the starting and ending extra clearance values.
-----------	--

Miter Corners Tab

Lets you change 90-degree wire corners to 45 degrees for wires exiting pins and vias.

Miter Passes	<i>The number of times the layout editor performs mitering. The default is 4.</i>
Miter Pin and Via Exits	Changes 90-degree wire corners to 45 degrees for wires exiting pins and vias.
Slant Wrong-way Segments	Replaces segments running against the specified routing direction on a layer with 45-degree segments
Miter T Junctions	Miters wires meeting at a T junction.
Miter at Bends	Changes 90-degree wire corners to 45 degrees.

Elongate Tab

Increases etch/conductor length to adhere to timing rules.

Meander	Enables or disables a non-optimal wiring pattern that meanders between pins in a connection. The autorouter can use a meandering pattern to add length to a connection to meet minimum routing length requirements, while preserving routing area that might otherwise be used up with alternative elongation patterns.
Trombone	Enables or disables an elongation wiring pattern that folds back against itself, resembling the slide of a trombone. Options are: Minimum Gap—Specifies the spacing between etch/conductor when the autorouter uses accordion, sawtooth or trombone elongation patterns to follow a minimum length rule. The value must be correctly scaled for the current measurement units. Consider the current measurement unit when you enter a dimensional value. Maximum Run Length—Specifies the maximum length of a routed connection when the autorouter uses the trombone elongation pattern. The value must be correctly scaled for the current measurement units. Consider the current measurement unit when you enter a dimensional value.
Accordion	Enables or disables an elongation wiring pattern that runs in rectangular steps, resembling an accordion fold. Options are: Minimum Gap—Specifies the spacing between etch/conductor when the autorouter uses accordion, sawtooth or trombone elongation patterns to follow a minimum length rule. The value must be correctly scaled for the current measurement units. Consider the current measurement unit when you enter a dimensional value. Minimum/Maximum Amplitude—Specifies the bend height when the autorouter uses accordion or sawtooth elongation patterns to follow a minimum length rule. The values must be correctly scaled for the current measurement units. Use <i>Minimum Amplitude</i> to control the minimum height. This is a way to avoid very small bends. When <i>Minimum Amplitude</i> is unspecified (set to -1), the default minimum bend height is the greater of three times the wire width or one wire width plus one wire-wire clearance. When <i>Minimum Amplitude</i> and <i>Maximum Amplitude</i> are set to 0, the router is limited to the trombone pattern. The accordion pattern is not allowed. Consider the current measurement unit when you enter a dimensional value.
Sawtooth	Specifies an elongation wiring pattern that runs in a diagonal pattern, resembling the teeth of a saw blade. Options are: <i>Minimum Gap</i> —Specifies the spacing between etch/conductor when the autorouter uses accordion, sawtooth, or trombone elongation patterns to follow a minimum length rule. The value must be correctly scaled for the current measurement units. When you enter a dimensional value, remember to consider the current measurement unit. <i>Minimum/Maximum Amplitude</i> —Specifies the bend height when the autorouter uses accordion or sawtooth elongation patterns to follow a minimum length rule. The values must be correctly scaled for the current measurement units. Use <i>Minimum Amplitude</i> to control the minimum height. This is a way to avoid very small bends. When <i>Minimum Amplitude</i> is unspecified (set to -1), the default minimum bend height is the greater of three times the wire width or one wire width plus one wire-wire clearance. When <i>Minimum Amplitude</i> and <i>Maximum Amplitude</i> are set to 0, the router is limited to the trombone pattern. The accordion pattern is not allowed. When you enter a dimensional value, consider the current measurement unit.
Pattern Stacking	<i>Controls whether the autorouter can add elongation patterns (accordion, trombone, sawtooth) to a wire segment of an existing wire pattern. This condition only applies at the PCB level.</i>
Miter Corners	Adds small 45-degree corners on elongations using one of these patterns: accordion, trombone, and sawtooth.

specctra_in

The `specctra_in` command translates and imports data from a Allegro PCB Router XL session `.ses` file to design file.

Use this command to update your design database after placing and/or routing in Allegro PCB Router XL. You import the Allegro PCB Router XL `.ses` file.

You can also import a design database that you previously exported to Allegro PCB Router XL into a new design.

Set up any constraints and properties in the layout editor. The translator handles any definitions in Allegro PCB Router XL. Any special constraints defined in Allegro PCB Router do not get passed back to your design at this time.

For information on mapping properties to Allegro PCB Router, see *Mapping Allegro X PCB Editor Properties, Assignment Tables, and Rule Sets* in your product documentation.

Warning: Do not change the design file that you are using between exporting the layout editor data and importing the updated data from Allegro PCB Router.

Related Topics

- [Importing Data from Allegro PCB Router to a Design](#)

Import from Auto-Router Dialog Box

Access Using

- Menu Path: *File – Import – Router*

Use this dialog box to import placed and routed data from Allegro PCB Router.

<i>Auto-Router session</i>	Enter the .ses filename containing the Allegro PCB Router data. Use the <i>Browse</i> button to locate the file.
----------------------------	--

Importing Data from Allegro PCB Router to a Design

Follow these steps to import data from Allegro PCB Router to a design:

1. Run [dbdoctor](#).

⚠️ Do not change the Allegro X PCB Editor board or Allegro X Advanced Package DesignerL design during the time between exporting to and importing from Allegro PCB Router.
2. Run `specctra_in` to display the Import from Auto-Router dialog box.
3. Enter or browse for the file you want to import data from, in the *Session* box. This file has the `.ses` extension.
4. Click *Run* to translate the Allegro PCB Router data.
Informational messages about the status of the translation appear at the bottom of the dialog box.
5. Click *Close*.

Related Topics

- [specctra_in](#)

specctra_out

The `specctra_out` command exports data from your design database for use in Allegro PCB Router. The translation creates a `.dsn` file which automatically takes the name of the current layout editor file, unless you specify differently.

The FIXED property in your design translates to NO ROUTE and FIXED in Allegro PCB Router.

The translation protects any preexisting etch/conductor in the design database. You can "unprotect" any or all of the etch/conductors from within Allegro PCB Router.

Related Topics

- [Exporting Data from Design to Allegro PCB Router](#)

Export to Auto-Router Dialog Box

Access Using

- Menu Path: *File – Export – Auto-Router*

Use this dialog box to translate design data to a format that can be used in Allegro PCB Router.

<i>Auto-Router Design</i>	Enter the <code>.dsn</code> file name for the data you are exporting. By default, this field indicates the current layout editor design name. Use the <i>Browse</i> button to locate a pre-existing file.
---------------------------	--

Exporting Data from Design to Allegro PCB Router

Follow these steps to export data from a design to Allegro PCB Router:

1. Run [dbdoctor](#).

 Do not change the design file that you are using during the time between exporting the design data and importing the updated data from Allegro PCB Router.

2. Run `specctra_out` to display the Export to Auto-Router dialog box.
3. Enter or browse for the file you want to export data to, in the Auto-Router *Design* box.
The current path and design file is entered by default. This file has a `.dsn` extension.
4. Click *Run* to translate the layout editor data.
Informational messages about the status of the translation appear at the bottom of the dialog box.
5. Click *Close* to dismiss the dialog box.

Related Topics

- [specctra_out](#)

specctra checks

The `specctra checks` command lets you run router and alignment checks on the current design to identify routing problems prior to running Allegro PCB Router XL. A window displays the `specctra.log` with items that may not have a corresponding constraint in Allegro PCB Router XL or that may otherwise cause the router to fail.

Access Using

- Menu Path: *Route – Router – Router Checks (for Allegro X PCB Editor)*
- *Menu Path for APD: Route – Router – Router Checks*

Related Topics

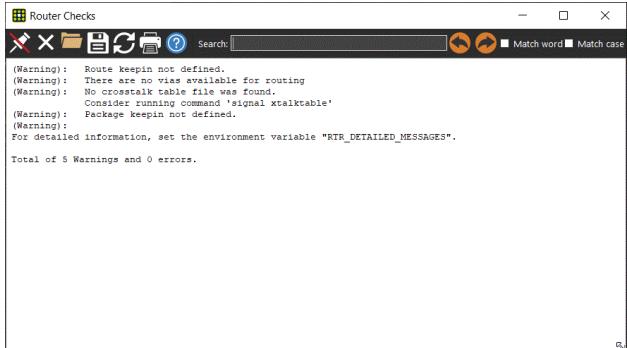
- [spif](#)
- [spif_batch](#)

Running Router and Alignment Checks on a Design

To run router and alignment checks in your design:

1. Type `specctra checks` in the Command window.

The Router Checks window appears.



spif

The `spif` batch command that does one of the following, depending on the arguments you use:

- Launches the Automatic Router dialog box. Does a pre-routing check in addition to running the Allegro PCB Router XL automatic router interactively.
- Launches the Allegro Router Interface dialog box. Transfers data between the layout editor and Allegro PCB Router XL.

The interactive command for pre-routing checks is [specctra checks](#), the interactive command for running Allegro PCB Router XL is [auto_route](#), and the interactive commands for transferring data between the layout editor and Allegro PCB Router XL are [specctra_in](#) and [specctra_out](#). In addition, the [route_by_pick](#) command routes chosen nets and components rather than the whole design.

The batch command [spif_batch](#) also does a pre-routing check.

Syntax

```
spif [-io <design_file> <design_name>.dsn [<design_name>.ses <design_file>] [<design_name>.ses <design_file> <new_design_file>]]
```

spif	Without any arguments, displays the Automatic Router dialog box from which you can check a design and then route it automatically with Allegro PCB Router XL.
-io	Displays the Allegro Router Interface dialog box, where you enter data you are transferring between the layout editor and Allegro PCB Router XL. You can also enter file names with this argument.

Related Topics

- [Running the Pre-Route Checker](#)

SPIF Dialog Boxes

Automatic Router Dialog Box

This dialog box is the same as the one that appears for the [auto_route](#) command, but it includes two additional buttons:

<i>Run Checks</i>	Runs a pre-route check of the current design to identify conditions that could result in routing failure.
<i>Script</i>	Allows you to record a script. See the script command for details.

Allegro Router Interface Dialog Box

Use this dialog box to enter the data you want to transfer from the layout editor to Allegro PCB Router XL and Allegro PCB Router XL to the layout editor.

Write do/dsn Files Tab

<i>Import From Design</i>	Specify the Allegro PCB Design. brd file you want translated into the Allegro PCB Router XL format.
<i>Export To SPECCTRA Design</i>	Specify the Allegro PCB Router XL. dsn file that is created from the Allegro PCB Design. brd file data. The default matches the board design name specified in the <i>Import From Design</i> field.

Update Design Tab

<i>Import From Router</i>	Specify the Allegro PCB Design. brd file that the Allegro PCB Router data is to be translated into. This is the same. brd file you specified in the <i>Write do/dsn</i> tab.
<i>SPECCTRA Session</i>	Specify the Allegro PCB Router. ses file you want to translate into the Allegro. brd file. The default matches the Allegro X PCB Editor design name specified in the <i>Import From Design</i> field.
<i>Export To Router</i>	Specify a name for the finished routed.brd . The default matches the Allegro X PCB Editor design name specified above. ⚠ If the Import and Export. brd file names are the same, the Import file is overwritten and you lose your starting board.

Buttons

<i>Run</i>	Runs the Allegro Router Interface program.
<i>Close</i>	Ignores your input and closes the dialog box. Dismisses the dialog box after execution of the spif program.

Running the Pre-Route Checker

To run the pre-route checker, follow these steps:

1. Run [spif](#) from your operating system prompt or from the spif icon.
 2. If you do not specify a file in the SPECCTRA Automatic Router Open dialog box, choose a design.
 3. In the Automatic Router dialog box, click *Run Checks*.
- A pre-route check is run on the entire design, and the Router Checks window displays any warnings or errors in the [specctra.log](#) file.

Running the Translator

To export a design to a Allegro PCB Router XL design file, use this syntax:

```
spif [-io [<design_file> <design_name>.dsn]
```

To import a Allegro PCB Router XL session file to design file and overwrite the design:

```
spif -io [<design_name>.ses <design_file>]
```

To import a Allegro PCB Router XL session file to a design file, update the design, and write the update to a new design file without overwriting the existing design:

```
spif -io [<design_name>.ses <design_file> <new_design_file>]
```

Related Topics

- [spif](#)

spif_batch

The `spif_batch` batch command that performs a pre-routing check on a chosen design.

Other commands that also perform pre-routing checks are the interactive command Automatic Router Parameters dialog box and the batch command `spif`, which also runs the Allegro PCB Router XL automatic router.

Syntax

```
spif [<filename>.brd] [-version]
```

Related Topics

- [Automatic Router Parameters Dialog Box](#)
- [spif](#)

Performing a Pre-Routing Check on a Design

To perform a pre-route check on your design, follow these steps:

1. Run `spif_batch` from your operating system prompt.
2. If you do not specify a file in the SPECCTRA Automatic Router Open dialog box, choose a design.
A pre-route check is run on the entire design.
3. To view any warnings or errors, open the `specctra.log` file in a text editor.

spin

The `spin` command rotates a graphic element around a point you define.

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 are:

- Symbols
- Pins
- Vias
- Clines
- Lines
- Shapes
- Figures
- Text

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

- *Options* to display all parameters relevant to the command when you need to quickly change one parameter. Changing a parameter here automatically updates its value on the *Options* panel as well.

Allegro Package Use of the Command

When you select a die or BGA symbol, your Allegro Package product spins all members of the die, including attached tiles and via structure elements.

Related Topics

- [Rotating a Graphic Element](#)
- [Rotating Multiple Graphic Elements](#)
- [Rotating an RF Clearance Assembly Group](#)

Spin Command: Options Panel

Access Using

- Menu Path: *Edit – Spin*

<i>Ripup Etch/Conductor</i>	Rips up any connection elements back to the closest pin, T connection, or via, and sets <i>Stretch etch</i> to <i>Off</i> . When this button is off, the layout editor leaves any connections associated with the element on the board/substrate as dangling lines, even though they are no longer connected.
<i>Stretch Symbol/via</i>	Rips up the first line segment of any connection attached to the element and adds an odd angle segment between the rotated element and the rest of the connection.
<i>Type</i>	Specifies the mode of rotation. Choose from: <ul style="list-style-type: none"> • Absolute to read the number entered in the <i>Angle</i> field as the angle at which to rotate the element. When you choose <i>Done</i> in the pop-up menu it automatically turns the element once to match that angle. • Incremental for dynamic control over turning the element. Use the number entered in the <i>Angle</i> field as the amount by which to increment each turn.
<i>Angle</i>	Specifies the angle of rotation, but has a slightly different meaning with each mode. In <i>Incremental</i> mode, this field specifies the number of degrees comprising each increment as you dynamically rotate the element. In <i>Absolute</i> mode, this field specifies the angle of rotation from the 0,0 orientation when the command is executed and the layout editor rotates the element immediately to that angle. Type a number between 0 and 360 or choose an option from the pop-up menu. Choose from 0, 45, 90, 135, 180, 215, 270, and 315. the layout editor provides accuracy to three decimal places.
<i>Point</i>	Indicates the position around which the element turns. Choose from: <ul style="list-style-type: none"> • Symbol Origin: The origin point of each selected item. Each individual item in the selection set will rotate around its origin. • Body Center: The point that is at the center of an invisible boundary it draws around the very edge of each selected item. Each individual item in the selection set will rotate around its center. • User Pick: Your mouse click. • Symbol Pin #: The pin number you specify. This field causes another field to be added to the <i>Options</i> panel. <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> ▲ For <i>Symbol Origin</i> and <i>Body Center</i>, the point of rotation is different for each item in the selection set. For <i>User Pick</i> and <i>Symbol Pin</i>, the point is same for all the selected items. </div>
<i>Symbol Pin #</i>	Appears when you choose SYMBOL PIN# as the rotation point. Type in the pin number.

Related Topics

- [Rotating Multiple Graphic Elements](#)
- [Rotating an RF Clearance Assembly Group](#)

Rotating a Graphic Element

To rotate a graphic element, perform the following steps:

1. Hover your cursor over an element. The tool highlights it and a datatip identifies its name.
2. Right-click and choose *Spin* from the pop-up menu.
3. Enter the desired rotation parameters in the *Options* panel or right-click and choose *Options* from the pop-up menu.
4. Rotate the element to the appropriate angle and click to position the element.
5. Click the next element to rotate.

Related Topics

- [spin](#)
- [Rotating an RF Clearance Assembly Group](#)

Rotating Multiple Graphic Elements

You can rotate multiple graphic elements in the design by following these steps:

1. Window select a group of elements. The tool highlights them.
2. Right-click and choose *Spin* from the pop-up menu.

The following prompt appears:

Pick reference point for controlling spin angle.

3. Enter the desired rotation parameters in the *Options* panel or right-click and choose *Options* from the pop-up menu.
4. Click to identify a location as the origin of the entire group.
5. Rotate the elements to the appropriate angle and click to position them.

Related Topics

- [spin](#)
- [Spin Command: Options Panel](#)

Rotating an RF Clearance Assembly Group

Follow these steps to rotate an RF clearance assembly group:

1. In the *Find* from enable *Groups*.
2. Hover your cursor over an RF clearance assembly group or window select a group of RF elements.
3. Right-click and choose *Spin* from the pop-up menu.
4. Enter the desired rotation parameters in the *Options* panel or right-click and choose *Options* from the pop-up menu.
5. Rotate the group to the appropriate angle and click to position the group.

The RF clearance assembly group appears at its new location.

Related Topics

- [spin](#)
- [Spin Command: Options Panel](#)
- [Rotating a Graphic Element](#)

Entering Simulation Details, Choosing Report Formats, and Simulating And Generating Reports

1. In the Signal Analysis dialog box, click *Reports*.
The Report Generator (case x) dialog box appears. The current case is named in the title bar.
2. Click to choose the Standard Reports tab.
3. To change to a different case, in the Current Case: field, click to display a list of available cases. Click to choose one.
The title bars and *Current Case*: fields in both the Report Generator (case x) and Signal Analysis [case x] dialog boxes change to reflect the new case.
4. In the Report Types area, click to choose one or more standard reports to generate.
5. In the Fast/Typical/Slow Mode area, click to choose one or more simulation modes.
6. In the Victim area, choose nets and drivers for simulation.
7. In the Aggressors area, choose nets and drivers for simulation and the switching mode.
8. In the Reflection Data Simulation area, choose a simulation Type and a stimulus Measurement point.
9. Click to choose whether or not to *Use Timing Windows* for Crosstalk simulation.
10. Click to choose whether or not to *Save Circuit Files* generated during the simulations.
11. Click to choose whether or not to *Save Waveforms* generated during the simulations.
12. If necessary, use *Preferences* to display the Analysis Preferences dialog box where you can modify simulation parameters.
13. Use *Create Report* to perform the simulations and generate the reports.

The report is created and shown in a text viewer.

Related Topics

- [signal probe](#)
- [Signal Probe Dialog Boxes](#)
- [Signal Analysis Tasks](#)

split plane create

The `split plane create` command lets you split planes on an ETCH/CONDUCTOR subclass that you specify. The added split-planes derive from the shape defined on the ROUTE KEEPIN/ALL layer and lines defined on the ANTI-ETCH corresponding to the ETCH subclass that you choose. The command is used in conjunction with `add line` and `split plane param`, as described in the procedure section, below.

Run this command during the manufacturing preparation stage of the design cycle. This command does not delete any existing shapes in symbols it encounters on the ETCH layer.

Related Topics

- [Creating a Split Plane](#)

Create Split Plane Dialog Box

Access Using

- Menu Path: *Edit – Split Plane – Create*

<i>Select layer for split plane creation</i>	Choose TOP, GND, POWER, or BOTTOM or enter any etch layer as the layer upon which to create the split plane.
<i>Shape type desired</i>	<i>Dynamic:</i> Choose to create a positive shape whose copper area and voids automatically fill and update whenever you edit the shape's elements or its boundary. You can only add a dynamic shape to the etch/conductor class. <i>Static:</i> Choose to create a static solid filled shape whose copper area and voids are not dynamically filled or updated after you edit its elements or boundary.
<i>Create</i>	Click to create the specified split plane.
<i>Cancel</i>	Click to exit command without changes.

Creating a Split Plane

To create a split plane:

1. Run `add line`.
2. In the *Options* panel, choose ANTI-ETCH class and the subclass where you want the split plane.
3. In the *Options* panel, use the line width setting to control the clearance between the split planes.
4. Add the line to indicate where the split is to occur. We suggest that the line endpoints extend beyond the ROUTE KEEPIN/ALL shape that is used as the basis for the split plane.
5. Continue adding lines for the number of required split planes desired.
6. Optional: run the split plane parameter command to indicate the fill style of the shapes on the split plane using `split plane param`.

7. Run `split plane create`.

The Create Split Plane dialog box appears.

8. Enter the layer on which to create a split plane (should correspond to the layer chosen in step 2).
9. Choose a *Shape Type* of dynamic or static.
10. Click *Create*.
The display centers on and highlights a shape and a net data browser appears requesting a net be assigned to the shape.
11. Enter a net name you want associated with this shape.
12. Continue net assignment for each shape that was created.

The split plane fill parameters are read in to determine the shape fill if the plane is positive. If the plane is negative, solid fill is used.

⚠ If the ANTI ETCH/ANTI CONDUCTOR lines used for creating the split plane are added at a width that is less than the minimum shape - to - shape clearance stored in the constraint set, a window appears indicating possible DRC violations detected and asks if you want to continue.

⚠ If dynamic shapes are chosen, then split planes are automatically voided. If static shapes are used, then you must manually void the resulting shapes using *Shape – Manual Void – Element* ([Shape void element](#) command).

Related Topics

- [split plane create](#)

split plane param

Run `split plane param` to set parameters for split planes. A split plane is an imbedded plane with two or more copper areas associated to different nets. For details on creating split planes, see [split plane create](#).

Related Topics

- [Setting Parameters for Split Planes](#)

Split Plane Params Dialog Box

Access Using

- Menu Path: *Edit – Split Plane – Parameters*

Use this dialog box to specify the fill style of the shapes on a split plane.

<i>Style</i>	Indicates the fill style of the shapes on the split plane. The choices are Solid, Vertical, Horizontal, Diag_Pos, Diag_Neg, Diag_Both, or Hori_Vert.
<i>Hatch Set</i>	If the fill style is any other than Solid, you are prompted for this additional input:
<i>Line Width</i>	Indicates the width of the line.
<i>Spacing</i>	Indicates the spacing between lines.
<i>Angle</i>	Indicates the angle of the line.

Setting Parameters for Split Planes

You can set parameters for split planes by performing these steps:

1. Run `split plane param`.

A confirmation box that indicates split plane params are applicable only to positive planes and asks if you want to continue.

2. Do one of the following:

Click *No* if the shapes are on a negative plane. (For negative planes, only solid shapes are created.) The command is aborted.

Click *Yes* if the shapes are on a positive plane. The Split Plane Params dialog box appears.

3. Choose the fill style from the Style list box.

Depending on the style you choose, you are prompted for additional input such as Line Width, Spacing, and Angle on styles other than Solid.

4. Click *OK*.

A window appears asking if you want to override earlier parameters with the new ones you just changed.

5. Click *Yes* to save your changes and close the window. –or– Click *No* to ignore your changes and close the window.

Related Topics

- [split plane create](#)

spread between voids

The `spread between voids` command spreads the clines in a routing channel you specify. Use this command to correct return path issues that occur when clines overlap pad voids on adjacent layers. Typically, you apply the spreading function at the end of the design process after you complete routing, meet all other design constraints, and execute the `highlight sov` command to highlight any problems.

Access Using

- Menu Path: *Route – Resize/Respace – Spread Between Voids*

Spreading Between Voids

To spread the clines in a routing channel, follow this procedure:

1. From the *Route* menu, choose *Resize/Respace – Spread Between Voids* or type `spread between voids` at the command window prompt.
The *Options* panel displays the *Spread Between Voids* parameters.
2. Set the *Active Subclass* to the etch/conductor layer on which you want to edit.
Only segments on the active layer will spread.
3. Optionally, specify the void clearance you want to apply to the spreading function.
4. In the design window, choose two elements (pins or vias) to define a channel.
When you choose the second element, the layout editor automatically finds the segments that run between the two elements and verifies that the segments can spread within the channel while meeting the specified spacing requirements and void clearance parameters.
5. To perform another spreading, choose two new elements to define another channel.

spread clines

The `spread clines` command spreads out the clines in a routing channel you specify. By spreading out clines evenly, you can increase manufacturing yields.

Access Using

- Menu Path: *Route — Router — Spread Clines (APD only)*

Related Topics

- [Spreading Clines](#)

Spread Clines Command: Options Panel

Access Using

- Menu Path: *Route — Router — Spread Clines*

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

When you run the *spread clines* command, the *Options* panel changes to display the command parameters.

This option...	Does this...
Stretch mode	Enables <i>Stretch</i> mode.
Add vertex mode	Enables <i>Add vertex</i> mode.
<i>Single layer only</i>	If enabled, limits spreading to just the active layer. When a layer is selected, it becomes visible. If disabled, allows spreading on all visible layers.  In <i>Add vertex</i> mode, <i>Single layer only</i> is enabled by default and cannot be disabled. In <i>Stretch</i> mode, you can enable or disable <i>Single layer only</i> .
Hug	If enabled, allows the Spread Clines feature to avoid DRC violations by applying the <i>Hug</i> function to the segments that are modified.
Consider adjacent plane voids	If enabled, allows you to enter a value for the spacing to adjacent plain voids. (A negative value indicates an overhang amount.) If disabled, voids are ignored.
Allow spread with DRCs	If enabled, clines are spread even if DRC violations result. If hug is enabled, reasonable effort is made to avoid DRC violations. Where they cannot be avoided, an explanatory message appears. If disabled, cline spreading will abort if the spreading cannot be accomplished without creating DRC violations.

Spreading Clines

Follow this procedure to spread out the clines in a routing channel:

1. From the *Route* menu, choose *Router*, then choose *Spread Clines* or type `spread clines` at the command prompt.
The *Options* panel displays the spread clines parameters.
 2. Specify the options you want to apply to the spreading function.
 3. In the design window, select two elements (pins or vias) to define a channel.
When you select the second element, APD automatically finds the segments that run between the two elements. It then defines the channel and verifies that the segments to be operated on can be spread within the channel while meeting the specified spacing requirements.
 4. If you are in *Add vertex* mode, a box defining the channel is drawn on the screen. You can then lengthen the newly created segments by dragging either end of the box. You can also right-click and choose *Rotate* from the pop-up menu to rotate the segments. You can work with any combination of lengthening and rotating the segments to achieve the best results.
-  You can stretch the segments in either direction by moving the cursor in either direction. For instance, if you select two pins that are one above the other, moving the cursor to the left or right stretches the segments in that direction. Click to turn off stretching for whichever side your cursor is on when you click. To continue to the next set of lines, double-click or right-click and choose *Next* from the pop-up menu.

 If the spreading process generates any DRC errors, the effected clines are highlighted. The highlighting disappears when you select the next channel to spread.
5. To perform another spreading, select two new elements to define another channel.

Related Topics

- [spread clines](#)

spreadsheet to symbol

The `spreadsheet to symbol` command lets you import information from a standard spreadsheet tool such as Microsoft Excel to update a placed component. You can use this command to exchange information with your system architect, front-end tools, or as part of your manufacturing documentation set when signing off a design.

Related Topics

- [Importing Information from a Spreadsheet](#)

Symbol Update from Spreadsheet Dialog Box

Access Using

- Menu Path: *File – Import – Symbol Spreadsheet*

<i>File Name</i>	Specifies the name of the file to be exported.
<i>File type</i>	Lets you select any one of the file types as input: <i>TXT</i> (text), <i>CSV</i> (comma separated value), or <i>XML</i> (open spreadsheet XML). The XML format retains the highlight color for pins or nets. The cell delimiter in <i>TXT</i> files is tab and in <i>CSV</i> files is comma. By default <i>XML</i> is selected.
<i>Worksheet</i>	Select the worksheet of the spreadsheet from which you want to import data. By default, the first worksheet is selected. This field is available only for <i>XML</i> file types.
<i>Grid</i>	Specifies what the lines of data in a cell represent. The available entries are <i>Pin Number</i> , <i>Pin Name</i> , <i>Port Name</i> , <i>Net Name</i> , <i>Pin Use</i> , <i>Swap Code</i> , <i>Padstack</i> , <i>Rotation</i> , and <i>Net Groups</i> .
<i>Delimiter</i>	To input secondary text from the file, use this field to indicate the delimiting string that the tool uses between the Primary and Secondary text. The default setting is "", but you can enter any character or string, for example, "...". This field is enabled only when you enable the <i>Secondary text</i> field. Available only for <i>TXT</i> and <i>CSV</i> files.
<i>Spreadsheet cells have data labels</i>	Select to indicate that each line of the cells in the imported file contain keyword identifiers that should be removed to obtain the actual data. Not selected by default. This field is available only if the cells in the selected spreadsheet have data labels. Following are the keywords: <ul style="list-style-type: none"> ◦ PINNUMBER ◦ PINNAME ◦ PORTNAME ◦ NET ◦ PINUSE ◦ SWAPCODE ◦ PADSTACK ◦ ROTATION ◦ NETGROUPS
<i>Spreadsheet has row and column headers</i>	Select to specify the first row and first column in the worksheet as headers.
<i>Add/Delete pins based on cell contents</i>	Select to indicate that pins should be added or deleted based on the imported spreadsheet. Pins will be added if a cell that should be empty is not, using default values based on other pins of the symbol, with overrides applied to specific values as indicated in the cell contents. Pins will be deleted if a cell is empty but there is a pin in that position in the symbol's pin matrix.
<i>Create new nets defined in spreadsheet</i>	Select to create any new nets defined in the spreadsheet.
<i>Allow deassignment of pins</i>	Select to allow deassignment of pins.
<i>Assign cell colors to nets</i>	Select to highlight the nets with the color of the corresponding cells in the spreadsheet.

S Commands
S Commands--spreadsheet to symbol

<i>Rows and columns defined by</i>	Select how you want to define rows and columns in the imported symbol. They can be defined by <i>Component pin pitch</i> , by <i>Unique rows/columns</i> , or you can create a <i>Custom</i> definition by providing the required <i>X/Height</i> and <i>Y/Length</i> values for the rows and columns.
<i>Update</i>	When you click this button, the tool updates the component, closes the dialog box, and waits for you to select the next component.
<i>Cancel</i>	When you click this button, the tool closes the dialog box without updating the component, and waits for you to select a new component.
<i>Help</i>	When you click this button, the tool displays context-sensitive help for this command.

Importing Information from a Spreadsheet

To import information from a spreadsheet, follow these steps:

1. Run the `spreadsheet to symbol` command.
2. Select the component to be updated.
The Symbol Update from Spreadsheet dialog box appears.
3. Specify the file name and directory where the file is located.
4. Choose the fields to be read from the file in the grid lists.
5. Click *Update* to update the component.
6. Select another component and follow Steps 3 to 5
-or-
Right-click and choose *Done* to exit the command.

Related Topics

- [spreadsheet to symbol](#)

stab

An internal Cadence engineering command.

stacked via report

The `stacked via report` command displays the Stacked Via Report window that lists vias, stacked vias, the number of instances and, in case of misalignment—the number, location, and version.

status

In the layout mode, you can use the Status tab to verify the current state of shapes and DRCs and update them if they are out of date. An out of date dynamic shape is one for which the *Dynamic Fill* mode has been set to *Fast* or *Disabled* on the *Global Dynamic Shape Parameters* dialog box (non-*Smooth Dynamic Fill* mode). You can also assess the number of unplaced symbols or unrouted nets. In the symbol mode, you can view the number of connect and mechanical pins in the design.

When dynamic shapes are out of date, changing the dynamic fill mode on the *Status* tab produces the following behaviors:

Changing fill mode from	and using this button	produces this result
<i>Disabled</i> to <i>Fast</i>	OK	no update of dynamic shapes changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Fast</i>
<i>Disabled</i> to <i>Smooth</i>	OK	no update of dynamic shapes changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Smooth</i>
<i>Fast</i> to <i>Smooth</i>	OK	no update of dynamic shapes changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Smooth</i>
any selection/no selection	<i>Update to Smooth</i>	updates dynamic shapes to <i>Smooth</i> changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Smooth</i>

Related Topics

- [Displaying Status for RF Components](#)

Status Dialog Box

Access Using

- Menu Path: *Display – Status*

Status Tab

<i>Connect pins</i>	Displays the number of connect pins in the design. (symbol mode only).
<i>Mechanical pins</i>	Displays the number of mechanical pins in the design. (symbol mode only).
Symbols and Nets	
<i>Unplaced symbols</i>	Displays the number and percentage of <i><unplaced symbols>/<total symbols></i> in the design. A green color box means all symbols are placed; yellow, some placed; and red, none placed (layout mode only). Clicking the color box produces the Unplaced Symbol Availability Check report, which lists the availability of unplaced symbols and their location on disk.
<i>Unrouted nets</i>	Displays the number and percentage of <i><unrouted or partially nets>/<total nets></i> in the design. A green color box means all nets are routed; yellow, some routed; and red, none routed. (layout mode only).
<i>Unrouted connections</i>	Displays the number and percentage of <i><unrouted connections>/<total pin-to-pin connections></i> in the design, including nets with the NO_RAT property. A green color box means all connections are routed; yellow, some routed; and red, none routed (layout mode only). The value derives from the netlist's From-To connections and is based on placed components, as is the percentage. Clicking the color box produces the Unconnected Pins report, which lists all unconnected pins in the design with hyperlinks to X/Y coordinates, net names, and total unconnected pins.
<i>Shapes</i>	
<i>Isolated shapes</i>	Displays the number of shapes on nets without connections, known as isolated shapes. Isolated shapes may occur during voiding, or when you add shapes to nets without pins or vias to which to connect. A green color box means no shapes are isolated; yellow, some shapes remain isolated. Clicking the color box produces a report summarizing the data.
<i>Unassigned shapes</i>	Displays the number of copper shapes unassigned to a net. A green color box means no shapes are unassigned; yellow, some shapes remain unassigned. Clicking the color box produces a report summarizing the data. Clicking on the hyperlinked x/y coordinates in the report brings you to that shape location in the design.
<i>Out of date shapes</i>	Displays the number of <i><non-Smooth dynamic shapes>/<total dynamic shapes></i> in layout mode only. A red color box indicates the <i>Dynamic Fill</i> mode for all dynamic shapes has been set to <i>Fast</i> or <i>Disabled</i> on the Global Dynamic Shape Parameters dialog box, making all dynamic shapes out of date (non-Smooth Dynamic Copper Fill mode) as a result. Out of date dynamic shapes prevent artwork output when you run <i>film param</i> , <i>odb_out</i> , and <i>stream out</i> . A yellow color box indicates a portion of all dynamic shapes are out of date in the design. A green color box indicates the <i>Dynamic Fill</i> for all dynamic shapes has been set to <i>Smooth</i> , making all dynamic shapes up-to-date (<i>Dynamic Fill</i> set to <i>Smooth</i>).
	Clicking the color box produces a report, sorted by layer, showing the status of each dynamic shape on the board as follows: <i>Smooth</i> : ready for artwork <i>Out of date</i> : update required <i>No Etch</i> : shape has no etch, possibly due to a route keepout. Delete the dynamic shape or add etch to produce artwork. <i>Update to Smooth</i> : Click to automatically void and run DRC on all dynamically filled shapes, making all dynamic shapes up-to-date (Dynamic Copper Fill mode set to Smooth) and produce artwork quality output (regardless of whether you chose <i>Fast</i> or <i>Disabled</i> in the <i>Fill Mode</i> field above). Changes the current <i>Dynamic Copper Fill</i> mode on the <i>Global Dynamic Shape Parameters</i> dialog box to .

	To cancel dynamic filling of complex shapes for a large design, you can use the <code>Esc</code> key to stop the process, which leaves the shapes out of date. If several shapes are in the midst of dynamically filling when you invoke the <code>Esc</code> key: Shapes already dynamically filled remain completed. Shapes in the process of dynamically filling remain unfilled and marked out of date. Shapes whose dynamic fill is yet to be updated remain filled but marked out of date. <i>Dynamic Fill</i> : Controls automatic voiding and edge smoothing for all dynamically filled shapes. Use this field to change the dynamic copper fill mode while you are evaluating the status of dynamic shapes without opening the <i>Global Dynamic Shape Parameters</i> dialog box. The setting you choose here then defaults to the Global Dynamic Shape Parameters dialog box. <i>Smooth</i> : Choose to automatically void and run DRC on all dynamically filled shapes and produce artwork quality output. <i>Fast</i> : Select to see connectivity without full edge smoothing and thermal hookups in a fast fill mode to obtain true clearances around elements and resolve intersections with other voids. Artwork quality results and artwork are not created. <i>Disabled</i> : Select to globally defer dynamically filling all dynamic shapes you subsequently create or modify to speed performance. Use this option to edit etch for medium to large ECOs, manual ECOs or to run batch programs such as netin, glossing, testprep add/replace vias, for example. Shapes created under this global setting are not voided, and DRC does not run. They are marked out of date to be filled later. Artwork cannot be produced.
--	---

DRCs and Backdrills

<i>DRC errors</i>	Indicates whether DRC markers are up-to-date. The status can be Out Of Date or Up to Date. A red color box indicates DRC is out of date or Batch DRC is required. A yellow color box indicates DRC is up to date, but DRC errors exist. A green color box indicates DRC is up to date and no DRC errors exist.
<i>Shorting errors</i>	Click to display the total number of net short errors. It is only enabled when online DRC is enabled.
<i>Update DRC</i>	Click to display the total number of errors. It is only enabled when online DRC is enabled.
<i>Waived DRC errors</i>	Displays the count of waived DRC errors that exist in the design. Waived DRC errors are never considered out-of-date. A green color box indicates there are no waived DRC errors present in the design. A yellow color box indicates there are waived DRC errors.
<i>Waived shorting errors</i>	Click to display the total number of waived net short errors. It is only enabled when online DRC is enabled.
<i>On-Line DRC</i>	Specifies whether you run DRC online (<i>On</i>) or in batch mode (<i>Off</i>). Default is <i>On</i> . You should leave DRC mode on so that as you change the design, you get immediate feedback about design rule violations. For better performance, turn it off, but you should run a batch DRC update before manufacturing the board.
<i>Out of date backdrills</i>	Indicates whether DRC markers are up-to-date. The status can be Out Of Date or Up to Date. A red color box indicates backdrill data is out of date. Click <i>Update Backdrill</i> to update the status. A grey color box indicates no backdrill data (side, start layer, must-cut-layer) saved on pins/vias yet. A green color box indicates backdrill data on pins/vias are synced.
<i>Update Backdrill</i>	Opens the Backdrill Setup And Analysis dialog box to perform backdrilling again and update the saved data (side, start layer, must-cut-layer) on pins/vias.
<i>OK</i>	Closes the dialog box.
<i>Refresh</i>	Click to display the most recent status for symbols, nets, and shapes.

RF Status Tab

<i>Import Logic from 'packaged' folder</i>	Displays an Open browser window for indicating the path of file packaged folder.
<i>Load</i>	Loads packaged (<code>pstxnet.dat</code> , <code>pstxprt.dat</code>) files to compare schematic design data with layout design data.
<i>Changed Status Count</i>	
<i>Unplaced Components</i>	Displays the number and percentage of <code><unplaced components>/<total components></code> in the design. A green color box means there is no change between schematic and physical design. A yellow color box shows that there are differences between schematic and physical design data. Clicking the color box produces the <code>details_about_unplaced_RF_component(s)</code> report, which lists the availability of unplaced components.
<i>Changed logic nets</i>	Displays the number and percentage of <code><changed logic nets>/<total nets></code> in the design. A green color box means connectivity of none of the nets are changed between schematic and physical design. A yellow color box shows that connectivity of some of the nets are changed between schematic and physical design data. Clicking the color box produces the <code>details_about_logic_changed_net(s)</code> report, which lists connectivity details for all the nets with changed logic. The report displays pin numbers and their locations for each net when they are added in the design.
<i>Components with changed parameters</i>	Displays the number and percentage of <code><components with changed physical parameters>/<total components></code> in the design. A green color box means of none of the components parameters are changed. A yellow box shows that some of the components parameters are changed between schematic and physical design. Clicking the color box produces the <code>details_about_RF_parameter_changed_component(s)</code> report, which lists details of all the components with changed parameters and their locations.
<i>Components with changed RF type</i>	Displays the number and percentage of <code><components with changed RF type>/<total components></code> in the design. A green color box means of none of the components type is changed. A yellow color box shows that some of the components type is changed. Clicking the color box produces the <code>details_about_RF_type_changed_component(s)</code> report, which lists all the details of components with changed RF type and their locations.
<i>Components added</i>	Displays the number and percentage of <code><components added>/<total components></code> in the design. Clicking the color box produces the <code>details_about_newly_added_RF_component(s)</code> report, which lists all the newly added components and their locations.
<i>Components deleted</i>	Displays the number and percentage of <code><components deleted>/<total components></code> in the design. Clicking the color box produces the <code>details_about_deleted_RF_component(s)</code> report, which lists deleted components.
<i>OK</i>	Closes the dialog box.
<i>Refresh</i>	<i>Click to display the most recent status for symbols, nets, and shapes.</i>

Displaying Status for RF Components

To display the status for RF components:

1. Run the `status` command or Choose *Display – Status*.
The *Status* dialog box is displayed.
2. Open *RF Status* tab.
3. Browse path of schematic packaged directory.
4. Click *Load* to load the packaged files.
5. Click *Refresh* to display the RF Status.
6. Click *OK* to close the dialog box.

Related Topics

- [status](#)

step_out Batch Command

The `step_out` batch command lets you export an Allegro layout as a STEP model for use in a mechanical design environment.

 This command is available only on Windows and Linux.

For more information, see the [Allegro User Guide:Defining and Developing Libraries](#) in your documentation set.

Syntax

```
step_out [-uplsmnadcbz] [-o <output_file>] <brd>
```

-u	Defines output units in the layout editor unit of measure. The default is database units. (INCH, MILLIMETER, and MICRON).
-o	Defines the name of the output file. The default name is <code><design_name>.stp</code> .
-p	Defines STEP protocol. The default is AP214. Available protocols are: AP203, AP214, AP242.
-l	Includes secondary STEP model, if exists. The default is Off.
-f	Ignores STEP model definitions for mapped symbols. The default is Off.
-s	Defines source tool. The default is <i>CDNS_Allegro</i> .
-m	Specifies package with mapped STEP model. The default is Off.
-n	Specifies package without mapped STEP model. The default is Off.
-a	Specifies mechanical assembly or enclosure if any exists. The default is Off.
-d	Specifies mechanical drill holes. The default is Off.
-i	Specifies electrical through pin holes. The default is Off.
-v	Specifies electrical through via holes. The default is Off.
-c	Specifies external copper (traces/pads/shapes). The default is Off.
-t	Specifies internal copper (traces/pads/shapes). The default is Off.
-b	Specifies bare board. The default is Off.
-z	Specifies .zip file. The default is Off.
<brd>	Required. design file name.

You can access this information by typing `step_out` at your operating system's command prompt.

Examples

- To generate a STEP file test.stp which contains packages with STEP models, packages without STEP models, mechanical drill holes, and generate a .zip file:

```
step_out test.brd -o test -m -n -d -z
```

- To generate a STEP file test.stp with output units MILLIMETER and STEP protocol AP-203:

```
step_out test.brd -o test -u MILLIMETER -p AP203 -mndz
```

- To generate STEP file test.stp which contains bare board, mechanical drills, and external copper:

```
step_out test.brd -o test -u MILLIMETER -p AP203 -bdc
```

step out

The `step out` command lets you export an Allegro layout as a STEP model for use in a mechanical design environment.

Related Topics

- [Exporting an Allegro Layout as a STEP Model](#)
- [Defining and Developing Libraries Overview](#)

STEP Export Dialog Box

Access Using

- Menu Path: *File – Export – STEP*

Output file name	Specifies the name for the output file. The default value is the base name of the active design. To search for existing files, click... to display the file browser.	
Output units	Specifies the unit for the STEP model export. The default value is the unit of the active design. Choose Millimeter, Micron, and Inch.	
STEP Protocol	Specifies output protocol formats. Choose, AP-203, AP-214, and AP-242. The default is AP-214.	
Source identification	Specifies the tool of origin within the STEP model data. The default is the current release of PCB Editor.	
Export Options		
Parts		
Parts with STEP models	Includes the STEP models mapped in the current design.	
	<i>Use secondary STEP models</i>	Includes secondary STEP models.
	<i>Ignore STEP model definitions</i>	Ignores STEP model definitions for mapped symbols.
Parts without STEP models	Includes symbols without STEP models mapping. The unmapped symbols are exported as defined by the PACKAGE GEOMETRY/PLACE_BOUND_TOP/BOTTOM.	
Assemblies and enclosure parts	Includes the STEP3D_MECH models created in the STEP Package Mapping tool.	
Selected and highlighted parts only	Includes selected and highlighted parts.	
Drill Holes		
Mechanical holes	Includes mechanical holes defined in the current board drawing.	
Electrical through holes	Includes electrical through holes defined in the current board drawing.	
	<i>Pins</i>	Includes electrical through pin holes defined in the current board drawing.
	<i>Vias</i>	Includes electrical through via holes defined in the current board drawing.
Copper Layers		
External copper (Traces, Pads, Shapes)	Includes external traces, pads, and shapes on the external (ETCH/TOP and ETCH/BOTTOM) layers.	

S Commands
S Commands--step out

<i>Internal copper (Traces, Pads, Shapes)</i>	Includes internal traces, pads, and shapes on the internal layers. Exports 3D copper that includes internal traces, pads, and shapes. By default, copper is exported as 2D geometries to keep the export data file small in size. Set the user preference environment variable <code>step_3d_copper</code> for 3D copper export with bigger file size.
<i>Bare board</i>	Includes physical board.
<i>Compress output file (.zip)</i>	Compressed the exported STEP file into a <code>.zip</code> file.
<i>Export</i>	Choose to export the current design as a STEP model. Starts the export process.
<i>Close</i>	Closes the STEP Export dialog box without running the STEP export program.
<i>Viewlog</i>	Displays the <code>step_out.log</code> that contains messages generated during STEP export, and is available only after you have exported data from the design.

Exporting an Allegro Layout as a STEP Model

Follow this procedure to export an Allegro layout as a STEP model:

1. Run the `step out` command or Choose *File – Export – STEP*.
The *STEP Export* dialog box is displayed.
2. Specify the file name and directory to save the files.
3. Specify the output unit for STEP export.
4. Specify the *STEP Protocol*.
5. Specify *Export Options*.
6. Optionally, choose to create compressed output file.
7. Click *Export* to start the export process.
8. Choose *Viewlog* to view the logfile.
9. Click *Close* to close the dialog box.

Related Topics

- [step out](#)

step pkg map

The `step pkg map` command lets you map device or package and mechanical symbols to Standard for the Exchange of Product (STEP) models and save the mapping data in STEP mapping file in an ASCII or XML format. The STEP model supports more detail component modeling to ensure proper clearances and positioning and provides precise representation in 3d viewer.

You can use both device or package modes in a design. But the device mapping overrides package mapping in 3d viewer and on exporting STEP models.

The STEP mapping data can be reused when importing logic into another design. During logic import, the mapping data is automatically imported during and attached to the devices and symbols in the design. You can also import mapping data from different source design.

You can map STEP models both in PCB Editor and Symbol Editor.

 This command is available only on Windows and Linux.

Related Topics

- [Mapping STEP Model to Device/Package](#)
- [Mapping STEP Model to Mechanical Symbol](#)
- [Exporting STEP Models](#)
- [Importing STEP Models](#)
- [Defining and Developing Libraries](#)

Device/Package STEP Mapping Dialog Box

Access Using

- Menu Path: *Setup – Step Package Mapping*

Specify STEP models mapping parameters for devices and package through this dialog box. The minimum vertical screen resolution is set to 1050 for this dialog box to display all the fields.

<i>Mode</i>	Choose the mapping mode as follows: <ul style="list-style-type: none">◦ <i>Device</i>: saves the mapping properties on the component definition.◦ <i>Package</i>: saves the mapping properties on the symbol definition
<i>Available Devices</i>	Displays list of the devices present in the current design.
<i>Device filter</i>	Filters the devices by name.
<i>Devices missing STEP model</i>	Filters the devices that are unmapped.
<i>Available Packages</i>	Displays list of the symbols present in the current design.
<i>Name filter</i>	Filters the symbols by name.
<i>Symbols missing STEP model</i>	Filters the symbol that are unmapped.

S Commands
S Commands--step pkg map

Mapping Status	Specifies the mapping status of the selected symbol as <i>Primary</i> or <i>Secondary</i> .
Add Mech...	Creates a board or mechanical symbol that represents the mechanical model (enclosure).
Delete Mech...	Deletes the selected mechanical model from the <i>Available Packages</i> list and removes the mapping.
Available STEP Models	Displays lists of all STEP models available in the library path specified by the <i>steppath</i> environment variable.
Name filter	Filters the STEP models by name.
Path	Provides information of the <i>steppath</i> environment variable value.
Display origin of STEP model	Displays the origin of the STEP models in the graphics pane.
Graphics Pane	Displays the package/device model and STEP model graphics with the board section.
View	Specifies different modes of viewing a STEP model in the graphics pane.
Transparent	Specifies the transparency mode for the STEP model in the graphics pane.
Overlay	Merges the package and STEP model views into a single view.
Hide board	Removes the graphical image of the board in the viewer.
STEP color	Uses the colors defined in the STEP model, if enabled.
Primary/Secondary STEP model	
Delete	Deletes primary STEP model associated with the selected device or package.
Primary model	Choose to specify the primary mapping for the selected device or package.
Delete	Deletes secondary STEP model associated with the selected device or package.
Secondary model	Choose to specify the secondary mapping for the selected device or package.
Map STEP Model	
Rotation X, Y, Z	Specifies the rotation in degrees in X, Y and Z directions to correctly position the STEP model in relation to device/package.
Offset X, Y, Z	Specifies the offset in Millimeters in X, Y and Z directions to correctly position the STEP model in relation to device/package.
Arrow key increment	Specifies the increment value in Millimeters for arrow key movements.
Arrow Buttons	Moves STEP model Up/Down/Left/Right by increment value in the 2D view (Top, Bottom, Front, Back, Left, Right).
Mouse Button	Enables the mouse buttons actions for moving, panning and zooming the STEP models in the graphical pane.
?	Displays information on arrow and mouse button actions.
Report	Creates a report of symbols and devices in the working directory. The <code>HTML</code> file (<code>STEP_Device_Package_Mapping_Report.html</code>) includes mapping status, STEP model names, rotation, and offset values.
Save	Saves the mapping data of the current symbol or device displayed in the graphics pane.
Import	Imports the mapping data from a STEP device/package mapping file (<code>.map</code>) into a design and attach the STEP facet files into the design for 3D display. <div style="border: 1px solid #ccc; padding: 5px; width: fit-content;">⚠ Before importing, extract the STEP facet files into the <code><working_directory>/stepFacetFiles4Map/</code>.</div>

<i>Export</i>	Saves the mapping data into an XML format to a file (STEP device/package mapping file(<code>.map</code>)) for re-use or import at the location specified by the <code>step_mapping_path</code> environment variable. Also saves the facet representation of the STEP geometry and assembly for 3d display into a <code>.zip</code> file (<code>stepFacetFiles4Map.zip</code>) at the location specified by the <code>step_facet_path</code> environment variable.
<i>Purge</i>	Deletes STEP device/package mapping information and the STEP facet data from the database. This option saves the mapping data to a file (<code>.map</code>) and the STEP facet files (<code>stepFacetFiles4Map.zip</code>) into the working directory.
<i>Close</i>	Close the STEP Package Mapping dialog box without saving the mapping.

Related Topics

- [Mapping STEP Model to Mechanical Symbol](#)
- [Exporting STEP Models](#)
- [Importing STEP Models](#)

Mapping STEP Model to Device/Package

Follow this procedure to map STEP models to devices or packages:

1. Run the `step pkg map` command or Choose *Setup – Step Package Mapping*.
The *STEP Package Mapping* dialog box appears.
2. Choose modes as *Device* or *Package*.
3. Choose the package from the *Available Package* lists.
The package symbol is displayed in the graphic pane.
4. Choose the STEP model from the *Available STEP Models* lists.
The STEP model is displayed in the graphic pane.
5. Choose Primary (Secondary) STEP model option to map primary STEP model to the package.
6. Specify the *View* from the pull-down list.
7. Set *Transparent* mode from the pull-down list.
8. Choose *Overlay* to merge the symbol/device and STEP model view.
9. Optionally, choose *Hide board* to hide the board graphics.
10. Specify the rotational and offset values for correct placement of STEP model.
11. Alternatively, use arrow keys for STEP model placement.
12. Click *Save* to save the mapping data for the selected symbol/device.
13. Click *Report* to create the report of symbols/devices, mapped STEP models and other mapping details.
14. Click *Export* to export the mapping data to STEP package/device mapping file (`.map`).
15. Click *Close* to close the dialog box.

Related Topics

- [step pkg map](#)
- [Exporting STEP Models](#)
- [Importing STEP Models](#)

Mapping STEP Model to Mechanical Symbol

Follow this procedure to map STEP models to mechanical symbols:

1. Run the `step pkg map` command or Choose *Setup – Step Package Mapping*.
The *STEP Package Mapping* dialog box is displayed.
2. Choose *Mode as Package*.
3. In the Available Package section, choose *Add Mech* to add mechanical symbol or board.
A prompt is displayed to enter the name of mechanical symbol.
4. Enter the name of mechanical symbol with prefix *STEP3D_MECH_* for enclosure, cage or bracket and click *OK*.
5. Choose the mechanical symbol added from the Available Package lists.
The mechanical symbol is displayed in the graphic pane.
6. Choose the STEP model from the Available STEP Models lists.
The STEP model is displayed in the graphic pane.
7. Choose Primary (Secondary) STEP model to map primary STEP model.
8. Specify the View from the pull-down list.
9. Set *Transparent* mode from the pull-down list.
10. Choose *Overlay* to merge the symbol and STEP model view.
11. Optionally, choose *Hide board* to hide the board graphics.
12. Specify the rotational and offset values for correct placement of STEP model.
13. Alternatively, use arrow keys for STEP model placement.
14. Click *Save* to save the mapping data for the selected mechanical symbol.
15. Click *Report* to create the report of symbols/devices, mapped STEP models and other mapping details.
16. Click *Export* to export the mapping data to STEP Package Mapping file (*.map*).
17. Click *Close* to close the dialog box.

Related Topics

- [step pkg map](#)
- [Device/Package STEP Mapping Dialog Box](#)
- [Importing STEP Models](#)

Exporting STEP Models

To export STEP models:

1. Run `enved` command or Choose *Setup – User Preferences*.
The *User Preferences Editor* appears.
2. Set `step_mapping_path` and `step_facet_path` environment variables. These variables specifies the location for saving mapping data.
3. Click *OK* in the *User Preferences Editor*.
4. Run the `step pkg map` command or Choose *Setup – Step Package Mapping*.
The *STEP Package Mapping* dialog box appears.
5. Map the STEP model to devices and packages present in the design.
6. Click *Export* to start the export process.
The file browser appears and display the directory defined in the `step_mapping_path` variable.
7. Enter the name of mapping file and click *Save* in the file browser.
The progress bar show the progress.
8. Click *OK* in the confirmation dialog that displays the name and pat of mapping files.
9. Click *Close* to close the *STEP Package Mapping* dialog box.

Related Topics

- [step pkg map](#)
- [Device/Package STEP Mapping Dialog Box](#)
- [Mapping STEP Model to Device/Package](#)

Importing STEP Models

To import STEP models:

1. Run `enved` command or Choose *Setup – User Preferences*.
The *User Preferences Editor* appears.
2. Set `step_mapping_path` and `step_facet_path` environment variables. These variables specifies the location of mapping data.
3. Click *OK* in the *User Preferences Editor*.
4. Run the `step pkg map` command or Choose *Setup – Step Package Mapping*.
The *STEP Package Mapping* dialog box appears.
5. Click *Import* to start the import process.
The file browser appears and display the directory defined in the `step_mapping_path` variable.
6. Select the name of mapping file and click *Open* in the file browser.
The progress bar shows the import process.
7. Click *Close* to close the *STEP Package Mapping* dialog box.

Related Topics

- [step pkg map](#)
- [Device/Package STEP Mapping Dialog Box](#)
- [Mapping STEP Model to Device/Package](#)
- [Mapping STEP Model to Mechanical Symbol](#)

stop

The `stop` command typed at the command window prompt stops both script and macro recording processes and closes the `.scr` file into which the script or macro was being recorded. This command performs the same function as the *Stop* button in the Scripting dialog box. For information on other scrip commands, see [script](#).

Access Using

- Menu Path (to scripting dialog box): *File – Script*

stopwatch

The `stopwatch` command offers electronic timing options within the tool. Command line options offer capability similar to that of a handheld stopwatch. Normally, these are embedded within a script for timing purposes. The `stopwatch` command reports elapsed wall time to a tenth of a millisecond in `hh:mm:ss:ff` format. If you execute the `stopwatch` command without specifying one or more options, the layout editor displays the list of options. Reenter the command with the appropriate options.

Syntax

```
stopwatch [<option>]
```

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

Example

```
stopwatch reset  
<Allegro commands>  
stopwatch stop
```

stream_out Batch Command

The `stream_out` batch command extracts the film records to create a class/subclass to stream layer filter table. This batch command also uses the stream full-geometry view to extract all geometric information from the layout editor database and converts only those class/subclasses included in the layer filter table.

⚠ The batch command defaults to board naming conventions. For example, if you have a CONDUCTOR class in the layer conversion file and want to run the command for Allegro X Advanced Package Designer (APD), you should either change the class to ETCH or specify the tool (`setenv APD=1`) from the command prompt.

Arcs and circles are converted to line segments before conversion to stream because stream does not allow arcs and circles.

If you attempt to export a layout editor design to GDSII stream format, and dynamic shapes are out-of-date, `stream_out` fails. Run `status` to use the Status tab to verify the current state of dynamic shapes and DRCs and update them if they are out of date. You cannot export until you update dynamic shapes or DRCs.

⚠ The command can also be run in interactive mode from the command window prompt. See [stream out](#).

For additional information on GDSII stream format, see *GDSII Bi-Directional Manufacturing Interface* in your product documentation. For information on converting geometric data from a GDSII Stream file (.sf) and creating a layout editor design file, see the [load stream](#) command.

⚠ Before running `stream_out` from the operating system command prompt either stand-alone or as part of a script, set `allegro_mcm` or `sip` depending on the files you are using. For example, in Linux, run the following for an MCM file: `export allegro_mcm=1`.

Syntax

```
stream_out [-udon [f|r|s] p2Rct] <-c filename.cnv> <design_name>
```

-u	Optional. Defines output units in the layout editor unit of measure. The default is database units. (Allowable units are: MM, CM, IN, MIL, and MIC).
-d	Optional. Specifies the accuracy in database units per user unit in integer powers of 10 (10, 100, 1000, etc.). The default is the database accuracy.
-o outputfile	Optional. Changes the name of the output stream file from <code><design_name>.sf</code> to <code><outname>.sf</code> (60-character maximum). The default is <code><design_name>.sf</code> .
-n	Optional. Specifies the name of the top GDSII Structure. The name is limited to 32 characters. Default is <code>STR_1</code> .
-f r s	Optional. Specifies the path endpoint style, namely, flush, round, or square. Default is round (-r). The -f option converts all etch/conductor endpoints to end flush digitized points using stream path type 0, which overrides the default path type for etch/conductor clines of 1. Cannot be used in combination with -r or -s. The -r option converts all etch/conductor endpoints to end with a semicircle with its center at digitized end points. The default value. Cannot be used in combination with -f or -s. The -s option converts all etch/conductor endpoints to end with a square path that extends beyond the digitized endpoints by 1/2 the width of the path using stream path type 2, which overrides the default path type for etch/conductor clines of 1. Cannot be used in combination with -f or -r
-p	Optional. Specifies a flat geometry. When present, stream data is written flat, without hierarchy. This is off by default. (Program outputs hierarchical data).
-P	Optional. Specifies merging of overlapping objects. When present in flatten mode, elements on the same layer will be combined to remove overlaps before being written. Options for what to merge follow. Valid arguments are: LAYER, LAYER_AND_DATATYPE, and DATATYPE. The default is off. (Program does no polygon merging).
-2	Optional. Specifies Dracula Format. Output a GDSII file compatible with the Dracula program.
-R	Optional. Specifies rectangle format. When present, all rectangles are exported as GDSII Boundaries. When not present, unfilled rectangles are exported as GDSII Paths; filled rectangles are exported as GDSII Boundaries. Off by default.

S Commands
S Commands--stream_out Batch Command

-C	Optional. Specifies cline as boundary. When present, all clines are exported as GDSII Boundaries. When not present, clines are exported as GDSII Paths. Off by default.
-S	Optional. Specifies cline as segment-based boundary. When present, all clines are exported as GDSII Boundaries with rounded outside corners at vertices, to most closely match the display in the main layout window. To use, the -C option must also be set. Off by default.
-t	Optional. Specifies text height as magnification. When present the text height is written into the magnification field in user units.
-a	Optional. Specifies the number of segments per circle to be used when converting arcs to segments. Valid arguments are any integer from 3 to 360. Default is 32. Note that specifying a high value will increase the size of the output file and may produce very small segments. Only the default value is recommended for manufacturing. Cadence does not support importing files that were produced with a non-standard number of segments per circle.
-ad	Optional. Specifies the maximum tolerance which, if exceeded, causes an arc to use more segments than that of defined in the Minimum segments per complete circle to vectorize an arc. The actual number of segments used may vary based on the number required to get the tolerance down to the specified value. Default value for this field is the half of the value specified in the -a parameter.
-A	Optional. Specifies the angle of rotation to apply to the output geometries. The valid arguments are numbers from 0.000 through 360.000. ⚠️ Rotating the entire design during manufacturing output should be done with caution, just as mirroring geometries should be. The resulting output is no longer a direct match to the substrate as designed in the database.
-e	Optional. Specifies the vectorizing type. The valid arguments are 0 (default), 1 (inside metal), and 2 (outside metal).
-g	Optional. Specifies the mirror geometries during output. By default, there is no mirroring. When set, all geometries will be mirrored through the Y-axis. ⚠️ Mirroring and rotation of geometries should be done with caution, as this means the GDSII output no longer matches the substrate design.
-c <filename>	Required. Includes the name of the layer conversion file that contains the stream layer to class/subclass mapping. <filename> is the specified .cnv file.
<design_name>	Required. Design file name.

You can access this information by typing `stream_out` at your operating system's command prompt.

stream out

The `stream out` command lets you convert a layout editor design to GDSII stream format. It converts only those classes or subclasses that are included in the layer conversion file.

 When dynamic shapes are out-of-date, the layout editor displays a *Dynamic Shapes Need Updating...* button on the Stream Out dialog box.

If you attempt to export a layout editor design to GDSII stream format, an error message appears: "Dynamic Shapes are out of date, please update them." Click *Dynamic Shapes Need Updating...* to open the *Status* tab, which becomes active, blocking any use of the Stream Out dialog box until you update dynamic shapes and/or DRC before proceeding to export.

You can also use [stream_out Batch Command](#) as a batch command from the command prompt.

For additional information on GDSII stream format, see *GDSII Bi-Directional Manufacturing Interface* in your product documentation. For information on converting geometric data from a GDSII Stream file (.sf) and creating a layout editor design file, see the [load stream](#) command.

Related Topics

- [Stream Out Edit Layer Conversion File](#)
- [Converting an Allegro Design to GDSII Stream Format](#)
- [Editing the Stream Out Layer Conversion File](#)

Stream Out Dialog Box

Access Using

- Menu Path: Manufacture – Stream Out

Top structure name	Indicates the name for the top structure (cell). The initial value is the name of the design file. To specify a different name, enter a value. If you do not specify a name, the top structure name will be the same as the design, without the filename extension (.brd, .mcm).
Output file name	Indicates the name for the output file. The initial value is the base name of the active drawing. To search for existing files, click ... to display the file browser. The file extension .sf is appended to the name, unless you supply a file extension.
Layer conversion file	Indicates the name of the conversion file that maps the database to the GDSII stream layers. To search for existing files, click ... to display the file browser.
Edit	Displays Stream Out Edit Layer Conversion File where you can create or edit the layer conversion file.
Units	
Drawing units	Displays the drawing units of the active drawing.
Output units	Initial value matches the active drawing's units. Indicates the unit of measurement for the GDSII output. Choose Mils, Inch, Microns, Millimeter, or Centimeter.
Output precision	The initial value matches the active drawing's accuracy, specified in orders of magnitude. A higher number represents higher precision. For example, if the drawing accuracy is 2 decimal places, the initial output precision is 100. Changing the number to 1000 has the same effect as is if the drawing accuracy was 3 decimal places.
Scale factor	Indicates how the entries are to be scaled vertically and horizontally. Is independent of units value and is applied after any units conversion. For example, a value of 0.5 reduces each entry by 50 percent; a value of 2.0 increases each entry by 100 percent. The default is 1.0.
Geometry	
Line end cap style	Specifies how to convert endpoints from the layout editor design. Choose one of the following: <ul style="list-style-type: none"> • Round: converts all endpoints to ends with a semicircle with its center at digitized endpoints. • Flush: converts all endpoints to end flush digitized points. • Square: converts all endpoints to ends with a square path that extends by half the width of the path beyond the digitized endpoints.
Minimum segments per complete circle	Specifies the number of segments created for a complete circle when an arc object is converted into straight line segments. The recommended value, also the default is 32. You can, however, specify any value from 3 to 360, the recommendation being for multiples of 2 and a minimum of 8. Note that the segments mentioned are for a full circle. As a result, for an arc that is half a circle, the segments will be half that for a circle.
Maximum distance between arc and segment	Specifies the maximum tolerance which, if exceeded, causes an arc to use more segments than that defined in the Minimum segments per complete circle to vectorize an arc. The actual number of segments used may vary based on the number required to get the tolerance down to the specified value. Default value for this field is the half of the value of <i>Minimum aperture for gap width</i> .
Vectorizing type	Specifies the vectorizing type. The valid arguments are: <i>default</i> , <i>inside metal</i> , and <i>outside metal</i> .

Mirror geometry	<p>Specifies the mirror geometries during output. By default, there is no mirroring. When set, all geometries will be mirrored through the Y-axis.</p> <div style="border: 1px solid red; padding: 5px; margin-top: 10px;"> <p>! Mirroring and rotation of geometries should be done with caution, as this means the GDSII output no longer matches the substrate design.</p> </div>
Flatten geometry	Specifies whether stream_out data is flat or without hierarchy. The initial setting is Off.
	Off: GDSII structures are generated for all used padstacks and GDSII structure references are created for each instance of the padstack. Results in smaller output file size and faster processing. On: all pads are processed individually as GDSII boundaries. Choose to export unfilled rectangles as GDSII boundaries. Otherwise, unfilled rectangles are exported as GDSII paths. Disabled by default.
Output to Dracula	Choose to output a GDSII file for use in Dracula and output voids within a shape and pads as shapes as shape entities.
Output all rectangles as boundaries	Specifies that all rectangles should be output as boundaries, creating a bounding polygon for all the segments that fill the rectangle.
Output all clines as boundaries	Specifies that all clines should be output as boundaries, creating a bounding polygon for all the cline's segments.
Round outside corners at vertices	Specifies cline as segment-based boundary. When present, all clines are exported as GDSII Boundaries with rounded outside corners at vertices, to most closely match the display in the main layout window. This option is enabled only if the Output all clines as boundaries check box is selected.
Output cross-hatch shapes as segment boundaries	Specifies that all the conductor bands and the non-conductor bands in the cross-hatch shapes should be output as boundaries, creating polygons for all the bands. By default, only the conductor bands of cross-hatch shapes are output as boundaries. The first image below shows a shape as displayed after loading a GDSII file created with the Output cross-hatch shapes as segment boundaries option selected. The second image shows the same shape as displayed after loading a GDSII file created without selecting the Output cross-hatch shapes as segment boundaries option.
	  <p>Cross-hatch shape with option selected Cross-hatch shape without selecting option</p>
Output text height as magnification	Specifies that the text height should be output as the magnification value.
Add pin text to top level in hierarchy	Changes the ownership of a text whose parent is a symbol or a pin to that of the top structure (cell) in the GDSII output. The top cell name is specified in the <i>Top structure name</i> field.
Export	Choose to export the layout editor data into a GDSII format using the layer conversion file you specified.
Viewlog	Displays the <i>stream_out.log</i> that contains messages generated during data export, and is available only after you have exported data from the design.
Close	Closes the Stream Out dialog box without running the Stream Out program.

Related Topics

- [Converting an Allegro Design to GDSII Stream Format](#)
- [Editing the Stream Out Layer Conversion File](#)

Stream Out Edit Layer Conversion File

This dialog box displays specifications related to layers and the current mapping of classes and subclasses to stream layers. Initially, displays layer mappings that currently exist in the layer-conversion file display. If the specified layer conversion file is empty, or if it does not exist, all classes or subclasses appear as unmapped.

Select all	<p>Selects or deselects all classes or subclasses in the current design that currently display.</p> <p>⚠️ The BONDING WIRE class is supported in conversion files. The subclasses are the wire profile names from the database. The wires assigned to the mapped wire profile are exported to that layer. Use profile names that are all uppercase and do not contain any special characters, to ensure that the names are also legal in the exported file.</p> <p>The BOND_FINGER class is only available if you have enabled the separation of via and bond finger objects by setting the <code>stream_bond_finger_class</code> variable in user preference.</p> <p>The tool automatically creates a connecting element at each end of the wire. The pads are the same diameter as the wire on the profile layer and on the layer of the element to which the wire connects. Importing of bond wires through these formats is not supported. During import, the BONDING WIRE class entries in the conversion files are ignored. See an example in the <i>GDSII Bi-Directional Manufacturing Interface</i> section in the <i>Transferring Logic Design Data User Guide</i>.</p>
Class filter	Controls which classes appear. Initially, this field defaults to All, and all classes in the current design appear. Enter your own filters, which are added to the existing list for reuse in the current session.
Subclass filter	Controls which subclasses appear. Initially, this field defaults to All, and all subclasses in the current design display. Enter your own filters, which are added to the existing list for reuse in the current session.
Class	Displays the classes you chose using the Class Filter field.
Subclass	Displays the subclasses you chose using the Class Filter field.
layer	Lets you change the stream layer to which a class or subclass is mapped.
Data Type	Displays the data type.
Map selected items	Use these fields to specify the stream layers for mapping to classes and subclasses.
Layer	Selects a stream layer for mapping. Initially this contains only layers read from the specified layer-conversion file. For an empty or new layer-conversion file, no entries appear here.
Data Type	Specifies the numeric value for the data type. This is -1 by default.
Map	Maps chosen classes and subclasses of the current design to selected stream layers. Only items that display and that are chosen are mapped.
Unmap	Clears the mapping for all currently chosen class/subclasses in the grid for selected layers.
Show Selected Layers	Displays only selected layers.
Restore Layer Visibility	Resets layer visibility to the state before the Edit Layer Conversion File was opened.
OK	Writes the current mapping information for subclasses to the layer conversion file. Subclasses for which no mappings are specified is not written out to the conversion file and are consequently not imported into the editor.
Cancel	Cancels input and closes the dialog box.
Help	Displays context-sensitive help.

Related Topics

- [stream out](#)
- [Editing the Stream Out Layer Conversion File](#)

Converting an Allegro Design to GDSII Stream Format

To convert an Allegro design to GDSII stream format, follow these steps:

1. Create positive ETCH/CONDUCTOR subclasses (layers) only.
2. Ensure the voids for positive shapes on the positive etch layers have been voided properly.
3. Run `status` to use the Status tab to verify the current state of dynamic shapes and DRCs and update them if they are out of date.
4. Create a layer conversion file using a text editor to assign a stream layer number (0 - 225) to each class/subclass to be exported.
5. Run `stream out` to display the Stream Out dialog box.

Using Manufacture – Stream Out with the above stream-layer-conversion file, where output file name was `stream_out.cnv`, for example, the log file writes:

Stream_out log file for design streamout.

Layout units are: mils.

User_units are: MILS.

Conversion file is /home/blm/ppaulson/stream_out.cnv.

Stream created with 10 database units per user_unit.

There is no scaling of the output coordinates.

Film FAB will go on layer 10.

Film PRIMARY will go on layer 20.

Film GND1 will go on layer 25.

Film INT1 will go on layer 30.

Film VCC1 will go on layer 35.

Film GND2 will go on layer 40.

Film INT2 will go on layer 45.

Film VCC2 will go on layer 50.

Film SECOND will go on layer 55.

Film PMASK will go on layer 60.

Film PMASK will go on layer 60.

Film SMASK will go on layer 61.

Film PLEGNED will go on layer 62.

streamout created. stream_out completed.

A file called <filename>.sf (in this case, `streamout.sf`) is created that can be read back into your design.

Related Topics

- [stream out](#)
- [Stream Out Dialog Box](#)
- [status](#)
- [Creating a Stream Layer Conversion File Using a Text Editor](#)

Editing the Stream Out Layer Conversion File

You can edit the Stream Out Layer Conversion File or preview data in chosen classes or subclasses before exporting, by following these steps:

1. Click *Edit* on the Stream Out dialog box to display the Stream Out Edit Layer Conversion File dialog box, which displays the current mapping of the classes and subclasses in the layer conversion file to stream layers. If the specified layer conversion file is empty, or if it does not exist, all classes or subclasses appear as unmapped.
2. Enter the classes and subclasses you want to list in the Class filter and Subclass filter fields, respectively. The initial default is All. Filters you enter become part of the drop-down list, which you can reuse in the current session.
3. Use the Layer column to change mappings for subclasses on a one-by-one basis if necessary.
4. Use the Select check box to choose individual classes and subclasses to be mapped, or use Select all to choose all listed classes and subclasses.
5. In the Map selected items section, choose a stream layer for the class and subclass from the Layer field, which contains only layers read from the specified layer conversion file.
6. Click the Map button to complete the mapping for all currently chosen classes and subclasses of the current design to stream layers you choose. Or choose Unmap to clear the mapping for all currently chosen subclasses.
7. Click OK to write current mapping information for layers to the layer conversion file and return to the Stream Out dialog box. Subclasses for which no mappings are specified are not written to the layer-conversion file and therefore are not exported into the editor.
8. Click Export in the Stream Out dialog box to export the data or Close to close the dialog box.

Related Topics

- [stream out](#)
- [Stream Out Dialog Box](#)
- [Stream Out Edit Layer Conversion File](#)

stream padstacks

The `stream padstack` command displays the *Stream Structure Import* dialog box, which lets you import the GDSII files to create the padstack definitions for pins, vias and add the vias to the constraint via list. This composes the routing structures of the interconnect or fanout pieces.

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

Related Topics

- [stream out](#)
- [Creating Padstack Definitions for Pins and Vias](#)

Stream Structure Import Dialog Box

Access Using

- Menu Path: Menu path: *Si Layout – Import GDS Structures*

The following table describes the fields of the *Stream Structure Import* dialog box:

<i>Batch import directory structures</i>	Allows you to import multiple stream files from the directory specified in the <i>Directory</i> field. This check box is selected by default. If is not selected, user can pick a single structure.
<i>Directory</i>	Select the path of the directory to pick up all matching stream files for batch import. This field is enabled only if the <i>Batch import directory structures</i> check box is selected.
<i>Stream file(s)</i>	<ul style="list-style-type: none"> • If <i>Batch import directory structures</i> is selected, specify the wildcard (for example: *.gds) to match the files to be imported. • If <i>Batch import directory structures</i> is deselected, specify the exact file name to be imported.
<i>Layer conversion file</i>	Indicates the name of the conversion file that maps the database to the GDSII stream layers. You can also browse for existing files.
<i>Layer mapping</i>	Displays the Stream In Edit Layer Mapping dialog box.
<i>Scale factor</i>	Indicates how the entries are to be scaled vertically and horizontally. This field is independent of the units value and is applied after any units conversion. For example, a value of 0.5 reduces each entry by 50 percent; a value of 2.0 increases each entry by 100 percent. The default is 1.0.
<i>Objects to Import</i>	Specifies the list of nets, ground, signal, and power, you want to import from the GDS file. By default, all the check boxes are selected in all the columns except for the <i>CLines/Shape (Floating)</i> columns.
<i>Add padstacks to the default via list</i>	If selected, all the defined vias are added to the Constraint Manager vias list. This check box is selected by default.
<i>Derive single layer pin padstacks</i>	If selected, single layer pin padstack definitions are generated for the pads that are mapped to PIN layers or vias with pads on the top or bottom layers.
<i>Import</i>	Imports the GDSII file data.
<i>Close</i>	Closes the dialog box.
<i>Help</i>	Displays the help topic for this command.

Related Topics

- [Stream In Edit Layer Mapping Dialog Box](#)

Creating Padstack Definitions for Pins and Vias

To create the padstack definitions for pins and vias, follow these steps:

1. Choose *Si Layout - Import GDS Structures*.
The Stream Structure Import dialog box is displayed.
2. Do one of the following to import a single or all GDS files:

- To import all matching GDS files, select the directory in which the files exist, in the *Directory* field.
 - To import a single GDS file, deselect the *Batch import directory structures* check box and select a GDS file in the *Stream file(s)* field
3. Select the *.cnv* file in the *Layer conversion file* field.
 4. Select the objects you want to import in the *Objects to Import* table for the available net types.
 5. Click *Import*.

GDS file data is imported and padstack definitions are created for the selected objects.

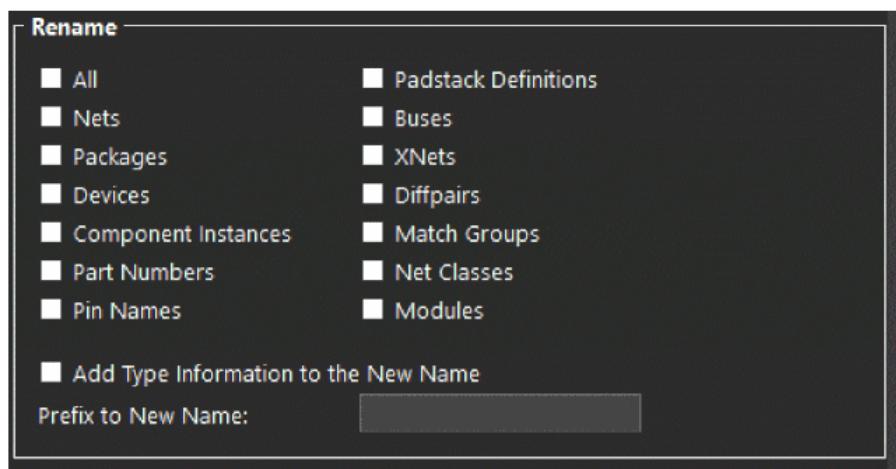
Related Topics

- [Stream In Edit Layer Mapping Dialog Box](#)
- [stream padstacks](#)

strip_design

The `strip_design` command strips Intellectual Property (IP) from the database by renaming or deleting objects from the database. The stripped database can be used to share IPC-2581, artwork, or stream data outputs.

The command renames any or all of the following objects from the database:



The command allows IP data to be deleted based on three options:

- General Delete Options: Removes all or some of the following physical data.

S Commands

S Commands--strip_design

Delete

- All
- Symbol Text (Excluding refdes and pin number)
- User Defined Properties (Including Definitions)
- LOGICAL_PATH and PHYSICAL_PATH property instances
- LIBRARY_PATH property instances
- Constraint Manager based view attachments
- Design Partition History

- Optional Delete Options: Removes physical data in addition to the general options based on the following list.

(These options, if deleted, may [impede Cadence's ability to debug problem](#))

- All
- User Defined Subclasses (Including associated data)
- Text associated with Design Root
- Format Symbols
- Electrical Constraints
- All other Constraint Data (Including DFA)
- Alternate Symbols and TDF
- Front End Properties and Paths
- Attached Signal Models
- Reset cross-section parameters to default
- CM advanced constraints attachment

- Group Delete Options: Removes physical data from the design based on a net selection.

Group Delete

- Delete items currently not visible
- Select the Nets to keep the items
- No Group Delete

 Do not use this option when rename nets is enabled in the *Rename* section.

Related Topics

- [Stripping IP from Design Database](#)

IP Strip Design (for Cadence Testcases) Dialog Box

Access Using

- Menu Path: *File – Export – Strip design*

General		
	<i>Rename</i>	Lists object types to rename
	<i>Delete</i>	Lists physical data to delete
Optional Delete		Additional data to delete from the database. ⚠️ If selected these options may cause error when debugging database.
	<i>Group Delete</i>	Provides options for group delete: <ul style="list-style-type: none">• Delete items currently not visible• Select the Nets to keep the items: Opens Net Selecting for IP Stripping dialog box to choose nets from the list.• No Group Delete
Include old and new name mapping for renamed elements in the log		Enables mapping list of renamed data
Save Settings		Saves selected options to an output configuration file
OK		Starts the strip design process

Stripping IP from Design Database

Save the design before running the command. Follow these steps to strip IP from the design database:

1. Run `strip_design` command.
The IP Strip Design dialog box is displayed.
2. In the *Rename* section of the *General* tab choose database objects and select the following options to rename the objects in the stripped design:

Rename Object	Option Required
To add element type as prefix	Enable <i>Add Type information to the New Name</i> checkbox
To add user-defined prefix	Specify prefix in the <i>Prefix to New Name</i> field
To add both element type and user-defined prefix	Enable <i>Add Type information to the New Name</i> checkbox and specify user-defined name in the <i>Prefix to New Name</i> field

3. In the *Delete* section of the *General* tab, enable options to delete the selected data from the database.
4. If you are interested in deleting additional data, open *Optional Delete* tab and enable the required options.
5. Click *OK* to execute the command.
When completed the database is automatically saved with the suffix _stripped.
6. Choose *File – Viewlog* to view the result of IP stripping in the log file.

Related Topics

- [strip_design](#)

stroke_editor

The `stroke_editor` command is a batch command that launches the Stroke Editor and lets you edit an existing `.strokes` file or create your own `.strokes` file. For additional information on a `.strokes` file, see the *Getting Started with Physical Design* user guide in your product documentation.

You can also use the `stroke editor` command within the layout editor.

Syntax

```
stroke_editor  
<filename  
>
```

Example

```
stroke_editor my_allegro.strokes
```

Related Topics

- [stroke editor](#)

Removing Strokes from a Stroke File

You can remove strokes from a stroke file by following these steps:

1. Run the `stroke editor` command to launch the Stroke Editor.

The Stroke Editor splash screen appears followed by the Stroke Editor window. The Stroke Editor loads the currently active `.strokes` file.

2. Open the file from which you want remove strokes.

The list of existing strokes appears in the List of Strokes Area at the right side of the window.

3. In the List of Strokes Area, click the stroke you want to remove from the file.

The stroke pattern appears in the Graphics Area, and the associated command appears in the *Command* field.

4. Click the right button on the specified stroke in the List of Strokes Area and choose *Delete* from the pop-up menu.

5. Click *Yes* in the Stroke Editor dialog box.

The stroke and associated command are removed from the file.

Related Topics

- [stroke editor](#)
- [Stroke Editor Window](#)
- [Adding New Strokes to a Stroke File](#)

stroke editor

The `stroke editor` command launches the Stroke Editor and lets you edit an existing `.strokes` file or create your own `.strokes` file. For additional information on a `.strokes` file, see the *Getting Started with Physical Design* user guide in your product documentation.

You can also use the `stroke_editor` batch command.

Related Topics

- [Adding New Strokes to a Stroke File](#)
- [Changing Existing Strokes](#)
- [Removing Strokes from a Stroke File](#)

Stroke Editor Window

Access Using

- Menu Path: *Tools – Utilities – Stroke Editor*

<i>File</i>	
<i>Open</i>	Choose this to open a new strokes file.
<i>Close</i>	Choose this to <i>close the existing strokes file</i> .
<i>Save</i>	Choose this to <i>save the strokes file</i> .
<i>Save As</i>	Choose this to <i>save the strokes file with another name</i> .
<i>Exit</i>	Choose this to exit the <i>Stroke Editor</i> .
<i>Help</i>	
<i>Stroke Editor Help</i>	Choose this to get <i>help on the Stroke Editor</i> .
<i>Allegro Help</i>	Choose this to get help on Allegro X PCB Editor.
<i>About Stroke Editor</i>	Choose this to get information about the Stroke Editor.
<i>Command</i>	<i>Type the command in this field to which you are associating the stroke drawn in the Graphics Area.</i>
<i>Add</i>	<i>Click this button to associate the stroke in the Graphics Area and the command defined in the Command field.</i>
<i>Clear</i>	<i>Click this button to remove the existing stroke from the Graphics Area.</i>

Related Topics

- [Changing Existing Strokes](#)
- [Removing Strokes from a Stroke File](#)

Adding New Strokes to a Stroke File

To add new strokes to a stroke file:

1. Run the `stroke editor` command to launch the Stroke Editor.

The Stroke Editor splash screen appears followed by the Stroke Editor window. The Stroke Editor loads the currently active `.strokes` file.

2. Open the file to which you want to add strokes.

3. Click in the Graphics Area.

A small red cross appears, which signifies the starting point of the stroke.

4. Click on the cross and draw the specified stroke; then release the mouse button.

5. Type a command in the *Command* field, and click *Add*.

The stroke and the associated command are listed in the List of Strokes Area.

If a stroke is similar to one already listed in the file, the Resolve Stroke Conflict dialog box appears. It lists the conflicting commands and asks that you choose only one. Once you choose the specified command, the layout editor adds the stroke and associated command to the List of Strokes Area and removes the conflicting stroke and command from the list, if necessary.

6. From the menu bar, choose *File – Save* to save the file.

Related Topics

- [stroke editor](#)
- [Removing Strokes from a Stroke File](#)

Changing Existing Strokes

To change existing strokes:

1. Run the `stroke editor` command to launch the Stroke Editor.
The Stroke Editor splash screen appears followed by the Stroke Editor window. The Stroke Editor loads the currently active `.strokes` file.
2. Choose *File – Open* to open the specified file for editing.
3. In the List of Strokes Area, click the stroke you want to update, then click the right button and choose *Edit* from the pop-up menu.
The stroke pattern appears in the Graphics Area, and the associated command appears in the *Command* field.
4. Edit the stroke and then click *Update* in the Command Area.
The updated stroke appears in the List of Strokes Area.

Related Topics

- [stroke editor](#)
- [Stroke Editor Window](#)

strokefile

The `strokefile` command loads a user-defined file of command strokes into the layout editor so that you can use the customized command strokes in the Workspace Editor. You specify a file of command strokes that you created using the Stroke Editor.

For additional information on creating a `.strokes` file, see the *Getting Started with Physical Design* user guide in your product documentation and the [stroke editor](#) command.

Syntax

```
strokefile <filename>
```

Specifying a File Containing Your Own Strokes

1. Type `strokefile <filename>` at the command window prompt.

The layout editor looks for stroke files in this order: in the current working directory, the `\pcbenv` directory, or in `$cdsroot\share\pcb\text` directory, unless you specify a full path name in the filename argument.

subclass

The `subclass` command changes the *Subclass* field in the *Options* panel of the Control Panel to the subclass you specify. The subclass name can only be one that is recognized as a current subclass of the class displayed in the *Options* panel.

Syntax

```
subclass[+] [--] [subclass_name]
```

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

Examples

The following command changes the subclass to GND.

```
subclass GND
```

The following example uses the [funckey](#) command to create a function alias that increments the subclass to the next subclass of the current class.

```
funckey + subclass --
```

swap

The `swap` batch command executes the automatic swap program in a system window. You must choose necessary parameter options in a design window before execution. If you do not designate an output name for the design, the layout editor overwrites the input design. Check the `swap.log` file for all information related to the processing.

Syntax

```
swap [-version] input_filename output_filename
```

-version	Prints the version.
input_filename	Is the argument for the name of the design to be swapped. Do not type the file extension.
output_filename	Is the optional argument for the name of the file to which you want the swapped design to be written. If you do not enter an output drawing name, the command writes over the <code>input_filename</code> .

swap area design

The `swap area design` command lets you define the package/part keepin as the automatic swapping area.

For more details and the prerequisites for this command, see *Automatic Swapping* in your product documentation.

Access Using

- Menu Path: *Place – Autoswap – Design*

Defining the package/part keepin as the Automatic Swapping Area

1. Run `swap area design`.
The area within the package/part keepin boundary is chosen as the swapping area.
2. To set the automatic swapping parameters and perform automatic swapping, run `swap param`. –or– To perform automatic swapping, run `swap execute`.

swap area list

The `swap area list` command displays the LIST AREA dialog box showing the current active area of the design for automatic swapping. For more details, see the *Placing the Elements* user guide in your product documentation.

Access Using

- Menu Path: *Place – Autoswap – List*

swap area room

The `swap area room` command lets you enter the names of the rooms in your design as the area for automatic swapping.

For more details and the prerequisites for this command, see the *Placing the Elements* user guide in your product documentation.

Access Using

- Menu Path: *Place – Autoswap – Room*

Defining Rooms for Automatic Swapping

To define rooms for automatic swapping, follow these steps:

1. Run `swap area room`.
A dialog box appears that lets you specify a room name.
2. Type the name of a room and click *OK*.
3. Type the name of another room and click *OK*. –or– Click *OK* again without entering a name to close the dialog box.
4. To set the automatic swapping parameters and perform automatic swapping, run [swap param](#). –or– To perform automatic swapping, run [swap execute](#).

swap area window

The `swap area window` command lets you define up to 16 window areas in your design for swapping.

For more details and the prerequisites for this command, see the *Placing the Elements* user guide in your product documentation.

Access Using

- Menu Path: *Place – Autoswap – Window*

Defining Window Areas for Swapping

Follow these steps to define window areas for swapping:

1. Run `swap area window`.
A dialog box appears that lets you specify a room name.
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. If you want to define more windows, repeat steps 2 and 3.
5. When you are finished, choose *Done* from the pop-up menu.
6. To set the automatic swapping parameters and perform automatic swapping, run [swap param](#). –or– To perform automatic swapping, run [swap execute](#).

swap components

The `swap components` command swaps components in a design window.

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 from the pop-up menu.

Valid object:

- Components

In the pre-selection use model, the command is only available if the selection set comprises exactly two components that you have chosen. If you choose components and clines, for example, a warning displays for each invalid element, and the tool ignores it.

You can use the `reports` command to generate the Component Report. For more details, see the *Placing the Elements* user guide in your product documentation.

Related Topics

- [Swapping Components in Pre-Selection Mode](#)
- [Swapping Components in Verb-Noun Mode](#)

Swap Components Command: Options Panel

Access Using

- Menu Path: *Place – Swap – Components*

When you access the command in the pre-selection use model from the right mouse button pop-up menu, the *Maintain Symbol Rotation* option is enabled by default. The *Options* panel is not available for you to change settings.

<i>Comp1</i>	The component (represented as a reference designator) that you want to swap.
<i>Comp2</i>	The component you want to swap with <i>Comp1</i> .
<i>Maintain symbol rotation</i>	Checked by default to retain the original symbol rotation of the component being swapped. When unchecked, preserves the rotation of the components as well.

Related Topics

- [Swapping Components in Verb-Noun Mode](#)

Swapping Components in Pre-Selection Mode

To swap components in pre-selection mode:

1. Choose *Setup – Application Mode – Placement Edit Mode* to access the placement application mode, or right click and choose *Application Mode – Placement Edit*.
2. Choose two components.
3. Right click and choose *Swap Components* from the popup menu.

Related Topics

- [swap components](#)

Swapping Components in Verb-Noun Mode

To swap components in verb-noun mode, perform these steps:

1. Run `swap components`.
2. Click on the component you want to swap. –or– Type the reference designator in the *Comp 1* box on the *Options* panel and press *Enter*.
The component is highlighted.
3. Click on the second component. –or– Type the reference designator in the *Comp 2* box on the *Options* panel.
4. To complete the swap and remain in swap mode, click on the first component of the next swap.
5. When you have finished swapping, choose *Done* from the pop-up menu.

Related Topics

- [swap components](#)
- [Swap Components Command: Options Panel](#)

swap execute

The `swap execute` command runs the automatic swap process. It uses the settings on the Automatic Swap dialog box ([swap param](#) command) and the area defined in the swap area commands ([swap area design](#), [swap area list](#), [swap area room](#), [swap area window](#)).

For more details and the prerequisites for this command, see the *Placing the Elements* user guide in your product documentation.

Running the Automatic Swap Process

1. On the command line, run `swap execute`.

When swapping is complete, you can display the swap log to review any warning or error messages.

swap functions

The `swap functions` command swaps functions or gates in a design window.

You can use the `reports` command to generate the Function Report.

For more details, see the *Placing the Elements* user guide in your product documentation.

Access Using

- Menu Path: *Place – Swap – Functions*

Swapping Functions or Gates in a Design

Follow this procedure to swap functions or gates in a design:

1. Run `swap functions`.
2. Click on any pin that is associated with the first function that you want to swap.

The layout editor highlights the pins and ratsnest lines of the chosen function, as well as the pins of all the functions on the design that can be swapped with the function you picked. The highlighted pins are of the same device and function type.

 If you see a function that you think is swappable although it is not highlighted, choose the function. A message is displayed that explains why that particular function is not swappable.

3. From the highlighted functions, click on a pin from the second function that you want to swap.

The pins of the functions that you are swapping and their ratsnest lines remain highlighted.

1. To complete the swap and remain in swap mode, click on a pin in the first function of the next swap.
2. When you have finished swapping, choose *Done* from the pop-up menu.

swap param

The `swap param` command displays the Automatic Swap dialog box where you can do the following:

- Establish swap parameters that control automatic swapping
- Execute the automatic swapping process for functions and pins

You can define up to 10 passes for automatic swapping. The program completes each swap pass by running the function `swap` first, then the pin `swap`.

Cadence recommends setting a high number for each swap time so enough time elapses to perform the necessary swaps. Each pass ends when either time runs out or no logical swap candidates are found. The program automatically moves to the next pass when it has completed all appropriate swaps for a given pass.

For more details and the prerequisites for this command, see the *Placing the Elements* user guide in your product documentation.

Related Topics

- [Swapping Automatically](#)

Automatic Swap Dialog Box

Access Using

- Menu Path: *Place – Autoswap – Parameters*

Use this dialog box to set parameters for up to 10 swapping passes. By default, two swap passes are set with a time limit of 60 minutes each.

Set a high time limit to allow completion of the swap pass. When one pass is complete, the next pass starts automatically.

<i>Swap pass</i>	Indicates the number of the swap pass.
<i>Function time</i>	Indicates the time, in minutes, for the function swap pass. The default is 60 minutes for passes 1 and 2, 0 minutes for passes 3 through 10. The function swap is executed before the pin swap.
<i>Pin time</i>	Indicates the time, in minutes, for the function swap pass. The default is 60 minutes for passes 1 and 2, 0 minutes for passes 3 through 10.
<i>Inter-room</i>	Permits swapping between rooms for the swap pass.
<i>Swap</i>	Saves the settings and runs automatic swapping.
<i>Close</i>	Saves the settings and closes the dialog box.

Swapping Automatically

To swap the swap components automatically:

1. Run `swap param`.
The Automatic Swap dialog box appears.
2. For each swap pass that you want to use, specify the maximum time allowed and indicate whether you want the layout editor to perform swaps between rooms.
3. Click *Swap* to apply the parameters and swap the components. –or– Click *Close* to apply the parameters and close the dialog box.

If you choose *Swap*, all swappable function pairs are examined, then all swappable pin pairs. The process continues to search for eligible swaps that shorten the total design wire length until it either runs out of time or finds no more suitable swap candidates. When swapping pins on ECL nets, automatic swap maintains the correct ECL scheduling.

Related Topics

- [swap param](#)

swap pins

The `swap pins` command lets you swap pins when their names occur in the same PINSWAP statement of their device file.

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 from the pop-up menu.

Valid object:

- Pins

In the pre-selection use model, the command is only available if the selection set comprises a single swappable pin. Pins eligible to be swapped with this single pin become highlighted, and the tool prompts you to choose one of them. If the selection set comprises objects ineligible for swapping, a warning displays for each invalid element, and the tool ignores it.

Co-design Environment

During co-design, the `swap pins` command in APD modifies the component definition by swapping the full logical pin to the physical pin assignment. When complete, the `VERILOG_PORT_NAME` property on the component definition pin must match `VERILOG_PORT_NAME` property on the pin for the co-design die. However, when you swap a pin in APD, the tool does not immediately perform the corresponding swap in the IC design database. Therefore the `VERILOG_PORT_NAME` property on the component definition pin is not swapped. To show that the pin swap has been initiated in APD, the tool swaps the Verilog port names onto the `VERILOG_PORT_NAME` property on the pins. When you see that the `VERILOG_PORT_NAME` property on the pin differs from the `VERILOG_PORT_NAME` property on the component definition pin, it means that a swap has been initiated, but not completed in IO Planner (IOP).

Cadence recommends that you then use the `die editor` command in APD to start up IOP. Use the `deleteBumps` command to remove the old bumps and assignments. Manually load the `.io` file generated by APD into IOP using the `loadIoFile <refdes>.io` command to send the pin swaps from APD to IOP. Then click the *Redraw* icon.

 If you set the `ICP_SEND_IO_TO_IOP` environment variable, IOP automatically loads the IO file and the pin swaps in IOP.

The next time you use the IOP *Update Package* command, it finishes updating the die representation in APD. At that time, the tool swaps the `VERILOG_PORT_NAME` property on the component definition pin and removes the `VERILOG_PORT_NAME` property on the pin. This indicates that you have completed the pin swap operation in IOP and confirmed it by updating in APD. When this swap process is complete, you can generate the chips view in APD and you will see that the pins have swapped.

Also, in a co-design environment, any pin can be swapped with any other pin of the co-design die regardless of pin use or swap code.

You can use *Tools – Reports (reports)* command to generate the Component Pin report.

For more details, see the *Placing the Elements* user guide in your product documentation.

Related Topics

- [Swapping Pins in Pre-Selection Mode](#)
- [Swapping Pins](#)
- [Swapping Pin Pairs of Two Differential Pair Nets](#)
- [Swapping Pins of a Single Differential Pair Net](#)

Swap Pins Command: Options Panel

Access Using

- Menu Path (Allegro X PCB Editor): *Place – Swap – Pins*
- *Menu Path* (Allegro Package SI L): Place – Swap Pins

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

<i>Active Class and Subclass</i>	Specifies the layer on which the pin you want to exchange exists.	
<i>Swap properties with pins</i>	Specifies that any properties assigned to the pin remain attached when swapping. Enabled by default in the layout editor, but deselected in your Allegro Package product.	
<i>Ignore FIXED property</i>	Specifies that the layout editor ignores the FIXED property letting you move the chosen pin. Enabled by default in the layout editor, but deselected in your Allegro Package product.	
<i>Ripup on swap</i>	Specifies that the layout editor removes etch/conductor during pin swapping. Enabled by default.	
<i>Diff Pair Swap</i>	When checked, differential pair pin swapping is enabled using the mode designated by one of the following options.	
	<i>Swap Two Pairs</i>	Specifies swapping two pins at one end of a differential pair with two pins at one end of another differential pair.
	<i>Swap Polarity</i>	Specifies swapping the negative and positive pins at one end of the same differential pair.

Related Topics

- [Swapping Pins](#)
- [Swapping Pin Pairs of Two Differential Pair Nets](#)
- [Swapping Pins of a Single Differential Pair Net](#)

Swapping Pins in Pre-Selection Mode

To swap pins in pre-selection mode, follow this procedure:

1. Choose *Setup – Application Mode – Placement Edit Mode* to access the placement application mode, or right click and choose *Application Mode – Placement Edit*.
2. Choose the first pin to exchange.
3. Right click and choose *Swap Pins* from the popup menu.

The chosen pin, ratsnest lines, and all pins available to swap become highlighted.

To determine if a pin is swappable although it is not highlighted, click to choose the pin. The command window prompt displays an explanation of why the pin is not swappable.

1. From the highlighted pins, choose the second pin to exchange.
All circuit elements unhighlight, except the two pins that are swappable and the ratsnest lines.
The chosen pins swap positions.

Related Topics

- [swap pins](#)
- [Swapping Pin Pairs of Two Differential Pair Nets](#)
- [Swapping Pins of a Single Differential Pair Net](#)

Swapping Pin Pairs of Two Differential Pair Nets

You can swap pin pairs of two differential pair nets by following these steps:

1. Choose *Place – Swap – Pins* (the layout editor) or *Place – Swap Pins* (Allegro Package S1 L).
2. In the Options panel, enable (check) the *Diff Pair Swap* option, then select *Swap Two Pairs* mode.
3. Choose a pin of a differential pair net to indicate the initial pin pair to swap.

The chosen pin, ratsnest lines, and eligible pins of other differential pair nets to swap with become highlighted.

 Eligible pins are those that belong to the same swap group and have the same polarity as the chosen pin. In cases where conflicting differential pair pin definitions exist, Allegro determines the swap eligibility using a precedence rule.

 To determine why a pin is not swappable, click on the pin. The command console displays a brief explanation.

4. From the highlighted pins, choose a pin of another differential pair net to indicate the pin pair you intend to swap with.
The two pin pairs of the differential pair nets are swapped and the result is displayed but not yet committed.
5. To swap other differential pin pairs, right click, choose *Next* from the popup menu, then repeat steps 3 and 4.
- or -
Right click and choose *Done* from the popup menu to commit the results and exit the `swap pins` command.

Related Topics

- [swap pins](#)
- [Swap Pins Command: Options Panel](#)
- [Swapping Pins in Pre-Selection Mode](#)

Swapping Pins of a Single Differential Pair Net

Perform these steps to swap pins a single differential pair net:

1. Choose *Place – Swap – Pins* (the layout editor) or *Place – Swap Pins* (Allegro Package S1 L).
2. In the Options panel, enable (check) the *Diff Pair Swap* option, then select *Swap Polarity* mode.
3. Choose a pin at one end of a differential pair where you intend to swap pin polarity.
The polarity of the pin pair is swapped and the result is displayed but not yet committed.
4. To select another differential pair net to pin swap, right click, choose *Next* from the popup menu, then repeat step 3.
- or -
Right click and choose *Done* from the popup menu to commit the results and exit the `swap pins` command.

Related Topics

- [swap pins](#)
- [Swap Pins Command: Options Panel](#)
- [Swapping Pins in Pre-Selection Mode](#)
- [Swapping Pins](#)

symbol

The `symbol` command is used in conjunction with an active command to choose an individual symbol for manipulation by the active command.

Choosing a Symbol to Execute Active Command on it

Follow this procedure to choose a symbol to execute the active command on it:

1. Run a command; for example, `move`.
2. Type in `symbol`, followed by a reference designation at the command window prompt.
The design window refocuses by zooming in and centering on the chosen symbol.
3. Manipulate the element according to the active command.
4. To complete the command, choose *Done* from the right button pop-up menu.

symbol_check

The `symbol_check` command generates a report that lists the availability of unplaced symbols and their location on disk.

Clicking the Unplaced Symbols color box on the Status tab, accessed by running the `status` command, also produces the Unplaced Symbol Availability Check report, an example of which appears below.

```
(-----)
(
(     Unplaced Symbol Availability Check
(
(
(     Drawing      : ls.brd
(
(     Software Version : 15.x )
(
(     Date/Time      : Thu Mar 19 10:16:56 2006 )
(
(
(-----)
Current PSMPATH consists of:
.
symbols
..
../symbols
D:\Work\work\share\local\pcb\symbols
D:\Work\work\share\pcb\pcb_lib\symbols
D:\Work\work\share\pcb\allegrolib\symbols
All Symbols Found
Total symbols placed:    0 out of 0
```

symboledit

The `symboledit` command activates the Symbol Edit application mode that enables you to easily edit changeable symbols, such as BGAs, in a design. When you are in the Symbol Editor application mode, you can perform operations on symbols from the options in the context-menu.

The Symbol Edit application mode configures the tool for a specific task by populating the right mouse button pop-up menu only with commands that operate on the currently selected element(s). This customized environment maximizes productivity when you use multiple commands on the same design elements or those in close proximity in the design.

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 pop-up menu. This pre-selection use model lets you easily access commands based on the design elements you have 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.

- ⚠ In the pre-select use model, the command ends after the current operation completes. However, the post-select use model lets you perform the command on more than one design element until you choose *Done*. Note that for commands that have an *Apply Changes* or similar button associated, *Done* and *Cancel* will rollback the action if chosen before clicking the button to commit the changes.

In addition to the commands associated with different elements, this application mode also allows you to copy symbols or parts. When you generate parts, a `.dra` file is created for the symbol along with any dependencies such as padstack files in case of a BGA, for instance.

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.

- ⚠ By default, the Symbol Edit application mode does not allow you to edit symbols with the *LOCKED* property. To be able to edit symbols with the *LOCKED* property, set the `symed_allow_locked_comp_edits` variable in the `Symbol_editor` category under `lc_packaging` in the User Preferences Editor dialog box (`Setup – User Preferences`).

For more information on using the Symbol Edit application mode, see the *Getting Started with Physical Design* user guide in your documentation set.

Related Topics

- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symboledit Dialog Boxes and Options Panel

Access Using

- Menu path: *Setup – Application Mode – Symbol Edit*

• Icon:



Symbol Edit application mode - Add component: Options Panel	Symbol Edit application mode - Add Driver: Options Panel	Symbol Edit application mode - Add Keepin/Keepout: Options Panel
Symbol Edit application mode - Bump/Ball attributes: Options Panel	Symbol Edit application mode - CTE Compensation: Options Panel	Symbol Edit application mode - Die properties: Options Panel
Symbol Edit application mode - Edit Boundary: Options Panel	Symbol Edit application mode - Grid Add: Options Panel	Symbol Edit application mode - Pin Add: Options Panel
Symbol Edit application mode - Pin Move/Copy/Modify: Options Panel	Symbol Edit application mode - Pin Numbering settings: Options Panel	Symbol Edit application mode - Pin Pitch Settings: Options Panel
Symbol Edit application mode - Pin Text Settings: Options Panel	Symbol Edit application mode - Place Driver: Options Panel	Die Abstract Write Dialog Box
Refresh Co-Design Die: Options Panel	Refresh Co-Design Die Finish Form	Net for Component Pin Dialog Box
Driver Move: Options Panel	Align Driver: Options Panel	Respace Driver: Options Panel
Swap Driver: Options Panel		

Related Topics

- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Add component: Options Panel

Ref des	Specifies the reference designator for the component
Symbol name	Specifies the symbol name. By default, this is the same as the <i>Ref des</i> value.
Comp type	Specifies the component type. The values available are: <i>Die/Interposer</i> (the default), <i>Discrete</i> , <i>Package</i> , and <i>Plating Bar</i> .
Upper right	Specifies the upper-right corner of the component as X and Y coordinate values.
Lower left	Specifies the lower-left corner of the component as X and Y coordinate values.
Pad layer	Specifies the pad layer. The default value is the top substrate surface. Available if <i>Die/Interposer</i> or <i>Package</i> is selected as <i>Comp type</i> .
Chip attachment	Specifies whether the die is <i>Wire Bond</i> or <i>Flip-Chip</i> (the default). Available if <i>Die/Interposer</i> is selected as <i>Comp type</i> .
Orientation	Specifies the orientation of the die with respect to the package. <i>Chip down</i> is the default selection if <i>Flip-Chip</i> is selected as Chip attachment. <i>Chip up</i> is the default selection for <i>Wire Bond</i> . Available if <i>Die/Interposer</i> is selected as <i>Comp type</i> .
Create component	Creates the component with specified configuration.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Add Driver: Options Panel

 The option to add a driver in the Symbol Edit application is available only for co-design dies in Allegro X Advanced Package Designer (APD).

Bubble	Specifies the bubble mode to use when placing drivers. This can be one of <i>Clockwise</i> , <i>Counter-Clockwise</i> , <i>Both</i> , or <i>None</i> . The bubble determines what drivers are pushed or shoved to prevent overlaps with the driver being modified when it is placed into its new position. Default is <i>None</i> .
Mirror	Specifies the mirror type to use when the driver is placed. Defaults to not-mirrored for un-mirrored dies (for chip-up wirebond) or mirrored for chip-down dies such as a flip-chip, chip down die.
Rotation	Specifies the rotation to use when the driver is placed. Defaults to <i>Automatic</i> . In addition to North, South, East, and West, you can specify <i>Automatic</i> and <i>Maintain</i> . Automatic rotates to face the nearest die side and Maintain keeps the same rotation relative to the closest die side.
Size	Displays the size of the driver definition (or set of definitions) currently on the cursor. This is useful for hand-computing the placement for the reference pick.
Current LEF Library	Shows the currently active LEF library. You can choose to activate a different LEF library, which will update the available driver definitions tree.
LEF/DEF Library Manager	Opens the LEF Library Manager to update or reconfigure the LEF libraries and, as a result, driver definitions, prior to selecting those to be placed from the tree view.
Available Definitions	Shows all the driver definitions in the current LEF library in a tree view. Select the driver definitions to be added. As each definition is clicked, it is added to the cursor to the immediate right of the last selected driver, to allow for multiple drivers to be added at the same time.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Element Selection in Find Filter and Corresponding Symbol Edit Tasks](#)

Symbol Edit application mode - Add Keepin/Keepout: Options Panel

Class	Select from <i>ROUTE KEEPOUT</i> , <i>VIA KEEPOUT</i> , or <i>COMPONENT KEEPOUT</i> to create a keepout for routes, vias, or components. <i>ROUTE KEEPOUT</i> is selected by default.
Subclass	Select the layer to apply the keepout. <i>ALL</i> is selected by default.
Offset Top/Bottom/Left/Right	Specify the offset of the keepout in relation to the Top, Bottom, Left, or Right of the selected component or symbol. The default value is <i>0UM</i> for all the offsets.
Upper right X/Y	Specify the coordinates for the upper-right corner of the keepout. By default, the upper-right corners of the keepout and the component will be same.
Lower left X/Y	Specify the coordinates for the lower-left corner of the keepout. By default, the lower-left corners of the keepout and the component will be same.
Add to symbol	Click to add a keepout with the specified settings.
Finish adding outlines	Click to end the command.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Bump/Ball attributes: Options Panel

Remove pin customizations	Removes existing customizations for selected pins. If set for a component, removes customizations for all pins on the component.
Dmax	Specifies the maximum diameter for the solder bumps or solder balls. (If the value of Dmax is set to 0, solder bumps or solder balls will not be modeled.) ⚠️ Using a value that is too large risks solder bump overlap. A value of zero for Dmax indicates that the bumps are not modeled.
D1	Specifies the bottom diameter of the solder bumps or solder balls. ⚠️ This value must be less than or equal to Dmax.
D2	Specifies the top diameter of the solder bumps or solder balls. ⚠️ This value must be less than or equal to Dmax.
Height	Specifies the height of the bumps or the balls.
Conductivity	Specifies the conductivity for the solder bumps or solder balls.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - CTE Compensation: Options Panel

<i>Ignore fixed property</i>	Enables adjusting the coefficient of thermal expansion (CTE) regardless of the <code>FIXED</code> property attached to the component.
<i>Apply only to selected symbol instance</i>	Applies CTE adjustment only to the selected instance of the symbol in the design.
<i>Stretch routing</i>	Stretches the end point of any connected clines to the new location of the pins. Vias connected to the pins are also moved to keep them centered on the pin, but their connected clines are stretched. If routing structures are used for connecting from the pins, the structure slides with the first connection to the other end of the structure that is stretched. If this option is disabled, the clines, vias, and structures are not moved.
<i>CTE expansion</i>	Specifies the thermal expansion for the component in the X and Y dimensions.
<i>Create new symbol definition</i>	Creates a new symbol definition and updates its CTE compensation, instead of applying the CTE adjustment to the definition of the selected symbol.
<i>Create ghost pins</i>	Creates ghost pins. These are non-intelligent images of the pins at their original locations and are placed on a non-electrical layer. Ghost pins are used as reference points when analyzing the internals of the die, such as RDL routing, macros, drivers, or blockages.
<i>Apply</i>	Applies the CTE adjustment to the symbol as defined.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Element Selection in Find Filter and Corresponding Symbol Edit Tasks](#)

Symbol Edit application mode - Die properties: Options Panel

<i>Ref des</i>	A read-only field that provides the instance name of the chosen die.
<i>Device</i>	A read-only field that specifies the device name for the chosen die.
<i>Die extents</i>	
<i>North scribe</i>	Specifies the amount that the physical die is larger than the represented extents on the North side of the die.
<i>South scribe</i>	Specifies the amount that the physical die is larger than the represented extents on the South side of the die.
<i>East scribe</i>	Specifies the amount that the physical die is larger than the represented extents on the East side of the die.
<i>West scribe</i>	Specifies the amount that the physical die is larger than the represented extents on the West side of the die.
<i>Shrink</i>	A read-only field indicating the percentage of the original IC design size that the die instance has in the package design. For example, a 10% shrink means that the resulting die will only be 90% of the size of the original die.
<i>Attachment type</i>	
<i>Wire bond/Flip-chip</i>	Indicates the current chip attachment type for the die. You can alternate between these two selections.
<i>Apply Changes</i>	Saves the changes made to the settings for this die. The dialog box remains open to additional editing.
<i>Reset Symbol Properties</i>	Resets the die back to the state before you made any changes.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Edit Boundary: Options Panel

Upper Right	
X/Y	Specifies the upper right coordinates for the symbol's bounding box. If the symbol is rectangular, then editing these settings makes all the necessary updates for the new symbol extents. Extents must always completely surround all pins and drivers belonging to the referenced component.
Lower Left	
X/Y	Specifies the lower left coordinates for the symbol's bounding box. If the symbol is rectangular, then editing these settings makes all the necessary updates for the new symbol extents. Extents must always completely surround all pins and drivers belonging to the referenced component.
Update symbol extents	Confirms the new symbol extents specified above.
Select symbol outline	Selects interactively the shape in the drawing that becomes the symbol's new outline. This shape should already exist where it should be for the symbol instance it belongs to. This allows you to make changes to the symbol instance's outline, such as to add notches, using the standard shape editing tools, then update the symbol definition to reflect these changes.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Grid Add: Options Panel

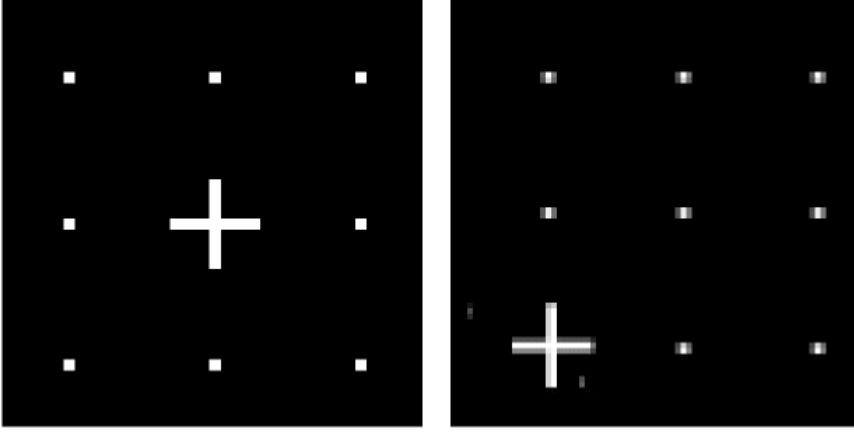
Name	Specifies the name assigned to the grid. The name must be unique. ⚠ The name <i>Base</i> is reserved for the base grid.
Priority	Specifies the priority for the grid. If two grids overlap, the higher priority grid is used for snapping in the overlap region. This defaults to the next lowest available priority. The base grid always has the lowest priority.
Autoconfigure from selected pins	Configures priority, X/Y pitch, edge insets, and inset corner based on the configuration of the selected pins.
Staggered grid	Determines whether the grid is staggered or full. A staggered pin grid has every second grid location removed.
X Pitch	Determines the actual spacing between grid points in the X-axis. The default is 1 database unit.
Y Pitch	Determines the actual spacing between grid points in the Y-axis. The default is 1 database unit.
Edge inset	Specifies the distance from the corner specified in <i>Inset corner</i> to the first legal grid point. For example, you might want the base grid to have an edge inset of 100um to keep pins from getting too close to the edge of the component. The default value is 0 database units, meaning the connect point on pins (most commonly the center) can be placed right up to the grid boundary.
Inset corner	Specifies the corner from which the edge inset is measured. Choices are top left, top right, bottom left, and bottom right. This defaults to the bottom left.
Snap grid extents to base grid	Snaps the grid extents to base grid.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Pin Add: Options Panel

Pin configuration	Shows the options for pin configuration.
Replace existing pins	Replaces existing pin in grid location with the pin being moved, added, or copied. If this option is not checked, the pin being moved, added, or copied is deleted if a pin exists at the grid location. Is not checked by default.
Rip up routing	Rips up connected clines and vias, including bond wires, when a pin is moved or deleted. Is not checked by default.
Stretch routing	Stretches all connected clines and vias, including bond wires, to maintain connection to a pin when it is moved. Is checked by default.
Number	Specifies the pin number for a single new pin. If multiple pins are selected, user will be prompted for a unique number for each pin. If the pin is placed in an auto-numbered grid, this value is not used.
Name	Specifies the logical name of the pin. By default, the value is same as the pin number for pins that are not power or ground pins. If database is associated with a logical design, existing names are not changed. In addition, for co-design dies, pin names must be selected from a list of unused port names.
Pin use	Specifies the pin use. By default, <i>B/I</i> for bi-directional is specified. For multiple pins, the pin use cannot be changed and ** is displayed. If design is associated with a logical design, existing pin use cannot be changed.
Swap code	Specifies the swap code for the pin. Pins with the same code must have compatible pin uses. The swap code 0 means a pin is not swappable. If pins are selected from multiple swap groups, ** is shown. If design is associated with a logical design, existing pin swap codes cannot be changed.
Net	Specifies the net to be assigned to the pin. Click the browse button to open the Net for component pin window and select a net. For multiple pins** is displayed.
Rotation	Rotates the pin when placed at current location. In addition to North, South, East, and West, you can specify Automatic and Maintain. Automatic rotates to face the nearest die side and Maintain keeps the same rotation relative to the die side the pin is closest to.
Padstack	Specifies the padstack to use for the pin. You can select from the list or click the browse button to define a new padstack or browse the library for an existing.
Pattern definition	Shows the options for pattern definition. You can use the options to define a pattern and add the pattern to a group, enabling group operations on the set of pins. The remaining fields defined for this <i>Options</i> panel are visible and available for edit if you select <i>Pattern definition</i> .
Add pins to user group	Adds the pins to a group. You can specify a group name. The default name is of the pattern <component_name>_PATTERN_<n>, where component_name is the name of the selected component and n is a number starting from 1. Selected by default.
Pattern style	Specifies the pin pattern style. You can select one of the following patterns: <ul style="list-style-type: none"> ◦ <i>Single</i>: Select to place pins either by pick or by window. When windowing, the area will be filled with pins at the pitch specified for the grid in that area. ◦ <i>Array</i>: Select to place an array of pins with pitch specified in the Options pane. ◦ <i>Ring</i>: Select to place pins as rings, the pitch and numbers for which is specified in the Options pane. ◦ <i>Text File</i>: Select to create pins at a set of X and Y coordinates listed in the text file. ◦ <i>Spreadsheet</i>: Select to import pin pattern defined in a spreadsheet. The default is <i>Single</i> .

Pattern origin	<p>Specifies the pattern origin in terms of <i>Cursor Center</i> (the default), <i>Cursor Lower-Left</i>, <i>Comp Center</i>, and <i>Comp Lower-Left</i>. The following image shows the pattern origin with <i>Cursor Center</i> and <i>Cursor Lower-Left</i>, respectively, for a 3X3 array of pins.</p>  <p>Comp Center places the pins centered on the component center, while Comp Lower-Left places the pins on the lower-left of the component.</p>
X offset	Specifies the offset of the pattern from the cursor in terms of the X coordinate. The default value is <code>0UM</code> . For example, if you specify <i>Comp Center</i> for Pattern origin and <code>2000UM</code> for X offset, the pin pattern will be placed at an offset of <code>2000UM</code> along the X axis from the component center.
Y offset	Specifies the offset of the pattern from the cursor in terms of the Y coordinate. The default value is <code>0UM</code> .
Array size (in pins)	Specifies the array size in terms of <i>Horizontal</i> and <i>Vertical</i> pins. The default value is 2 for both Horizontal and Vertical. Available only for <i>Array</i> pattern.
Array pin pitch	<p>Specifies the <i>Vertical</i> and <i>Horizontal</i> pin pitch. The default value is <code>0.02UM</code> for both Horizontal and Vertical. For both Horizontal and Vertical, you can set:</p> <ul style="list-style-type: none"> ◦ <i>Increase by</i>: Set to increase the pitch by a specified amount after a specific number of pins, as specified in <i>After every</i>. ◦ <i>After every</i>: Specify the number of pins after which pitch should be increased. <p>The increase in pitch is applied from the pattern origin. The default value for Increase by is <code>0M</code> and default for After every is 1 pins. Available only for <i>Array</i> pattern.</p>
Outer ring size (in pins)	Specifies the number of Horizontal and Vertical pins in the outermost ring. The default value is 3 for both Horizontal and Vertical.
Ring count	Specifies the number of rings. The value you specify may be modified depending on the outer ring size specified by you. The default is 1.
Ring pitch	Specifies the pitch of the rings, which is the distance between each consecutive ring. The default is <code>0.02UM</code> .
Diagonal pitch	Specifies the diagonal pitch. The value is calculated based on the ring pitch and pin pitch. If you change the calculated value, the pin and ring pitch will be changed automatically. The default is <code>0.03UM</code> .

Stagger pins	<p>Stammers the pins in the pattern. Not selected by default. Available for Array and Ring patterns. If you select Array pattern, you can set:</p> <ul style="list-style-type: none"> ◦ <i>Step count</i>: Specify the step count. The following image shows a 5X5 array with Step count set to 2, 3, and 4, respectively.  <ul style="list-style-type: none"> ◦ <i>Start with staggered position</i>: Starts the pattern with a staggered position. Not selected by default.
Exclude corners pins	Removes corner pins from the pattern. Not selected by default.
File name	Specifies the file to import pin pattern from. For spreadsheets with multiple worksheets for different components, you can select a worksheet to import using the list. Available for <i>Text File</i> and <i>Spreadsheet</i> patterns.
Highlight nets with cell color	Highlights nets with the color of the cell in the spreadsheet. Available for <i>Spreadsheet</i> pattern.
Pin pitch	<p>Specifies the pin pitch. For <i>Ring</i> pattern style, the pitch is for the pins on each ring. For text file and spreadsheet, you can set:</p> <ul style="list-style-type: none"> ◦ <i>Vertical</i>: Specify the vertical pin pitch. Default is 0.02UM. ◦ <i>Horizontal</i>: Specify the horizontal pin pitch. Default is 0.02UM. ◦ <i>Diagonal</i>: Specify the vertical pin pitch. Default is 0.03UM. <p>Available for Ring, Text File, and Spreadsheet patterns.</p>

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Pin Move/Copy/Modify: Options Panel

Replace existing pins	Replaces existing pin in grid location with the pin being moved, added, or copied. If this option is not checked, the pin being moved, added, or copied is deleted if a pin exists at the grid location. Is not checked by default.
Rip up routing	Rips up connected clines and vias, including bond wires, when a pin is moved or deleted. Is not checked by default.
Stretch routing	Stretches all connected clines and vias, including bond wires, to maintain connection to a pin when it is moved. Is checked by default.
Number	Specifies the pin number for a single new pin. If multiple pins are selected, user will be prompted for a unique number for each pin. If the pin is placed in an auto-numbered grid, this value is not used.
Name	Specifies the logical name of the pin. By default, the value is same as the pin number for pins that are not power or ground pins. If database is associated with a logical design, existing names are not changed. In addition, for co-design dies, pin names must be selected from a list of unused port names.
Pin use	Specifies the pin use. By default, <i>Bi</i> for bi-directional is specified. For multiple pins, the pin use cannot be changed and ** is displayed. If design is associated with a logical design, existing pin use cannot be changed.
Swap code	Specifies the swap code for the pin. Pins with the same code must have compatible pin uses. The swap code 0 means a pin is not swappable. If pins are selected from multiple swap groups, ** is shown. If design is associated with a logical design, existing pin swap codes cannot be changed.
Net	Specifies the net to be assigned to the pin. Click the browse button to open the Net for component pin window and select a net. For multiple pins ** is displayed.
Rotation	Rotates the pin when placed at current location. In addition to specifying rotation as <i>North</i> , <i>South</i> , <i>East</i> , and <i>West</i> , you can also type the required rotation (in degrees) into this field. Additionally, you can also specify <i>Automatic</i> , <i>Maintain</i> , or <i>Comp Center</i> behavior for the pin rotation. <i>Automatic</i> rotates to face the nearest die side and <i>Maintain</i> keeps the same rotation relative to the die side the pin is closest to. <i>Comp Center</i> rotates the rectangular pins towards the center of the component.
Padstack	Specifies the padstack to use for the pin. You can select from the list or click the browse button to define a new padstack or browse the library for an existing.
Auto select related net	Selects the related net if there is a one to one mapping between nets; for example, if a die net mapped to a single package net is selected, the related net in the package is selected. If more than one nets are related, displays **. ⚠ Available only for co-design die.
IC Net	Specifies the IC net to be assigned to the component pin. Click the browse button to open the Co-design net for component pin window and select a net. For multiple pins ** is displayed. ⚠ Available only for co-design die.
LEF Macro	Select to pick new cell master for a pin when modifying it. ⚠ This impacts only die footprint pad as compared to the Padstack field that impacts the package pad for a pin. ⚠ Available only for co-design die.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Pin Numbering settings: Options Panel

Name	Identifies the grid under edit.
Priority	Shows the priority of the grid under edit. This field cannot be edited.
Scheme	Specifies the pin numbering scheme to be used. This ranges from <i>Customized</i> (user must manually enter every pin name), to <i>Inherit from Base Grid</i> , all the way to patterns like <i>serpentine</i> and <i>spiral numbering</i> .
First pin	Specifies the corner of the grid where pin numbering should start for any auto-numbering pattern like horizontal or vertical. This defaults to top left.
Prefix	Specifies a text prefix to add to the pin label. This defaults to be an empty string.
Start at	Specifies the first pin number to be used, based on the scheme and prefix. If multiple grids in the component use the same numbering scheme, change the start at number to prevent duplicate physical pin numbers. For example, if you have one grid that starts at A1, your second grid might need to start at B13.
Pad letters (As)	Pads alphabet string so that all pin numbers are the same length if using a numbering scheme that contains alpha characters. Defaults to off.
Pad numbers (0s)	Pads numeric string so that all pin numbers are the same length if using a numbering scheme that contains numeric characters. Defaults to off.
Label with letters before numbers	Brings the alphabet part of the pin number before the numeric part. This defaults to on.
Omit letters per JEDEC standard	Removes letters from the alphabet sequence, such as I and O, to prevent confusion with their similar-looking numeric values (1/0). This defaults to off.
Label unused grid positions	If this is disabled, for patterns like horizontal, vertical, and spiral, if a grid position has no pin, then its label index is used by the next location that actually has a pin. Defaults to on.
Label non-staggered positions	Reserves pin numbers for the non-staggered positions, even though no pin can ever be placed there. This makes for more easily understandable numbering sequences for some patterns, such as serpentine. Defaults to off.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Pin Pitch Settings: Options Panel

Name	Specifies the name assigned to the grid. The name must be unique. ⚠️ The name <i>Base</i> is reserved for the base grid.
Priority	Shows the priority for the grid. If two grids overlap, the higher priority grid is used for snapping in the overlap region. This defaults to the next lowest available priority. The base grid always has the lowest priority.
Staggered grid	Determines whether the grid is staggered or full. A staggered pin grid has every second grid location removed.
X Pitch	Determines the actual spacing between grid points in the X-axis. The default is 1 database unit.
Y Pitch	Determines the actual spacing between grid points in the Y-axis. The default is 1 database unit.
Edge inset	Specifies the distance from the corner specified in <i>Inset corner</i> to the first legal grid point. For example, you might want the base grid to have an edge inset of 100um to keep pins from getting too close to the edge of the component. The default value is 0 database units, meaning the connect point on pins (most commonly the center) can be placed right up to the grid boundary.
Inset corner	Specifies the corner from which the edge inset is measured. Choices are top left, top right, bottom left, and bottom right. This defaults to the bottom left. ⚠️ The extra space for partial pin pitch left out if the grid outline is not an even multiple of the calculated value is in the opposite corner from the specified inset corner.
Keep relative pin spacing	Maintains relative spacing of pins.
Calculate values using current grid pins	Calculates and updates the X/Y pin pitch and edge offset values for the form fields based on pins on the grid.
Apply changes	Saves your changes.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Pin Text Settings: Options Panel

Pin Text	
Add text labels on pins	Determines whether pin text is created for each individual pin.
Offset X	Defines the offset of the text from the pin pad origin. The default is 0. This offset is applied globally to all pin number text on the pins. If multiple offsets already exist for the symbol under edit, this will display ** and text positioning will be unchanged.
Offset Y	Defines the offset of the text from the pin pad origin. The default is 0. This offset is applied globally to all pin number text on the pins. If multiple offsets already exist for the symbol under edit, this will display ** and text positioning will be unchanged.
Text Size	Specifies the text size to use. Displays ** if multiple sizes are currently in use. Text sizes are listed as block index plus height x width for easy reference and picking of the right size.
Border Text	
Enable border numbers	Determines the creation of border pin text. To avoid multiple entries at a given location, this field cannot be enabled if multiple pin numbering grids are defined, unless they all inherit from the base grid.
Left/Top/Right/Bottom	Specifies the sides to add pin text to. This is defaulted based on the current state of the symbol upon opening the form. It is recommended that text be placed on the top and left sides, or all four sides, depending on the numbering pattern.
Offset X/Y	Defines the offset of the text from the border of the symbol.
Text Size	Specifies the text size to use. Displays ** if multiple sizes are currently in use. Text sizes are listed as block index plus height x width for easy reference and picking of the right size.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Symbol Edit application mode - Place Driver: Options Panel

⚠ The option to place a driver in the Symbol Edit application is available only for co-design dies in Allegro X Advanced Package Designer (APD), and when drivers are set to visible.

Bubble	Specifies the bubble mode to use when placing drivers. This can be one of <i>Clockwise</i> , <i>Counter-Clockwise</i> , <i>Both</i> , or <i>None</i> . The bubble determines what drivers are pushed or shoved to prevent overlaps with the driver being modified when it is placed into its new position. Default is <i>None</i> .
Mirror	Specifies the mirror type to use when the driver is placed. Defaults to not-mirrored for unmirrored dies (for chip-up wirebond) or mirrored for chip-down dies such as a flip-chip, chip down die.
Rotation	Specifies the rotation to use when the driver is placed. Defaults to <i>Automatic</i> . In addition to North, South, East, and West, you can specify <i>Automatic</i> and <i>Maintain</i> . Automatic rotates to face the nearest die side and Maintain keeps the same rotation relative to the closest die side.
Size	Displays the size of the driver definition (or set of definitions) currently on the cursor. This is useful for hand-computing the placement for the reference pick.
Current LEF Library	Shows the currently active LEF library. You can choose to activate a different LEF library, which will update the available driver definitions tree.
LEF/DEF Library Manager	Opens the LEF Library Manager to update or reconfigure the LEF libraries and, as a result, driver definitions, prior to selecting those to be placed from the tree view.
Unplaced Drivers	Shows all the unplaced drivers for the active co-design die component in a tree view. Select the drivers to be placed. As each driver is clicked, it is added to the cursor to the immediate right of the last selected driver, to allow for multiple drivers to be placed at the same time.

Related Topics

- [symboloedit](#)
- [Symboloedit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Die Abstract Write Dialog Box

<i>Die to write abstract for</i>	Specify the co-design die name for which the die abstract is to be exported.
<i>Die abstract (.dia or .xda) file</i>	Select and specify the name and location for the die abstract file. selected by default.
Encounter I/O update (.txt) file	Specify the name and location for the Encounter I/O update file. selected by default.
Version	Specify the die abstract file version. Not selected by default. The default version is 4.0, which writes an unencrypted die abstract file. Note that the extension will be changed to .xda for 4.0 version.
Write	Click to export the files.
Close	Click to close the dialog box without exporting.
Help	Click for online help information.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Refresh Co-Design Die: Options Panel

<i>File</i>	Specify the file representing an updated view of the selected distributed co-design die or update the die from the active IOP session for concurrent co-design dies, which is the default. ⚠ The IC design name in the updated die abstract file for distributed co-design die must match the die being edited.
Synchronize IC and Package nets	Synchronizes IC and package nets. Selected by default.
<i>Ignore FIXED property</i>	Select to ignore the FIXED property. Not selected by default. ⚠ Refresh will fail for if FIXED property is assigned to any item that is being refreshed.
Log all changes	Select to log all changes.
IC Library Manager	Click to open the IC Library Manager dialog box.
Apply	Click to apply the changes.
Cancel	Click to cancel the command without making any changes.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Refresh Co-Design Die Finish Form

<i>Run to purge unused nets on exit</i>	Runs the purge unused net command to remove any unused nets from your design. Selected by default.
<i>Run derive assignment on exit</i>	Runs the derive assignment command to check your display for unconnected shapes and incomplete netlists and to automatically assign the connections from the existing conductor pattern. Selected by default.
<i>Push the new connectivity out to the rest of the design</i>	Propagates new signal names to the connected elements. Not selected by default.

Net for Component Pin Dialog Box

	Use the filter list box to display nets with a specific pattern. Type asterisk (*), the default value, to display all nets.
	Click to select a net from the displayed list.
Database	Select to list all nets from the current database.
Library	Select to list all nets from library.
DC Nets	Select to display only DC nets.
OK	Click to apply changes and close the dialog box
Cancel	Click to close the dialog box without making any changes.
Help	Click to get online help about the dialog box.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Driver Move: Options Panel

Reorient in place	Select to reorient the drivers at the current location without moving it. The selected drivers will no longer appear on the cursor. Not selected by default.
Stretch routing	Select to stretch routing clines and vias connected to the pads to the new driver destination. Selected by default.
Driver Rotation	Specifies the rotation to use when the driver is placed. Defaults to Automatic. In addition to <i>North</i> , <i>South</i> , <i>East</i> , and <i>West</i> , you can specify <i>Automatic</i> and <i>Keep Current</i> . Automatic rotates to face the nearest die side and Keep Current keeps the same rotation. In conjunction with <i>Reorient in place</i> , if a rotation of <i>Automatic</i> is selected, as the cursor is moved about the die, the placement of the drivers rotate by 90 degree increments depending on the side of the die the cursor is pointing to. A rotation of <i>Keep Current</i> will keep the current rotation no matter where you place it. The other options (North, South, East, West) control the specific rotation of the devices. Note that these options do not rotate the drivers as a group.
Drive Mirror	Specifies the mirror type to use when the driver is placed. Select from one of <i>Keep Current</i> , <i>Off</i> , or <i>On</i> . By default, <i>Keep Current</i> is selected, which keeps the current mirroring.
Cursor Reference	Specify the cursor in relation to the selected driver(s). You can select any one of <i>Pick_Point</i> , <i>Driver_Origin</i> , <i>Driver_Top_Left</i> , <i>Driver_Top_Right</i> , <i>Driver_Bottom_Left</i> , <i>Driver_Bottom_Right</i> , or <i>Pin_Origin</i> . <i>Driver_Origin</i> is selected by default. If <i>Pick_Point</i> is selected, you can choose <i>Snap Pick To</i> to precisely select the reference point. You may also just click a point in the driver as the reference point.
X offset	Specifies a cursor offset from the reference point in the X direction. This allows you to place the drivers precisely relative to some other object or point using <i>Snap Pick To</i> .
Y Offset	Specifies a cursor offset from the reference point in the Y direction. This allows you to place the drivers precisely relative to some other object or point using <i>Snap Pick To</i> .
Apply Reorient	Click to reorient selected drivers.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Align Driver: Options Panel

Stretch routing	Select to stretch routing clines and vias connected to die pads that are part of a driver to the new driver destination. Selected by default.
Align drivers as a group	Select to align the drivers as a group, maintaining the relative placement to each other. Not selected by default. Available only if you select more than one driver. When you align drivers as a group, the alignment happens relative to a reference driver.
Select reference point for alignment	Allows you to select the reference point on the selected drivers and then you select the destination for the alignment. If this option is not selected, alignment is done to the nearest edge of the selected drivers. Only the destination is selected to indicate where the drivers will align to. Not selected by default.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Respace Driver: Options Panel

Spacing/Overlap	<p>Specifies the space between respaced drivers.</p> <p>A value of 0 will have the drivers touching each other. A negative value will result in overlapping drivers, if the <i>symed_allow_overlapping_drivers</i> variable is set under <i>Ic_packaging – Symbol_editor</i> in User Preferences Editor (<i>Setup – User Preferences</i>).</p>
-----------------	--

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Swap Driver: Options Panel

Rip up routing	Select to rip up routings. Not selected by default.
Stretch routing	Select to stretch routing clines and vias connected to the pads to the new driver destination. Selected by default.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Accessing Command Help

To access command help for pop-menu options within an application mode:

1. Type `helpcmd` in the console window.
The Command Browser dialog box appears.
2. Select *Help* 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.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)

Symbol Edit Application Mode Tasks for Various Object Selection

The following table lists the elements selected in the Find Filter and the corresponding tasks that you can perform in the Symbol Edit application mode.

When You Select This Element in the Find Filter...	You Can Perform this Task...
No selection	<ul style="list-style-type: none">• Adding a Fully-Customized Component from Scratch
<i>Comps Symbols</i>	<ul style="list-style-type: none">• Adding a Pin• Adding a Grid• Specifying Pin Pitch Settings• Specifying Pin Numbering Settings• Changing Bump/Ball Attributes• Editing Die Properties• Specifying Pin Text Settings• Editing Boundary• Adding a Keepin or Keepout• Comparing components• Renaming a Component or Symbol• Writing Symbol Spreadsheets• Writing Device File to Disk• Copying a Component• Converting Vias to Pins• Specifying CTE Compensation for Components• Writing Symbol Spreadsheets• Writing Device File to Disk <p>The following tasks are available for co-design dies in Allegro X Advanced Package Designer (APD):</p> <ul style="list-style-type: none">• Editing Die Properties• Comparing components• Writing a die abstract file• Refreshing a Distributed Co-design Die• Renaming a Component or Symbol• Viewing and Editing IC Details

<i>Pins</i>	<ul style="list-style-type: none">• Deleting Pins• Moving Pins• Copying Pins• Swapping Pins• Changing Pin Attributes• Changing Bump/Ball Attributes• Writing Symbol Spreadsheets• Aligning Pins• Respacing Pins• Converting Pins to Vias
<i>I/O Drivers</i>	<ul style="list-style-type: none">• Changing Bump/Ball Attributes• Moving I/O drivers in a co-design die• Aligning I/O drivers in a co-design die• Respacing I/O drivers in a co-design die• Swapping I/O drivers in a co-design die• Changing I/O driver placement status

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)

Adding a Fully-Customized Component from Scratch

-  Ensure that the place bound or assembly layer visibility is on to be able see the outline of the component to confirm it and to be able to select the symbol to perform additional operations once it is defined.

You can add a fully-customized component such as a die, interposer, BGA, plating bar, or discrete by following this procedure:

1. In the Symbol Edit application mode, ensure no objects are selected.
2. Choose *Add component* from the pop-up menu.
3. Configure the controls in the *Options* panel.
4. Click *Create Component*.

The component is created and shown in the canvas.

-  A temporary pin is created on the component. This pin is automatically removed when you start adding actual pins to the component.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Adding a Pin

You can add pins to a component by defining the pin configuration and a pattern, if needed. You can add the pin pattern to a group to be able to select and perform operations on a group.

⚠ If pins and die are placed on same layer and the pin is not on the same layer as defined in padstack definition, a reference is created for the pin to avoid the need for an alternate padstack.

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Select the component to add pin to and choose *Add pin* from the pop-up menu.

⚠ If there are multiple instances of the same symbol or component definition, the new pins will be added to all the instances.

3. Configure the controls in the *Options* panel.
Specify pin configuration and pin pattern.
4. Click to place the pins. You can choose to rotate, mirror, or snap pick to from the pop-up menu while placing the pins.
The pins snap to the symbol's defined grid positions.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Adding a Grid

Pin numbers are assigned based on the pin pitch of the grid used for the numbering. This will be the grid the pin is assigned to, if that grid has a numbering pattern, or the symbol's base grid, if the pattern in the direct parent grid is set to "Inherit from base grid".

The grid is similar to a piece of grid paper with lines the pin pitch apart. The intersections are where the pins are supposed to be. If you are inheriting from the base grid, however, the pin may not lie directly on one these positions. In that case, the closest point to the pin's location is used.

If your grid has E5 at (500, 500) and E6 at (600, 500), but the pin is at (525, 500), it will be assigned pin number E5 as the closest point. You cannot have a pin number "E5.25" because it is a quarter of the way past the exact position for that numbered position in the pattern.

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Select the component to add grid to and choose *Add grid* from the pop-up menu.
3. Configure the controls in the *Options* panel.
4. Click to specify the starting of the grid and then click again to specify the rectangle for the grid outline.

Related Topics

- [symboledit](#)
- [SymbolEdit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Specifying Pin Pitch Settings

To specify pin pitch settings, perform these steps:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Select a grid.
3. Choose *Pin pitch settings* from the pop-up menu.
4. Configure the controls in the *Options* panel.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Specifying Pin Numbering Settings

Follow these steps to specify pin numbering settings:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Select a grid.
3. Choose *Pin numbering settings* from the pop-up menu.
4. Configure the controls in the *Options* panel.

After changing pin numbering settings, write a new device file to the disk and backannotate with the logical tool.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Viewing and Editing IC Details

You can view and edit IC details by following these steps:

1. In the Symbol Edit application mode, make sure Comps or Symbols is selected in the *Find* panel.
2. Select the die component and choose Show IC Details from the pop-up menu.

 This is a toggle option and you can choose Hide IC Details to hide the displayed details. If you select drivers displayed using Show IC Details, Hide IC Details is available.

IC details such as drivers and net information will be displayed and available for editing.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Changing Bump/Ball Attributes

To change bump/ball attributes, follow these steps:

1. In the Symbol Edit application mode, make sure *Pins*, *Symbols*, or *Comps* is selected in the *Find* panel.

 If you set *Comps* or *Symbols*, the values specified will be default for all non-customized pins on the selected object. You can set *Pins* to specify changes for selected pins only.

2. Select components.
3. Choose *Bump/Ball attributes* from the pop-up menu.
4. Configure the controls in the *Options* panel.
This information is used in the 3D Viewer to determine the unique radius when drawing a bump or a ball.
5. Click *Apply Changes*.
The bump geometry specified overrides the default.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Editing Die Properties

To edit die properties, perform the following steps:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Choose *Die properties* from the pop-up menu.
3. Configure the options pane.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Specifying Pin Text Settings

To specify pin text settings:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Choose *Pin text settings* from the pop-up menu.
3. Configure the controls in the *Options* panel.
4. Click *Apply changes*.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Element Selection in Find Filter and Corresponding Symbol Edit Tasks](#)

Editing Boundary

Perform these steps to edit a boundary in your design:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Choose *Edit boundary* from the pop-up menu.
3. Configure the controls in the *Options* panel.
4. Click *Update symbol extents* if you made changes to the X/Y coordinates or click *Select symbol outline* to select a new shape.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Adding a Keepin or Keepout

To add a keepin or keepout, follow this procedure:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Choose *Add Keepin/Keepout* from the pop-up menu.
3. Configure the controls in the *Options* panel.
4. Click *Add to symbol*.
The keepout is added to the symbol.
5. Similarly, add other keepouts and click *Finish adding outlines* when your are done.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Comparing components

You can compare components in your design by following these steps:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Choose *Compare component* from the pop-up menu to open the Component Compare dialog box.

Related Topics

- [compare comp](#)
- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Writing a die abstract file

This is only available for co-design dies in APD. Follow these steps to write a die abstract file:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Choose *Write die abstract* from the pop-up menu to open the Die Abstract Write dialog box.
3. Click *Write* in the dialog box after configuring it.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Swapping Pins

Perform the following steps to swap pins:

1. Choose *Place – Swap – Pins* (the layout editor) or *Place – Swap Pins* (Allegro Package SIL (`swap_pins` command)).

2. Choose the first pin to exchange.

The chosen pin, ratsnest lines, and all pins available to swap become highlighted.

To determine if a pin is swappable although it is not highlighted, click to choose the pin. The command window prompt displays an explanation of why the pin is not swappable.

1. From the highlighted pins, choose the second pin to exchange.

All circuit elements unhighlight, except the two pins that are swappable and the ratsnest lines.

2. To complete the swap and remain in swap mode, choose the first pin of the next swap.

3. When you finish swapping, right-click to display the pop-up menu and choose *Done*.

Related Topics

- [swap pins](#)
- [Swap Pins Command: Options Panel](#)
- [Swapping Pins of a Single Differential Pair Net](#)

Copying a Component

To copy a component, follow these steps:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Choose *Copy component* from the pop-up menu.
3. Specify a package name and click *OK*.
4. Specify a reference designator and click *OK*.
The new component is attached to the cursor.
5. Click to place a copy of the component.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Refreshing a Distributed Co-design Die

to refresh a distributed co-design die, follow this procedure:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.

2. Choose *Refresh co-design die* from the pop-up menu.

3. In the Options pane, browse for a file representing an updated view of the selected distributed co-design die.

This will refresh the die in the database from the disk file. Any updates made in the packaging tool that have not been exported will be lost.

4. Configure the other controls in the *Options* panel.

You can make library setting changes from this command.

5. Click *Apply*.

The Refresh Co-design Die Finish Form window appears.

6. Make changes and click *Finish*.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Symbol Edit Application Mode Tasks for Various Object Selection](#)

Writing Device File to Disk

To write device file to disk, follow these steps:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Choose *Write device file* from the pop-up menu.
The device file is written to the current working directory.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Element Selection in Find Filter and Corresponding Symbol Edit Tasks](#)

Writing Symbol Spreadsheets

Perform the following steps to write symbol spreadsheets:

1. In the Symbol Edit application mode, make sure *Pins* or *Comps* is selected in the *Find* panel.
 2. Select a component or select a subset or group of pins.
-  If you select a set of pins, only the selected pins will be exported.
3. Choose *Write symbol spreadsheet* from the pop-up menu.
 4. Configure the Symbol to Spreadsheet dialog box.
 5. Click *OK*.

Related Topics

- [symbol to spreadsheet](#)
- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)
- [Element Selection in Find Filter and Corresponding Symbol Edit Tasks](#)

Renaming a Component or Symbol

Follow these steps to rename a component or symbol:

1. In the Symbol Edit application mode, make sure *Comps* or *Symbols* is selected in the *Find* panel.
2. Choose *Rename component* or *Rename symbol* from the pop-up menu.
3. Specify the new name when prompted and click *OK*.

 If you select more than one symbols or components, separate prompts appear for each selected component or symbol.

Related Topics

- [symboledit](#)
- [Symboledit Dialog Boxes and Options Panel](#)
- [Accessing Command Help](#)

Converting Vias to Pins

1. In the Symbol Edit application mode, make sure *Comps* is selected in the Find window pane.
2. Select the component for which you want to convert vias to pins.
3. From the pop-up menu choose *Convert vias to pin* and then choose one of the two options, *Delete converted vias* to delete converted vias or *Keep converted vias* to retain the converted vias.
4. Select the vias you want to convert.
The created pins use the padstack, location, rotation, and net name of the vias.

Default pin number is based on auto numbering pattern; if custom numbering pattern is active, pin numbers will be the next available, unique number starting from 1.

Specifying CTE Compensation for Components

To specify coefficient of thermal expansion (CTE) compensation for components:

1. In the Symbol Edit application mode, make sure *Comps* is selected in the Find window pane.
2. Select the component for which you want to set convert vias to pins.
3. From the pop-up menu choose *CTE compensation*.
The Options panel shows the parameters that you can set for CTE compensation.
4. Set *Ignore fixed property* to move the vias with pin.
5. Set *Stretch routing* to stretch existing etches with pin move.
6. Specify the CTE expansion value for X and Y axis.
The value is applied to the pin location of the symbol.
A positive value will expand the symbol and a negative value will contract the symbol.
7. Set *Create new symbol definition* to create an alternative symbol with CTE compensation value.
This option is not selected by default. If not selected, the CTE compensation is applied to the symbol and no alternative is created.
The alternative symbol is named <*symbol_name*>_CTE.
8. Set *Create ghost pins* to add a ghost image of the original original pin positions to the symbol.
This options is set by default.
9. Click *Apply*.
CTE compensation is applied to the symbol.
If the component is now imported using one of the import commands, such as *die text in* or *def in*, the compensation will be applied to the imported component. To stop importing of CTE compensation for components, set *icp_disable_cte_auto_update* under *lc_packaging* in User Preferences Editor.

Deleting Pins

To delete pins when you are in the Symbol Edit mode, follow these steps:

1. In the Symbol Edit application mode, make sure *Pins* is selected in the Find pane.
2. Select the pins to be deleted.
3. Right-click and do one of the following to delete the pin:
 - To delete only pins, choose *Do not rip up connections*.
 - To delete the pins and their connected routing, choose *Rip up connected routing*.

If the pins are part of an auto-numbered grid, such as the clockwise spiral pattern, all pin numbers in the same grid are updated to account for the removed pins.

For pins of a co-design die component, the associated driver cell are unplaced when its pin(s) are deleted.

Moving Pins

1. In the Symbol Edit application mode, make sure *Pins* is selected in the Find window pane.
2. Select the pins to be moved.
The selected pins must be from the same component.
3. Choose *Move* from the pop-up menu.
If you selected more than one pin, click on any of the pins to pick a reference point.
4. Configure the controls in the Options window pane.
5. Click to place the pins. You can choose *Rotate* from the pop-up menu when placing the pins.

Copying Pins

1. In the Symbol Edit application mode, make sure *Pins* is selected in the Find window pane.
2. Select the pins to be copied.
The selected pins must be from the same component.
3. Choose *Copy* from the pop-up menu.
If you selected more than one pin, click on any of the pins to pick a reference point.
4. Configure the controls in the Options window pane.
5. Click to place the pins. You can choose *Rotate* from the pop-up menu when placing the pins.
6. Specify pin numbers and names if the destination is not an auto-numbered grid.

Changing Pin Attributes

1. In the Symbol Edit application mode, make sure *Pins* is selected in the Find window pane.
2. Select pins.
3. Choose *Change attributes* from the pop-up menu.
4. Configure the controls in the Options window pane.
5. Click *Apply Changes*.

Related Topics

- [Options Window Pane for the Symbol Edit application mode – Pin Move/Copy/Modify](#).

Aligning Pins

1. In the Symbol Edit application mode, make sure *Pins* is selected in the Find pane.
2. Select the pins you want to align.
3. Choose *Align* from the pop-up menu.
4. Choose any one of the submenu options to align the pins: *Top*, *Center Vertical*, *Bottom*, *Left*, *Center Horizontal*, or *Right*.
The pins will be aligned according to the option chosen. For example, *Left* will align the pins into a line with the *X* coordinate value of all pins being equal to the smallest *X* coordinate value of the selected pins.
For *Center Vertical* and *Center Horizontal*, the midpoint between the two extreme pin positions is calculated to determine the new position.
The final pin placement will be snapped to the pin placement grid, if defined. Note that the tool will ensure that pins do not overlap.

Respacing Pins

1. In the Symbol Edit application mode, make sure *Pins* is selected in the Find window pane.

2. Select the pins to respace.

You can select pins that are part of groups or different groups, as well as pins that are not part of any group.

3. Choose *Respace* from the pop-up menu.

The Options pane will show the pin pitch of the selected pins.

The displayed pitch is the greatest common divisor of pitches between all the selected pins, and respacing will maintain the same relative number of these pitches between each of the pins.

The *Vertical* and *Diagonal* fields will be disabled to indicate changing these values would not change the relative pin placements if all the selected pins are in a horizontal line, that is, have the same Y coordinate value. Similarly, the *Horizontal* and *Diagonal* fields will be disabled if all selected pins are in a vertical line with a common X coordinate value.

4. Click to set the reference.

Respacing will be performed relative to the reference point. For example, if you select a pin, the respacing will be performed relative to that pin. You can also select the center of a symbol as the reference point; for instance, to avoid pins moving out beyond the symbol extents.

The pitch is calculated based on the selected pins – only respacing is performed relative to the selected reference point. Also, note that you can select a pin as a reference if it is part of the set of pin selected for respacing.

5. Change the pin pitch to match your requirements.

The change in pitch will be reflected on the design canvas if a reference pin is specified.

The final pin placement will be snapped to the pin placement grid, if defined.

6. Click *Apply Changes* in the Options pane to commit the pitch change.

Converting Pins to Vias

1. In the Symbol Edit application mode, make sure *Pins* is selected in the Find window pane.

2. Select the pins to convert to vias.

3. From the pop-up menu choose *Convert pins to vias* and then choose one of the two options, *Delete converted pins* to delete converted pins or *Keep converted pins* to retain the converted pins.

Vias are created matching the source pin's placement, padstack, and net assignments. Properties are not copied to the new vias.

Moving I/O drivers in a co-design die

1. Ensure I/O drivers are visible and editable.

2. In the Symbol Edit application mode, make sure *Symbols* is selected in the Find window pane.

3. Select the drivers to be moved.

4. Choose *Move* from the pop-up menu.

5. Click to place the driver. You can rotate or mirror the drivers when placing them.

The drivers snap to the manufacturing grid when they are placed.

Related Topics

- [Viewing and Editing IC Details](#)

Aligning I/O drivers in a co-design die

1. Ensure I/O drivers are visible and editable.
2. In the Symbol Edit application mode, make sure *Symbols* is selected in the Find window pane.
3. Select the drivers to be aligned.
4. Choose *Align* from the pop-up menu.

5. Specify the options.

If you select the *Align drivers as a group* option, the drivers are aligned such that their relative placement to each other is maintained. If this option is not selected, the drivers are moved to the specified reference point.

It is important to select *Align drivers as a group*, to ensure drivers do not overlap. For example, if the drivers shown in the image are respaced laterally, they might overlap if *Align drivers as a group* is not selected.

If you set *Select reference point for alignment*, select the reference point.

6. Select the object to align to.

The nearest edge of the drivers will be aligned to the selected reference object. The drivers will, however, adjust to remain on the manufacturing grid.

Related Topics

- [Viewing and Editing IC Details](#)

Respacing I/O drivers in a co-design die

1. Ensure I/O drivers are visible and editable.
2. In the Symbol Edit application mode, make sure *Symbols* is selected in the Find window pane.
3. Select the drivers to be respaced.
4. Choose *Respace* from the pop-up menu.

5. Specify the *Spacing/Overlap* value.

A value of `0` will have the drivers touching each other. A negative value will result in overlapping drivers, if the `symed_allow_overlapping_drivers` variable is set under `lc_packaging - Symbol_editor` in User Preferences Editor (*Setup - User Preferences*). If the `symed_allow_overlapping_drivers` variable is not set, negative values will have the same result as `0`.

Related Topics

- [Viewing and Editing IC Details](#)

Swapping I/O drivers in a co-design die

1. Ensure I/O drivers are visible and editable.
2. In the Symbol Edit application mode, make sure *Symbols* is selected in the Find window pane.
3. Select one of the drivers.

You can also select the two drivers you want to swap.

4. Choose *Swap* from the pop-up menu.
5. If you selected only one driver in step 2, select the second driver to swap.

The two drivers will swap places so that the outer edge is at the same distance from the die edge as the original driver. The positions will snap to the manufacturing grid.

You might need to manually correct any overlaps resulting from swapping drivers of different sizes.

Related Topics

- [Viewing and Editing IC Details](#)

Changing I/O driver placement status

1. In the Symbol Edit application mode, make sure *Symbols* is selected in the Find window pane.
2. Select one or more drivers.
3. Choose *Change Driver Placement Status* from the pop-up menu.
4. Choose one of *Cover*, *Fixed*, or *Placed*.

Once you choose *Cover*, only *Hide IC Details* and *Change Driver Placement Status* options are available in the pop-up for that driver.

symbol to spreadsheet

The `symbol to spreadsheet` command lets you export information about a placed component to a standard spreadsheet tool such as Microsoft Excel. You can use this command to exchange information with your system architect, front-end tools, or as part of your manufacturing documentation set when signing off a design.

- ⚠ Any formula in the spreadsheet is preserved while importing information. Note that the formula syntax is not validated and formulas are not evaluated by Cadence tools. You must evaluate or edit the cell contents in your spreadsheet editing tool, say, Microsoft Excel.
- ❗ Use this command only for symbols with regular pin patterns. This command is not suggested for symbols with irregular pin pattern because the resulting spreadsheet will be large in size and the information might not be useful.

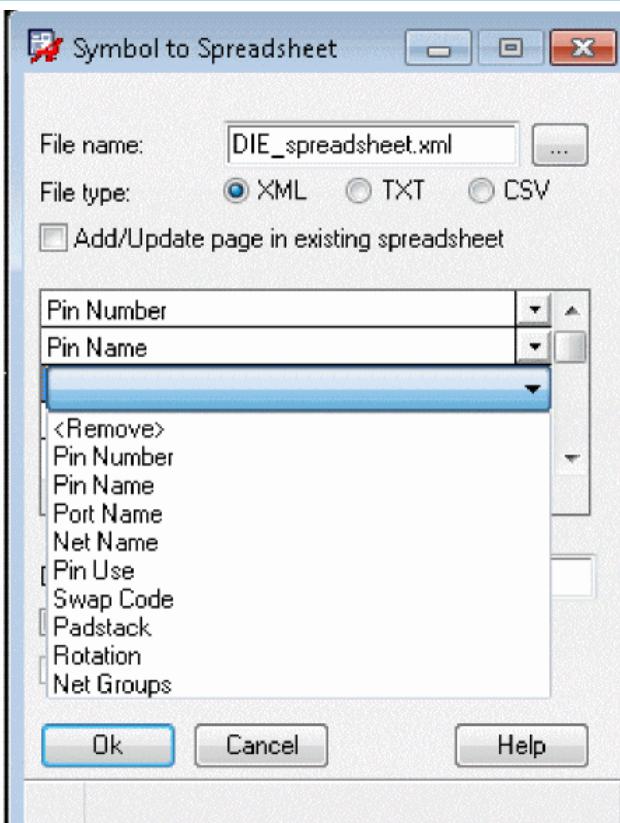
Related Topics

- [Exporting to a Spreadsheet](#)

Symbol to Spreadsheet Dialog Box

Access Using

- Menu Path: *File – Export – Symbol Spreadsheet*

File Name	Specifies the name of the file to be written. The default name is <refdes>_spreadsheet.<extension>.
...	Lets you browse to the directory and file name for the output file.
File type	Lets you select any one of the file types as output: TXT (text), CSV (comma separated value), or XML (open spreadsheet XML). The XML format retains the highlight color for pins or nets. The cell delimiter in TXT files is tab and in CSV files is comma. XML is selected by default.
Add/Update page in existing spreadsheet	Appends to or updates existing spreadsheet. If the specified spreadsheet has a worksheet with a matching name, it will be updated during export. The spreadsheet will be appended if there are no matching RefDes. Available for XML file type with support for multiple worksheets in the same file. Not selected by default.
Grid	 <p>Provides a list of available fields that can be written to the file. Initially two entries are shown, one with <i>Pin Number</i> selected and the other blank. When you select a second entry, a third blank entry is displayed and so on, allowing you to add all the available fields, if needed. The entries are <i>Pin Number</i>, <i>Pin Name</i>, <i>Port Name</i>, <i>Net Name</i>, <i>Pin Use</i>, <i>Swap Code</i>, <i>Padstack</i>, <i>Rotation</i>, and <i>Net Groups</i>. You can remove an entry by selecting <Remove>. You must select at least one entry to be able to export.</p>
Delimiter	Specifies the delimiting string that the tool uses between the different fields written to the file. The default setting is "\", but you can enter any character or string, for example, "...". This field is enabled only when you select to write more than one field. Also, this is only active for <i>TXT</i> or <i>CSV</i> files, not <i>XML</i> .

S Commands
S Commands--symbol to spreadsheet

Include data labels in cells	Includes keywords indicating what each line of data in a cell represents. Not selected by default. Following are the keywords: <ul style="list-style-type: none"> ◦ PINNUMBER ◦ PINNAME ◦ PORTNAME ◦ NET ◦ PINUSE ◦ SWAPCODE ◦ PADSTACK ◦ ROTATION ◦ NETGROUPS
Add row and column headers on the top/left sides	When enabled, this causes the first row and first column to list the associated numeric or alpha component of the pin number associated with the specified row or column. This option is checked by default, but it is only available for selected components that use an alphanumeric, grid-based pattern. If the tool does not detect an alphanumeric grid-based pattern in the component, the option is unavailable.
Rows and columns defined by	Select how you want to define rows and columns in the generated spreadsheet. They can be defined by <i>Component pin pitch</i> , by <i>Unique rows/columns</i> , or you can create a <i>Custom</i> definition by providing the required <i>X / Height</i> and <i>Y / Length</i> values for the rows and columns.
Pattern Rotation	Allows you to set a pattern rotation. You can choose one of the following from the drop-down menu: 0, 90, 180, or 270 degrees.
Pattern Mirror	Allows you to set pattern mirror. You can choose to have no mirroring, mirror it about the X-axis, or mirror it about the Y-axis.
OK	When you click this button, the tool writes the spreadsheet text file as configured, closes the dialog box, and waits for you to select the next component.
Cancel	When you click this button, the tool closes the dialog box without writing the file, and waits for you to select a new component.
Help	When you click this button, the tool displays context-sensitive help for this command.

Exporting to a Spreadsheet

To export component data to a spreadsheet:

1. Run the `symbol to spreadsheet` command.
2. Select the component to be exported.
The Symbol to Spreadsheet dialog box appears.
3. Specify the file name and directory where the file will be stored.
4. in the grid of lists, select the fields to be written to the file.
5. Click *OK* to generate the report.
6. Select another component and follow Steps 3 to 5
-or-
Right-click and choose *Done* to exit the command.

Related Topics

- [symbol to spreadsheet](#)

sna param

The `sna param` command displays the Signal Analysis Parameters dialog box. Though maintained for compatibility with older databases, it has been replaced by the `signal prefs` command, which displays the Analysis Preferences dialog box.

Signal Analysis Parameters Dialog Box

Use this dialog box to:

- Set default IBIS IOCell models, determine whether default IOCell models are used, and determine how buffer delays are obtained.
- Define preferences for routed and unrouted interconnect modeling and crosstalk checks and determine whether to do plane modeling.
- Set simulation defaults for pulse stimuli, simulation duration, waveform resolution, threshold measurement for delays, and debug mode. You can also define parameter Set default units of measure for reports.
- Set defaults for EMI single net simulations. You can also determine whether advanced EMI simulations are performed and set defaults for them.

S Commands
S Commands--sna param
