

B 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	8
B Commands	8
back	10
backdrill setup	11
Backdrill Pairs Question Dialog Box	12
Backdrill Setup and Analysis Dialog Box	13
Backdrill Layer Pair Initialization Dialog Box	18
Popup Menu Commands in Backdrill Setup and Analysis Dialog Box	19
Running Backdrill	21
backingstore	23
baf	24
batch	25
batch_drc	26
BATCH DRC Dialog Box	27
Making DRC Errors Visible	28
Running Batch DRC	29
Canceling DRC	31
bbvia	32
Using the -p <prefix> Switch	34
Using the -t Switch	34
Using the -c <cns> Switch	36
bend area create	36
Create Bend Area Dialog Box	38
Creating Bend Areas	40
bend area edit	41
Edit Bend Area Dialog Box	42

Editing Bend Areas	44
bga abstract export	45
	45
BGA Abstract Export Dialog Box	46
Exporting Interface and Assignment Information	47
bga abstract import	48
	48
BGA Abstract Import Dialog Box	49
Importing BGA Interfaces	50
bga editor	51
	51
Component Selection Dialog Box	52
Component Editing Dialog Box	54
New Padstack Information Dialog Box	62
Final Verification Dialog Box	63
Starting the BGA Editor	64
bga generator	65
	65
BGA Generator - General Information Dialog Box	66
BGA Generator - Pin Arrangement Dialog Box	67
BGA Generator - Pin Use Ratios Dialog Box	72
BGA Generator - Padstack Information Dialog Box	73
Padstack for Component Dialog Box	75
BGA Generator - Pin Numbering Dialog Box	77
BGA Generator - Preview Dialog Box	79
Creating a BGA	80
bga text in	82
	82
BGA Text In Dialog Box	83
Importing BGA Pin Data	89
bga text out	90
	90
BGA Text Out Dialog Box	91
Exporting BGA Pin Data	94
blank_waived_drcs	95
blank waived drcs	95
Concealing Waived DRC Error Markers in the Design	96

bmpflush	97
bmscheck	98
board keepout	99
Keepout Dialog Box	99
Opening the Keepout Dialog Box	100
Creating a Keepout	101
Editing a Keepout Area	102
Moving a Keepout	103
Deleting a Keepout	104
board outline	105
Design Outline Dialog Box	106
Opening the Design Outline Dialog Box	107
Creating a Board Outline	108
Editing a Board Outline	109
Creating a New Segment within an Existing Segment	110
Moving a Board Outline	111
Deleting a Board Outline	112
boardoutline import	113
Import Board File Dialog Box	114
Conflicts Dialog Box	115
Choosing a Source Board	117
Importing Board Geometry Information	118
Importing Electrical Rules	119
Importing Placed I/O Components	120
Importing Rooms	121
board plane	122
Plane Outline Dialog Box	123
bond wire length	124
bond wire location	126
bondwire text in	127
Bond Wire Text In Dialog Box	128
Importing Bond Wire Data	129
bpa	132
	133

BPA Command: Options Panel	134
Adding Text to BondPads	135
brd export	136
brd import	137
browse drcs	138
	138
Navigating in DRC Browser	145
Filtering in DRC Browser	147
DRC Browser Dialog Box	150
Waiving DRC by Group Select	151
build_pe_script	152
bundle blank	153
	153
Hiding Rat Bundles Associated with Selected Objects	154
Object Selection Shortcuts	155
bundle blank_all	156
Hiding All Rat Bundles	157
bundle blank_unselected	158
Hiding Unselected Rat Bundles	159
bundle create	160
Creating A New Bundle of Rats	161
bundle delete	162
	162
Deleting Selected Rat Bundles	163
Deleting All Rat Bundles in the Design	164
bundle edit	165
Adding or Remove Rats From a Bundle	166
bundle import	167
	167
Bundle Import Dialog Box	168
Importing Bundles From Another Design	169
Saving and Restoring Bundles of ECO-affected Designs	171
bundle properties	173
The bundle properties command displays a dialog box that lets you control the routing behavior, bundle characteristics (such as name), and initial visibility settings for one or more selected bundles. You can also create and manage ratsnest bundles in Constraint Manager.	173

Edit Bundle Property Dialog Box	174
Editing the Properties of a Single Bundle	180
Editing the Properties of Multi-selected Bundles	181
Removing Bundle Property Overrides	183
bundle restore	184
bundle show	185
	185
Displaying Rat Bundles Associated with Selected Objects	186
Displaying Selected Rat Bundles	187
bundle show_all	188
Displaying All Rat Bundles	189
bundle show_unplanned	190
Displaying Only Rat Bundles that are Not Planned	191
bundle split	192
	192
Splitting a Rat Bundle Into Two Bundles	193
Splitting a Rat bundle into Multiple Bundles	194
bundle toggle	195
	195
Toggling the Display of Rat Bundles Associated with Selected Objects	196
Toggling the Display of All Rat Bundles in the Design	197
button	198
bw doc	200

B Commands

back	backdrill setup	backingstore
baf	batch	batch_drc
bbvia	bend area create	bend area edit
bga abstract export	bga abstract import	bga editor
bga generator	bga text in	bga text out
blank_waived_drcs	bmpflush	bmscheck
board keepout	board outline	boardoutline import
board plane	bond wire length	bond wire location
bondwire text in	bpa	brd export
brd import	browse drcs	build_pe_script
bundle blank	bundle blank_all	bundle blank_unselected
bundle create	bundle delete	bundle edit
bundle import	bundle properties	bundle restore
bundle show	bundle show_all	bundle show_unplanned
bundle split	bundle toggle	button

B Commands
B Commands

bw doc		
--------	--	--

back

The `back` command sends a window from the front of the desktop to the background, behind other windows that may overlap it.

Syntax

`back`

backdrill setup

The `backdrill setup` command defines parameters for backdrilling, in which the unused portion of a pin or via plated thru hole is drilled out with a drill larger than the one originally used. This removes the plating in the unused stub portion of the hole, which might interfere with high-frequency signals on high-speed designs.

You can start backdrilling from any layer of the board. This capability is very helpful for board construction techniques used with some HDI and sub-laminate designs.

You can specify which types of hole (pins or vias) can be backdrilled, the etch layer of the board on which backdrilling may occur, and restrict the depths to which backdrilling occurs. Two system default passes let you quickly assess the result of backdrilling all pins and vias to the maximum depth permitted. All layer combinations are used.

Prior to committing to a backdrilling scheme, you can evaluate combinations of backdrill passes to assess their impact, using an analysis option. It provides visual cues for testpoint conflicts and stub and pin-length requirement problems. Backdrilling is not integrated into the DRC system. Backdrilling does not change antipads dynamically.

A dynamic backdrill mode is also available that maintains appropriate backdrill clearance and depth and updates backdrill information in real-time as updates are made in a design. Enabling dynamic backdrill mode is a two-step process. First, set a `backdrill_enable_dynamic` variable in User Preferences Editor under the *Manufacturing – Drilling* category. The second step is to select the *Enable Dynamic Backdrill* checkbox under the *Settings* tab of the Backdrill Setup and Analysis dialog box. When enabled, interactive editing commands update backdrill information on pins and vias in real-time without running multiple full-backdrill executions.

For more information on backdrilling, see the *Best Practices: Working with Backdrilling* guide in your documentation set.

Related Topics:

- [Backdrill Setup and Analysis Dialog Box](#)
- [Backdrill Layer Pair Initialization Dialog Box](#)
- [Popup Menu Commands in Backdrill Setup and Analysis Dialog Box](#)
- [Running Backdrill](#)

Backdrill Pairs Question Dialog Box

Access Using

- *Menu Path: Manufacture – NC – Backdrill Setup and Analysis*

<i>Deepest backdrill layer from top layer</i>	Initializes layer pairs from TOP side if the deepest layers are selected from both sides.
<i>Deepest backdrill layer from bottom layer</i>	Initializes layer pairs from BOTTOM side if the deepest layers are selected from both sides.
<i>OK</i>	Close to apply the layer pair initialization.
<i>Skip</i>	Skips layer payer initialization.

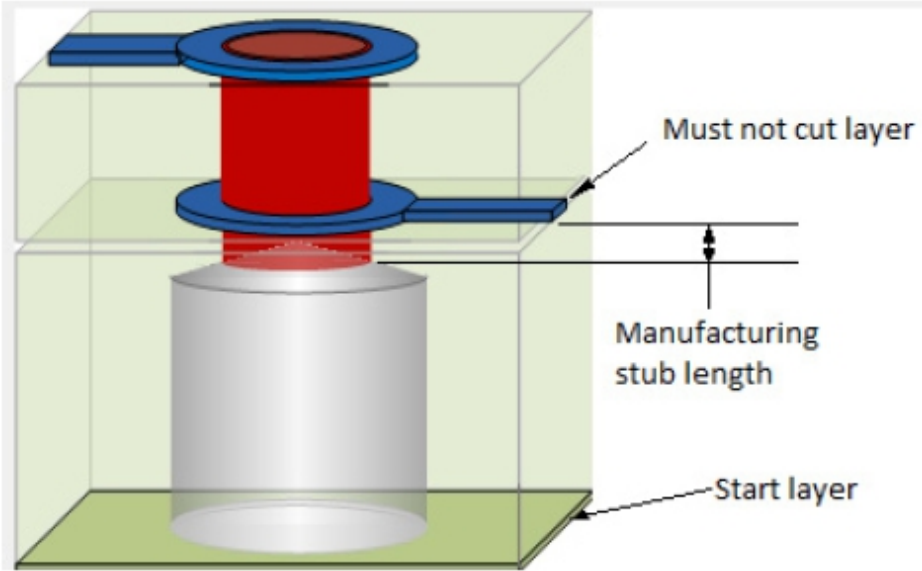
To open the Backdrill Setup and Analysis dialog box, you must have defined stackup information for copper and dielectric thickness on the Layout Cross Section dialog box, available by choosing *Setup – Cross-section* ([xsection](#) command).


Related Topics:


- [Backdrill Layer Pair Initialization Dialog Box](#)
- [Popup Menu Commands in Backdrill Setup and Analysis Dialog Box](#)
- [Running Backdrill](#)

Backdrill Setup and Analysis Dialog Box

<i>Layer Pairs Tab</i>		
	<i>Pair ID</i>	Set up backdrill layer pairs for analysis and backdrilling.
	<i>Enable</i>	Choose the etch layer and object definitions that guide backdrilling. Left click to toggle between enabling and disabling this column. You must choose values in the <i>From Layer</i> and <i>Objects</i> columns, or you cannot enable this layer pair, nor can the same objects and <i>From Layer</i> exist in more than one enabled layer pair.
	<i>Start Layer</i>	<p>Choose the etch layer to start backdrilling. This column displays all the board layers.</p> <ol style="list-style-type: none"> 1. Select <i>Top</i> and <i>Bottom</i> layers for simple backdrilling from the top or bottom sides of the board. 2. Select an internal etch layer for backdrilling starting from any layer of the board which is needed for HDI buried via and sub-laminate designs. <p>Read only for the two system default passes, and for all but the first pass-definition row of a pass set. Displays as blank and read only for subsequent passes of a pass set, but internally their value defaults from that of the first pass in the set.</p>
	<i>Objects</i>	Choose the type of holes to be considered as backdrilling candidates. Read only for the two system default passes, and for all but the first pass-definition row of a pass set. Displays as blank and read only for subsequent passes of a pass set, but internally their value defaults from that of the first pass in the set.
	<i>To Layer</i>	Choose a layer to which to backdrill from the drop-down list of board etch layer names. For pass set members, the etch layer names from the specified <i>From Layer</i> for the initial pass-definition row sequentially populate this column. If you choose no layer, backdrilling occurs to any layer using the specified <i>From Layer</i> .

	<i>Must Not Cut Layer</i>	<p>Specifies the layer that requires conductivity.</p> 
	<i>Depth</i>	<p>Read only value that indicates the depth to the required dielectric layer thickness beyond the etch specified in the <i>To Layer</i>, and defaults from the Layout Cross Section dialog box, available by choosing <i>Setup – Subclasses</i> (<code>define subclass</code> command). The two system passes display as blank, as backdrilling occurs on any layer (that is, any depth). A pass with no specified <i>From Layer</i> and <i>To Layer</i> displays as 0.0. Once you set the <i>From Layer</i> and <i>To Layer</i> columns, this column then automatically displays the depth to that layer from the side.</p>
	<i>Plunges</i>	<p>Specifies number of possible backdrill locations.</p>
<i>Layer Pair Initialization</i>	<p>Automatically generates backdrill Layer Pairs based on:</p> <ul style="list-style-type: none"> • Deepest Backdrill Layer from Top and Bottom Layers • Minimized electrical stub length or Minimize layer pairs <p>Click <i>Create</i> to update layer pairs.</p>	
Backdrilling Errors	<p>Displays count of backdrill errors</p>	

<i>Drill Parameters Tab</i>		
	Unit	Displays design units
	<i>Manufacturing stub length</i>	Specifies minimum tolerance for secondary drill operation.
<i>Padstack Parameters Tab</i>		
	Unit	Displays design units
	<i>Oversize backdrill diameter</i>	Specifies the value that is added to finished hole size of pins or vias in padstack definition and results in larger backdrill object.
	<i>Oversize antipad for negative layers</i>	Specifies the value that is added to backdrill hole size of pins or vias and results in larger antipad for all negative plane layers in the backdrill path.
	<i>Oversize keepout for backdrill layers</i>	<p>Specifies the value that is added to finished hole size of pins or vias and results in keepout on all signal layers in the backdrill path.</p> <div>  Oversize requests for backdrilled pads that are larger than the drill will be ignored while repairing shapes. </div>
	<i>Undersize regular pad for start layers</i>	Specifies the value that is subtracted from the backdrill hole size and results in smaller start or entry layer pad.
	<i>Oversize solder mask pad for start layers</i>	Specifies the value that is added to the backdrill hole size and results in soldermask pad start or entry layer.
	<i>Details</i>	Displays padstack report that lists of all padstacks in two lists: Padstacks with user-defined backdrill data and Padstacks without user-defined backdrill data
	<i>Export</i>	Click to save the padstack backdrill parameter file (.txt)

	<i>Import</i>	Click to import the padstack backdrill parameter file (.txt) in the current design
<i>Flag Codes Tab</i>	Specifies the backdrill flag codes displayed in design that identifies reasons backdrill generation errors.	
Settings tab		
	Suppress backdrilled pads	Enable to suppress pad on layers which are backdrilled
	Enable dynamic backdrill	Enable dynamic backdrilling <div style="border: 1px solid #fde725; padding: 5px; margin-top: 10px;">  This option is available when <i>backdrill_enable_dynamic</i> variable is set in the User Preferences Editor </div>
	<i>Disable backdrill error flag automatically</i>	Displays error flags in the design
<i>Disable dynamic shape update and DRC during backdrilling for better performance</i>	Disables dynamic shape updates during backdrill generation.	
No backdrill data on pins/vias	Enable to suppress backdrill generation on pins or vias	
<i>Backdrill Status</i>	The backdrill status is displayed as: <i>Red</i> : Backdrill data are out of date. Click <i>Backdrill</i> to update the status. <i>Green</i> : Backdrill data on pins/vias are in sync. <i>Grey</i> : Backdrill data is not generated on pins/vias.	
<i>Dynamic Backdrill State</i>	The backdrill status is displayed as: <i>Red</i> : Dynamic backdrill is not active <i>Green</i> : Dynamic backdrill active	

<i>OK</i>	Saves the settings in the design and closes the dialog box with no backdrill data generation.
<i>Cancel</i>	Closes the dialog box and discards the current backdrill pass definitions, reverting to those present when you originally opened the dialog box.
<i>Analyze</i>	Analyze design for each hole for backdrilling, but do not add backdrill data on pins/vias. This option generates a log file <code>backdrill_analysis.log</code> for review.
<i>View Log</i>	Displays the last <code>backdrill_analysis.log</code> file report.
<i>Backdrill</i>	Process each hole and save the backdrill data (side, start layer and must-cut-layer) on pin/via for display, manufacturing, reporting, NC drill legend, cross section chart, and so on.
<i>Purge</i>	Remove backdrill data (side, start layer and must-cut-layer) from pin/via.

Related Topics:

- [backdrill setup](#)
- [Popup Menu Commands in Backdrill Setup and Analysis Dialog Box](#)
- [Running Backdrill](#)

Backdrill Layer Pair Initialization Dialog Box

<i>Initialization</i>	
<i>Deepest backdrill layer from top layer</i>	Initializes layer pairs from TOP side if the deepest layers are selected from both sides.
<i>Deepest backdrill layer from bottom layer</i>	Initializes layer pairs from BOTTOM side if the deepest layers are selected from both sides.
<i>Create</i>	Updates Layer Pairs.
<i>Analysis</i>	
<i>Create layer pairs in the design based on backdrill analysis:</i>	
<i>Minimize electrical stub length</i>	Analyze all pins/vias in the design and list all the possible layer pairs with minimum stub length.
<i>Minimize layer pairs</i>	Analyze all pins/vias in the design and try to merge the adjacent layer pairs to minimize the total number of layer pairs.
<i>Create</i>	Updates Layer Pairs.

Related Topics:

- [backdrill setup](#)
- [Backdrill Pairs Question Dialog Box](#)
- [Running Backdrill](#)

Popup Menu Commands in Backdrill Setup and Analysis Dialog Box

Right-click in any column to display a popup menu from which you can choose one of the following:

<i>Insert Pair</i>	Appends another row after the pass set's last row when you choose the last row of a pass set.
<i>New Pair Set</i>	Adds a new pass-definition row as the start of a new pass set. However, the row is not necessarily always added as the last row. Given the row that is selected with the RMB click, the row that is added is after the last row of the pass set to which the selected row belongs. Inserting a pass-definition row after the first one of a pass set or between its intermediate passes makes it a member of the pass set. You cannot insert a new pass between the two system default passes (1 and 2). Its <i>From Layer</i> and <i>Objects</i> columns display as blank and read only, but internally their value defaults from that of the first pass. The <i>Passes</i> column for the pass set then rearranges the sequence. You cannot insert a pass into a pass set if doing so exceeds the board's total number of etch layers. Allegro X PCB Editor automatically limits a pass set to one fewer than the board's total number of etch layers.
<i>Enable Pair Set</i>	Activates all passes in the pass set to which the row belongs. You cannot enable the pass set if the objects and from layer it contains also exist in another enabled pass set.
<i>Disable Pair Set</i>	Deactivates all passes in the pass set to which the current row belongs.
<i>Delete Pair</i>	Removes the pass-definition row to which the current cell belongs. If the pass belongs to a pass set, the remaining passes in the <i>Passes</i> column renumber.
<i>Delete Pair Set</i>	Removes the pass set to which the current pass-definition row belongs.
<i>Delete All</i>	Removes all but the two system default passes.
<i>Enable All</i>	Activates all passes.
<i>Disable All</i>	Deactivates all passes.


Related Topics:

- [backdrill setup](#)
- [Backdrill Pairs Question Dialog Box](#)
- [Backdrill Setup and Analysis Dialog Box](#)

Running Backdrill

To run the backdrill command, perform the following steps:

1. Assign the BACKDRILL_MAX_PTH_STUB property to nets targeted for backdrilling. Cadence recommends using the General Properties worksheet in Constraint Manager to assign this property. Or you can use *Edit – Property* (`property edit` command).
2. Assign the BACKDRILL_EXCLUDE property to symbols, pins, or vias to exclude them from backdrilling.
3. Assign the BACKDRILL_MIN_PIN_PTH property to symbols or pins to ensure the backdrill depth does not violate minimum plating rules you specify. You can then control how much vertical depth is required for a pin to be properly plated.

 To graphically display any of these properties for ease of identification, use the Show Property dialog box's Graphics tab, available by choosing *Display – Property* (show property command).

4. Choose the etch layer to start backdrilling in the *From Layer* column.
5. Choose the type of holes to be considered as backdrilling candidates in the *Objects* column.
6. Choose the *Must Not Cut Layer* that define the layer that requires conductivity.
7. Verify in the *Depth* field the depth to the required dielectric layer beyond the etch specified in the *To Layer*.
8. Specify *Plunges* for each Pair ID.
9. Specify *Manufacturing stub length* in *Drill Parameters* tab.
10. Specify *Padstack Parameters*.
11. Click *Analyze* to execute a preview evaluation of the backdrilling impact of enabled pass definitions and automatically display the `backdrill_analysis.log` on screen. Use the log file to review the number of pins or vias backdrilled from respective sides of the board, excluded objects, stub violations unresolved by altering the backdrill pass definitions, and BACKDRILL_MIN_PTH_PIN violations.
12. To create a backdrill legend, enable the *Include Backdrill* option on the Drill Legend dialog box, available by choosing *Manufacture – NC – Drill Legend* (ncdrill legend command). To generate a drill output file for backdrilling, choose *Manufacture – NC – NC Drill* (nctape_full command) and enable the *Include Backdrill* option on the NC Drill Dialog Box.
13. Click *Backdrill* to generate backdrill data on pins/vias in the design.

14. Click *Purge* to removed backdrill data on pins and vias.
15. Click *View Log* to view display the `backdrill_analysis.log`.
16. Click *OK* to close the dialog box.

Related Topics:

- [backdrill setup](#)
- [Backdrill Pairs Question Dialog Box](#)
- [Backdrill Setup and Analysis Dialog Box](#)
- [Backdrill Layer Pair Initialization Dialog Box](#)

backingstore

The `backingstore` command stores a screen image in memory so repainting is unnecessary when a form is closed or a window moved. This feature saves repainting time, but requires additional memory for saving the screen data. This is an X window only feature and applies only to drawing data, not to objects such as forms or to the status window.

Syntax

```
backingstore {on | ifmapped | off}
```

on	Stores an image of the screen in memory. The feature can also be included in your global environment file by adding this line: <code>set display_backingstore = on</code>
ifmapped	Stores an image of the screen in memory, but deletes it if you minimize the application. After maximizing the application, you have to run the command again.
off	Prevents an image of the screen from being stored in memory for a particular drawing if you had previously enabled <code>backingstore</code> as part of your global environment file. The feature can also be included in your global environment file by adding this line: <code>set display_backingstore = off</code>

baf

The `baf` batch command back-annotates logic changes on a layout to the original schematic. This command runs only on UNIX platforms.

The `baf` command compares the current layout to the version created by the `netin` command. To perform this comparison, you must either have a copy of the layout created with `netin` or else process the netlist again, to create one.

The editor generates two output files. The first file is named `assignedname.baf`, which provides the results of the comparison of the two drawings. The second file, `backan.log`, provides information from the backannotation process.

Syntax

```
baf [-s|-o|-r] <unassigned>.brd <assigned>.brd
```

-s	Spares
-o	Other
-r	Reuse log
<unassigned>.brd	The name of the original layout created by the <code>netin</code> command
<assigned>.brd	The name of the current board

batch

The `batch` command runs batch commands provided with the tool from the console against the current database. For commands that act on a design, the results can be loaded back into the tool automatically.

To cancel a command, click the *Stop* button in the status window.

Syntax

```
batch [-b | -n] "<command_name> [<options>]"
```

-b	Runs the command in the background. This option does not update the design when the program is done. This option is useful for running commands whose output is not a new or updated design but another file, such as a report.
-n	Prevents the command from updating the design when the program is done.
<command_name>	The command you are running, for example the <code>report</code> command.
<options>	The options required by the command you are running. If the name of the input board is required, you can use <code>%s</code> in place of the board's name. You can place only one instance of <code>%s</code> in a command.

Example

In this example, you are running the Summary Drawing Report against the open board (`active.brd`). This is the batch command you run from the console:

```
batch "report -j %s output.rpt"
```

That command invokes the `report` command in this way:

```
report -j active.brd output.rpt
```

batch_drc

The `batch_drc` batch command runs design rule checking (DRC) for constraints. This allows you to view and resolve DRC violations on the whole design at once.

For more details about design rule checking, see *Creating Design Rules in the user guide*.

Syntax

```
batch_drc [-nographic <input_filename> [<output_filename>]] | [<input_filename> [<output_filename>]]
```

-nographic	Runs the command without displaying the BATCH DRC dialog box.
<input_filename>	Specifies the name of the design you are running DRC against. If you do not use the <code>-nographic</code> switch, you do not have to specify this file name in the command but you do have to enter it in the BATCH DRC dialog box.
<output_filename>	(Optional) Specifies the name of a new file in which you are saving the input design. If you do not specify an output file name, the tool uses the input file name, adding DRC information to the input design.

Related Topics:

- [Making DRC Errors Visible](#)
- [Running Batch DRC](#)
- [Canceling DRC](#)

BATCH DRC Dialog Box

Use this dialog box to enter the names of the design you are running DRC against and, if necessary, a different name for the file that contains both the design and the DRC information.

<i>Input Design</i>	Enter the name of the design you are running DRC against.
<i>Output Design</i>	(Optional) Enter the name of the file in which you are storing the input design and the added DRC information. By default, the tool enters the <i>Input Design</i> name here.

Related Topics:

- [Running Batch DRC](#)
- [Canceling DRC](#)

Making DRC Errors Visible

Before running design rule checking, make sure that any DRC violations are visible.

1. Run the `color192` command.

The Color dialog box appears.

1. Choose *Stack-Up*.
2. Check that the *DRC* box is selected for *All* (all layers).
3. Click *OK*.

Examples

- This first example uses the BATCH DRC dialog box, where you enter the information for running DRC:

```
batch_drc
```

- This example prefills the BATCH DRC dialog box with the input design name (and by default, the output design name, too). You can store the output of the command in a different file by entering a different output design name:

```
batch_drc design243.brd
```

- This last example runs the command without any graphical user interface. It supplies the command's results in an output separate from the original design:

```
batch_drc -nographic design243.brd design243a.brd
```

Related Topics:

- [batch_drc](#)
- [Canceling DRC](#)

Running Batch DRC

Follow these steps to run a Batch DRC:

1. Run the [status](#) command.
2. In the DRC Controls, deselect *On-Line DRC*.
3. Click *OK*.
4. Run the `batch_drc` command from an operating system prompt or (in Windows) from a Run command line.
 - To run the command without the graphical user interface, see Syntax. Do not continue with the rest of these steps.
 - To run the command with the graphical user interface, do not use the `-nographic` switch. Continue with step 5.
5. Complete the BATCH DRC dialog box, described above.
6. Click *Run*.
7. When the program is completed, if the `batch_drc.log` file does not appear, run the [viewlog](#) command to view a summary of the results.

Examples

- This first example uses the BATCH DRC dialog box, where you enter the information for running DRC:

```
batch_drc
```

- This example prefills the BATCH DRC dialog box with the input design name (and by default, the output design name, too). You can store the output of the command in a different file by entering a different output design name:

```
batch_drc design243.brd
```

- This last example runs the command without any graphical user interface. It supplies the command's results in an output separate from the original design:

```
batch_drc -nographic design243.brd design243a.brd
```

Related Topics:

- [batch_drc](#)
- [BATCH DRC Dialog Box](#)

Canceling DRC

To cancel a batch DRC, follow these steps:

1. To stop batch DRC, press `Control-C`.

As a result of cancelling, a red color box and *OUT OF DATE* appears next to *DRC errors*, indicating that DRC is out of date or Batch DRC is required. Use the [status](#) command to launch the dialog box.

Examples

- This first example uses the BATCH DRC dialog box, where you enter the information for running DRC:

```
batch_drc
```

- This example prefills the BATCH DRC dialog box with the input design name (and by default, the output design name, too). You can store the output of the command in a different file by entering a different output design name:

```
batch_drc design243.brd
```

- This last example runs the command without any graphical user interface. It supplies the command's results in an output separate from the original design:

```
batch_drc -nographic design243.brd design243a.brd
```

Related Topics:

- [batch_drc](#)
- [BATCH DRC Dialog Box](#)
- [Making DRC Errors Visible](#)

bbvia

The `bbvia` batch command creates blind/buried vias (BBVias) in a design. The command also generates a log file containing the following information:

- The time and date you invoke the command
- The command line switches and arguments
- A list of created vias
- A description of all warning and error conditions

The interactive version of this command is [auto define bbvia](#).

For more details about blind/buried vias, see *Preparing the Layout* in the user guide.

Syntax

```
bbvia [-p <prefix>] [-t] [-c <cns>] <padname> <startlayer> <endlayer> <input_layout> [<output_layout>]
```

-p <prefix>	(Optional) Specifies a prefix for the names of the vias created by this command. For more information, see Using the -p Switch .
-t	(Optional) Specifies that the command use the pad for the <padname> padstack from the TOP/SURFACE subclass instead of its pad for the highest layer in the via. For more information, see Using the -t Switch .
-c <cns>	(Optional) Adds the created vias to the Current Via List of the physical constraint set specified in the <cns> variable. For more information, see Using the -c Switch . You can enter this switch more than once on the command line.
<padname>	Specifies the padstack whose pads the command copies when it creates vias.
<startlayer>	Defines an ETCH/CONDUCTOR subclass that specifies the layer that starts the range of layers between which this command creates vias.
<endlayer>	Defines an ETCH/CONDUCTOR subclass that specifies the layer that ends the range of layers between which this command creates vias.

<input_layout>	Specifies the layout from which this command gets the layout cross section and padstack information.
<output_layout>	(Optional) Specifies the name of the layout after this command creates vias. If you omit this argument, this command names the output layout with the input layout name.

Related Topics:

- [Using the -t Switch](#)
- [Using the -c Switch](#)

Using the -p <prefix> Switch


The `-p <prefix>` switch attaches a prefix to the left of the names of the vias the `bbvia` command creates. You can use `-p <prefix>` to create more than one set of vias for a design by reentering the `bbvia` command with different `<prefix>` variables for the `-p` switch.

An important use of the `-p <prefix>` switch is to create different sets of vias from different padstacks. For example, the following two commands create two sets of vias for a design:

```
bbvia -p ps1 padstack1 top bottom mlc_drawing_1
```

```
bbvia -p ps2 padstack2 top bottom mlc_drawing_1
```

The vias created by the first command have the `ps1` prefix attached to their names and their pads are created from `padstack1`. The vias created by the second command have the `ps2` prefix attached to their names and their pads are created from `padstack2`. If you did not include the `-p <prefix>` switch in either of these commands, the second command would not result in a set of vias from `padstack2` because their names would conflict with those created by the first command.

 Via names cannot exceed 20 characters. If a via name exceeds this limit, the tool truncates all but the left most 20 characters. If you specify a long prefix and your design contains long ETCH/CONDUCTOR subclass names, the tool might truncate enough of a via name so that the truncated name matches an existing via. If this happens, the `bbvia` command does not create the via.

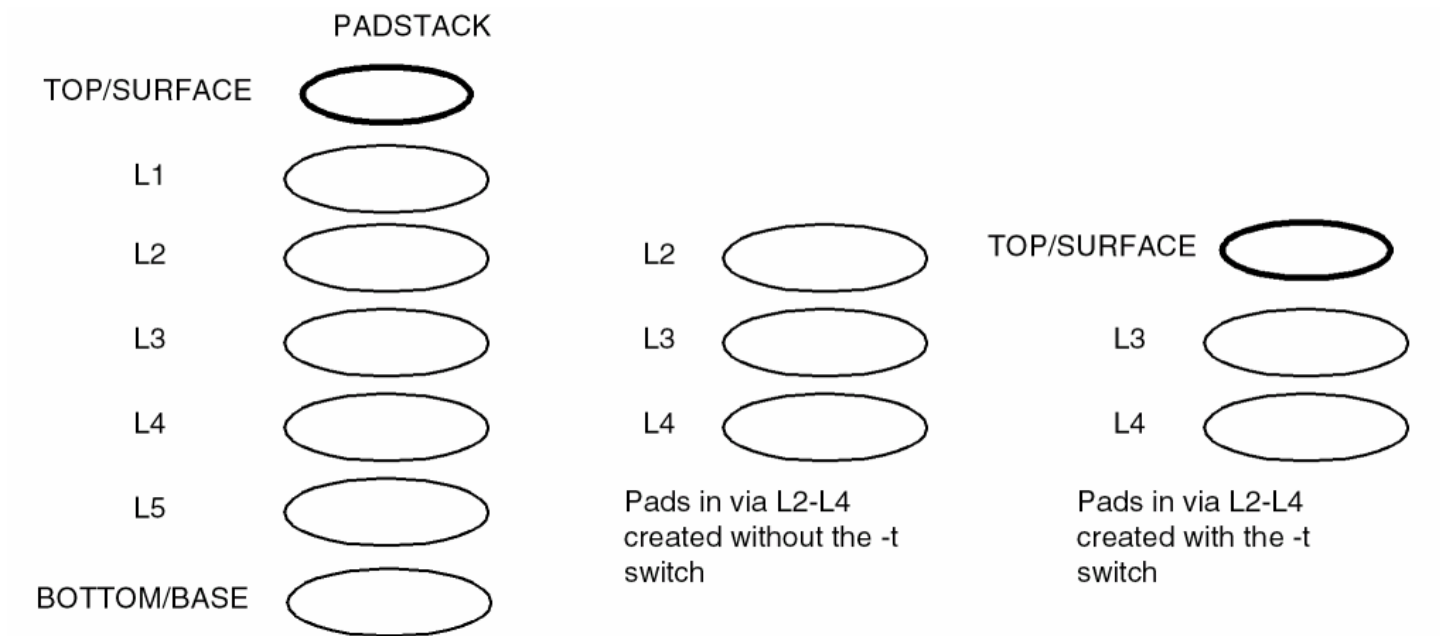
Related Topics:

- [Using the -c Switch](#)

Using the -t Switch

The `-t` switch specifies that the `bbvia` command use the subclass TOP/SURFACE pad in a padstack for the top pad of a BBvia instead of the padstack's highest layer pad. Use this switch in MLC technology where the topmost pad of a padstack specifies the punch that manufactures the via instead of a pad geometry. Figure 1-1 shows the pads in a via created by `bbvia` with and without the `-t` switch.

Figure -1 Pads Created with and without the -t Switch in the `bbvia` Command



Related Topics:

- [bbvia](#)

Using the -c <cns> Switch

Use `-c <cns>` to specify a physical constraint set name. The `bbvia` command adds the vias it creates to that physical constraint set's Current Via List. If you omit this switch, the vias created by `bbvia` are added to the DEFAULT physical constraint set's Current Via List.

You can specify that the vias created by `bbvia` apply to more than one physical constraint set by typing multiple entries of the `-c <cns>` switch. The following command applies the vias created by `bbvia` to the `thinline` and `hicurrent` physical constraint sets:

```
bbvia -c thinline -c hicurrent padstack1 top bottom mlc_drawing_1
```

Related Topics:

- [bbvia](#)
- [Using the -p Switch](#)

bend area create

The `bend area create` command creates a bend area that constitute the flex part of a rigid-flex PCB. The command provides options to create bend lines in the design. You can choose a location where the bend occurs in the flex part and also define angle and radius of the bend.

The bend area is created in the form of a rectangular shape on the RIGID FLEX/BEND_AREA subclass. The dimensions of the rectangle are derived from the length of the bend line, bend radius, and the angle specified for the bend.

There are options to define package and via keepouts areas on the PACKAGE_KEEPOUT/ALL and VIA_KEEPOUT/ALL subclasses around bend areas to avoid the placement of vias, components pads and stiffeners.

The bend line, bend area, and the keepout areas devise a BEND_GROUP that is identified by the name of the bend area. The information on bend areas can be exported for manufacturing process.

Related Topics:

- [Creating Bend Areas](#)

Create Bend Area Dialog Box

Access Using

- *Menu Path: Setup – Bend – Create*

Bend name	Specify the name of the bend area.	
Bend line start	Specify the location of the start-point of the bend line.	
	X:Specify the x coordinate of the start-point.	
	Y:Specify the y coordinate of the start-point.	
Bend line end	Specify the location of the end-point of the bend line.	
	X:Specify the x coordinate of the start-point.	
	Y:Specify the y coordinate of the start-point.	
Bending Parameters	Configures the bend area parameters.	
	Inner Side	Specifies the side of the flex where the bend line is to be created. You can choose either TOP or BOTTOM.
	Inner radius	Specify the bend radius, measured on the <i>Inner Side</i> of the bend.
	Angle	Specify the angle to create bend.
	Order	Specify the sequence of bend areas if multiple bend area with same parameters are created.
Bend Area Options		
	Via keepout	Enable to create via keepout. By default this option is enabled.
	Oversize	Specify the value to create via keep out of same or larger than size of the bend area.By default, the oversize value is 0.

B Commands

B Commands--bend area create

	Package keepout	Enable to create package keepout. By default this option is enabled.
	Oversize	Specify the value to create package keep out of same or larger than the size of the bend area. By default, the oversize value is 0.
Create	Click to create a bend area.	
Cancel	Closes the dialog box without saving bend area settings.	

Creating Bend Areas

Follow these steps to create bend areas:

1. Choose *Setup – Bend – Create* or enter `bend area create` command.
2. Specify a name for the bend area in the *Bend name* field.
3. Specify the X and Y coordinates for the starting and end points of the bend line.
4. Alternatively, you can pick the coordinates from the design canvas.
5. Specify the side of the PCB you want to create the bend area.
6. Specify the bend angle.
7. To create package and via keepouts of larger than bend area, specify the Oversize value
8. Click *Create*.
The bend area is be created with the associated keep out areas.
9. Right-click to choose *Done* from the pop-up menu to complete the command.

Related Topics:

- [bend area create](#)

bend area edit

The `bend area edit` command lets you modify the bending parameters for a bend area. You can also remove a bend from the design. When you delete a bend area all the objects associated with that area are also removed.

Related Topics:

- [Editing Bend Areas](#)

Edit Bend Area Dialog Box

Access Using

- *Menu Path: Setup – Bend – Edit*

Bend name	Specify the name of the bend area.	
Bend line start	Specify the location of the start-point of the bend line	
	X: Specify the x coordinate of the start-point.	
	Y:Specify the y coordinate of the start-point.	
Bend line end	Specify the location of the end-point of the bend line.	
	X:Specify the x coordinate of the start-point.	
	Y:Specify the y coordinate of the start-point.	
Bending Parameters	Configures the bend area parameters.	
	Inner Side	Specify the side of the flex where the bend line is to be created. You can choose either TOP or BOTTOM.
	Inner radius	Specify the bend radius, measured on the <i>Inner Side</i> of the bend.
	Angle	Specify the angle to create bend.
	Order	Specify the sequence of bend areas if multiple bends with same values are created.
Bend Area Options		
	Via keepout	Enable to create via keepout. By default this option is enabled.
	Oversize	Specify value to create via keep out of same larger size than the bend area. By default, the oversize value is 0.

	Package keepout	Enable to create package keepout. By default this option is enabled.
	Oversize	Specify value to create package keep out of same or larger size than the bend area. By default, the oversize value is 0.
Apply	Apply the editing changes to the bend area.	
Cancel	Closes the dialog box without saving bend area settings.	
Delete Bend	Deletes a bend area that is selected in the <i>Bend name</i> field.	

Editing Bend Areas

To edit bend areas, perform the following steps:

1. Choose *Setup – Bend – Edit* or enter `bend area edit` command in the command browser.
2. Choose bend area name from the drop-down list of the *Bend name* field.
The selected bend area is highlighted on the canvas.
3. Modify the X and Y coordinates for the starting and end points of the bend line.
4. Modify the side of the PCB you want to create the bend area.
5. Modify the bend angle.
6. Modify the *Oversize* value for via and package keepout.
7. Click *Apply*.
The modifications in the bend area parameters are reflected on the design canvas.
8. Click *Delete*.
The selected bend area is removed from the design.
9. Right-click to choose *Done* from the pop-up menu to complete the command.

Related Topics:

- [bend area edit](#)

bga abstract export

The `bga abstract export` command exports interface and assignment information. This command creates an XML file that contains the interface and assignment information for a co-design component that you select.

Related Topics:

- [Exporting Interface and Assignment Information](#)

BGA Abstract Export Dialog Box

Access Using

- *Menu Path: File – Export – BGA Abstract File*

Component	Specifies the placed BGA co-design component in the current design for which the interface information will be exported.
Output File	Specifies the name and location of the output XML file.
Browse	Open the File Selection box to browses the location of the output file.
OK	Exports the XML file and closes the dialog box.
Cancel	Exits the command with no action taken.
Help	Displays online help for this command.

Exporting Interface and Assignment Information

Follow these steps to export interface and assignment information:

1. From the open co-design project, choose *File – Export – BGA Abstract File* or enter `bga abstract export` command.
2. Specify the *Component* from the list
3. Click *OK*.

Related Topics:

- [bga abstract export](#)

bga abstract import

The `bga abstract import` command imports interfaces and updates existing co-design BGA in the open design with the interface. Note that the pin numbers and pin names in the imported file and the BGA must match. In addition, the interface hierarchies must match between the imported and existing ones. The import will create an interface hierarchy if one does exist already.

Related Topics:

- [Importing BGA Interfaces](#)

BGA Abstract Import Dialog Box

Component	Specifies the placed BGA co-design component in the current design on which the imported interface should be applied. By default, a co-design BGA with a matching component definition is selected.
Input File	Specifies the name and location of the input XML file that contains the interface definition.
Browse	Open the File Selection box to browses the location of the output file.
OK	Imports and applies the interlace definition and closes the dialog box.
Cancel	Exits the command with no action taken.
Help	Displays online help for this command.

Importing BGA Interfaces

To import BGA interface, perform these steps:


1. Enter the `bga abstract import` command.
2. Specify the *Component* from the list
3. Click *OK*.

Related Topics:

- [bga abstract import](#)

bga editor

The `bga editor` command edits a BGA symbol to represent the specific requirements of the current design without leaving the editor's environment. Using the BGA Editor, you can add, delete, swap, copy, move, modify, view, place, and unplace these design elements: pins and grids.

 The `bga editor` is being replaced by the new symbol editing application mode (*Application Mode – Symbol Edit* from the Setup menu or on right-click).

Additionally, you can create a new BGA component from within the editor by creating a brand new component or by creating a copy of an existing BGA.

The information in this topic describes the controls in the dialog boxes that comprise the BGA Editor, as well as a basic procedure for running the command. For detailed information on the capabilities and constraints of the BGA Editor, and some sample use models, see *Placing the Elements* in the user guide.

The `bga editor` command also generates symbol definitions, which includes a Design for Assembly (DFA) boundary. For additional information about meeting DFA requirements, see *Completing the Design* in the user guide.

Related Topics:

- [Component Editing Dialog Box](#)
- [New Padstack Information Dialog Box](#)
- [Final Verification Dialog Box](#)
- [Starting the BGA Editor](#)

Component Selection Dialog Box

Use this dialog box to choose a BGA symbol for editing or to view information about it. The dialog box contains specific selection options and a set of common controls.

Common Controls	
<i>Back</i>	Inactive
<i>Next</i>	Loads the selected component and its associated information into the editor and moves you to the component editing phase.
<i>Help</i>	Displays user documentation for the BGA Editor in your web browser.
<i>Cancel</i>	Terminates the editing session without changing your design.

Action

This section lets you edit or copy an existing component, or create a new one. Based on the data present in your design, the editor automatically comes up in either Edit or Create mode.

<i>Edit existing component</i>	Lets you select an existing component in the design for editing. You can select the component by clicking on it in the Design Window or selecting the proper reference designator from the drop-down list in the <i>Identifiers</i> frame of the dialog box. If there are no BGAs, this option is disabled.
<i>Create new component</i>	Lets you create a new BGA component. Choosing this option requires that you specify basic information about the component to be created. This option comes up automatically if your design contains no editable components.
<i>Copy existing component and edit the copy</i>	Lets you take an existing component from the design group and copy its information into a new component/symbol at a different location. The new object can then be edited and the original object left unchanged. If there are no components in your design that can be edited, this option is automatically disabled.
Component Details - Identifiers	

<i>Ref Des</i>	The reference designator of the component being edited. This field supports a drop - down list when the editor is in <i>Edit</i> or <i>Copy</i> mode. When in <i>Create</i> mode, you must enter an identifier. The default identifier is either the first reference designator in the list or the name of the component type selected for editing; for example, BGA. If you change modes during an editing session from <i>Edit</i> to <i>Create</i> , the editor appends a numeric value to the current reference designator to maintain a unique value.
<i>Name</i>	Specifies the name to be used for the object you are editing. The default identifier is either the first name in the list or the name of the component type selected for editing; for example, BGA. This option is active only in <i>Create</i> or <i>Copy</i> mode.
Component Details - Origin	
<i>X Coordinate Y Coordinate</i>	Indicates the location where the objects's origin is (if Editing) or should be (if creating and copying), in user-specified design units. The default location in <i>Create</i> mode is the center of your design.
Component Details - Placement	
<i>Rotation</i>	Specifies the degree of rotation to apply to the object when you place it. While you are in edit mode, the object reverts to 00 to facilitate editing.
<i>Mirror placed symbol</i>	When checked, mirrors the selected object.
Component Details - Dimensions	
<i>Width Height</i>	Specifies the dimensions of the object selected for editing. Defaults are 10,000 microns in <i>Create</i> mode if no BGA is present.

Related Topics:

- [New Padstack Information Dialog Box](#)
- [Final Verification Dialog Box](#)
- [Starting the BGA Editor](#)

Component Editing Dialog Box


Use this dialog box to edit the object you selected in the Component Selection dialog box. The Component Editing dialog box is composed of five tabs (or "pages") as well as a set of common controls. The tab that the dialog box opens to depends on the object you previously selected and the mode you are running the editor in.

Common Controls

<i>Item Info</i>	When checked, this button displays an Item Information window that lets you obtain information about the elements you want to edit. Selecting an <i>ItemType</i> from the drop-down list, then positioning the cursor over an instance of that type in your design causes the data associated with that object to appear in the Item Information window. You can view additional information by clicking <i>Display detailed info</i> (which opens a second information window) and then highlight the object under scrutiny by clicking <i>Highlight item</i> .
<i>Snap On/Snap Off</i>	This controls whether your cursor is snapped-to-point as it moves on and off the nearest grid points. The button text indicates the state awaiting activation, not the current condition. When the feature is inactive, the button reads <i>Snap On</i> ; to disable the feature, click <i>Snap Off</i> .
<i>Back</i>	Returns the editor to the component selection phase of the editing process. Moving back cancels all the edits you made in the editing phase of the process. A warning message requires you to confirm this choice.
<i>Next</i>	Completes all your editing changes and regenerates the objects being edited. This control does not move you to the finalization phase of editing under the following conditions: <ul style="list-style-type: none">• You are in an interactive command that the editor cannot automatically complete.• Object regeneration fails. In both cases, an error message is generated in the console window.
<i>Cancel</i>	Terminates the editing session without changing your design.
Pins Tab -Action	

<i>Add</i>	Lets you add new pins to the selected object and activates all fields in the <i>Attributes</i> frame. You add pins by either clicking in the Design Window or by drawing a window in the appropriate area. The first method adds a single pin that is snapped to the nearest grid point; the second method creates pins at each unoccupied grid point inside the window. Note: The selected padstack is displayed on the cursor during the <i>Add</i> process. Padstack color and rotation reflect the current pin use and rotation settings.
<i>Delete</i>	Specifies the default setting for this tab. Lets you delete by pick, <i>Temp Group</i> , or window.
<i>Copy</i>	Lets you copy one or more pins to another location in your design by pick, <i>Temp Group</i> , or window. Multiple-pin selection requires that you choose a reference point for the group. You can then rotate or mirror the group (<i>not</i> the individual pins themselves) before placing it at its new location, using the pop-up menu. The rotation of individual pins is controlled by the dialog box. Because <i>Copy</i> allows you to place multiple instances of your selection, the selected object remains attached to your cursor, until you click right and select <i>Next</i> from the pop-up menu.
<i>Move</i>	Similar to <i>Copy</i> . Lets you move one or more pins to another location in your design by pick, <i>Temp Group</i> , or window. Multiple-pin selection requires that you choose a reference point for the group. You can then rotate or mirror the group (<i>not</i> the individual pins themselves) before placing it at its new location.
<i>Modify</i>	Lets you change the attributes of existing pins by pick, <i>Temp Group</i> , or window. The attributes of your selections appear in the various fields, which you can then modify. If your selections have multiple pin uses, nets, or padstacks, double asterisks (**) appear.
<i>Swap</i>	Lets you pick two pins for swapping. All pin information is swapped <i>except</i> rotation, which remains with the location, not the swapped object. Other attribute options are disabled with this option.
<i>Replace Existing Pins on Copy/Move</i>	Enabled only when you choose the <i>Copy</i> or <i>Move</i> pin actions. Deletes existing pins at locations to which you copy or move a new pin.
<i>Stretch routing on Move</i>	Lets you stretch etch/conductor when you are moving pins.
<i>Rip up routing on Delete</i>	Lets you rip up etch/conductor when you are deleting pins.

Pins Tab - Attributes	
<i>Padstack Name</i>	<p>Lets you choose a padstack by:</p> <ul style="list-style-type: none">• Using the currently designated padstack that appears.• Entering a padstack name, which the tool loads if the name exists in the database or padstack library. If the padstack does not exist, the tool uses the currently designated padstack. If you add a group of pins that have multiple uses, double asterisks (**) appear here.• Clicking <i>Browse</i> displays the New Padstack Information dialog box. You can create a new padstack or choose a padstack from the design's library or database (if a valid padstack exists in the database). For details, see New Padstack Information Dialog Box. <p>When you choose existing pins to delete, modify, copy, or move, the tool updates this field with the padstack name in use if all pins have a common padstack. Otherwise, double asterisks (**) appear, indicating that selected pins use multiple padstacks. All padstack assignments are retained unchanged.</p>
<i>Rotation</i>	<p><i>Applies to any pins being worked on. Choices in the drop-down list include: Automatic, Keep Current, North, South, East, and West.</i> If you choose <i>Automatic</i>, then the design tool selects the appropriate N/S/E/W rotation based on which side of the symbol the pin exists. If you choose <i>Keep Current</i>, then the tool keeps the current pin setting. <i>North</i> uses a 0-degree rotation, <i>West</i> uses a 90-degree rotation, <i>South</i> uses an 180-degree rotation, and <i>East</i> uses a 270-degree rotation.</p>
<i>Pin Use</i>	<p>Lets you select the type of pin for editing, designated as follows:</p> <ul style="list-style-type: none">• <i>Power</i> – POWER• <i>Ground</i> – GROUND• <i>Signal</i> – BI, TRI, LOADIN, LOADOUT, OCA, OCL• <i>Unused</i> – UNSPECIFIED, NO_CONNECT
<i>Net</i>	<p>Lets you select the net on which the pin resides.</p>

<i>Swap Code</i>	<p>Allows you to control the swap group containing the individual pins. By default, the tool groups pins by their pin use only (pins with different pin uses must be in different swap groups). Changing the pin use setting in the dialog box changes the pin swap group to the corresponding group. The default swap group always matches the pin use.</p> <p>To establish subgroups of pins, specify a new swap code in this field and put the pins into that group instead. The 0 swap code is reserved for all pins that are not to be swapped.</p>
<i>Pin Name</i>	<p>Lets you modify the logical pin name associated with selected pins. When the tool starts up, the field is set to <i><Match Pin Number></i>. To modify the pin name, select the pins while in the <i>Modify</i> mode or specify the pin name prior to adding the pins in <i>Add</i> mode.</p> <ul style="list-style-type: none"> • If multiple values exist for the selected pins that you are modifying, the tool displays ** as the field value. If you do not modify this field, the pin names remain unchanged. Entering a new value sets all the pins to that logical pin name. • If you have previously customized the pin name for this item (or items), enter an empty string in this field to reset them to match the pin numbers. • When you change grid numbering settings and you customized pin names, the pins keep their values. If they are not customized, then they follow the pin number's new value after the renumbering to match the grid settings. <div style="border: 1px solid #fde725; padding: 10px; margin-top: 10px;"> <p> Whenever you return to the <i>Add Pins</i> mode, the tool resets this field to match pin numbers.</p> </div>
Pins Tab - Pin Counts	Real-time counters provide updates of the number of pins of each type in your design, designated as follows:
<i>Power</i>	POWER
<i>Ground</i>	GROUND
<i>Signal</i>	BI, TRI, LOADIN, LOADOUT, OCA, and OCL
<i>Unused</i>	UNSPECIFIED, NO_CONNECT

<i>Pins Tab - Apply Changes</i>	Required only when in <i>Modify</i> mode, this control communicates to the editor that you have completed making changes in the Component Editing dialog box.
Grids Tab - Action	
<i>Add</i>	Lets you add a new grid to the current floor plan of the object (after setting the parameter values). Create a window in the appropriate section of your design by moving your cursor to the first location point and clicking the mouse, then repeating the action for the second location point. Potential problems generate an error message that allows you to reselect a grid area and reset the values. You are prompted to confirm this action if it causes pins to be moved, deleted, or renumbered
<i>Delete</i>	Lets you delete a selected grid— <i>other</i> than the base grid—by picking it in the design. Since this action may also delete pins, you are prompted to confirm the action before completion. Grid settings are disabled in this mode.
<i>Modify</i>	Lets you select a grid for editing by picking the grid in the design window, then modifying the settings in the dialog box controls and clicking <i>Apply Changes</i> . Potential problems generate an error message that allows you to reselect a grid area or reset the values.
<i>Copy</i>	Lets you duplicate an existing grid and copy it to another location. The grid can be rotated by selecting <i>Rotate</i> from the right-button pop-up menu. Note: Rotating the grid 900 flips the horizontal and vertical pitch settings as well as the edge distances.
<i>Grids Tab - Attributes</i>	
<i>Name</i>	Specifies the grid being edited. Grids must have unique name for proper identification. The initial grid is, by default, named BASE GRID. You can change this name, but doing so does not alter the characteristics of the base grid. For example, you still cannot delete it.

<i>Priority</i>	Displays the priority drawing order of the selected grid. A lower integer corresponds to a grid drawn beneath a grid with a higher priority. For details on multiple grids and pin number patterning, see <i>Placing the Elements</i> in the user guide.
<i>Keep out</i>	Lets you create restriction areas in the grid for the selected element types: pins, tiles, and drivers. Design elements already in the grid are not affected, so must be deleted using the <i>Delete</i> option in the appropriate tab. This option acts as a lock against new additions.
Grids Tab - Pitch Settings	
<i>Horizontal/Vertical</i>	Lets you control the pin pitch to be used along the <i>x/y</i> axis. These controls can be turned off if a grid without pin pitch is allowed.
<i>Staggered pin configuration</i>	Enables a staggered pin placement grid in the selected grid, causing the values of the pin pitch settings to double. Deactivating this option decreases the pin pitch settings by half.
<i>Edge Inset X/Y</i>	Lets you specify the distance from the grid bounding box to where the grid point matrix starts. The values apply to all sides of the BGA. The exact offset is applied to the lower left corner; extra space that does not evenly divide into a pin pitch or edge inset distance is applied to the upper right side.
Grids Tab - Pin Numbering	
<i>Scheme</i>	Identifies the numbering method used in the selected grid, as defined by the choices in the drop-down list.
<i>First pin</i>	Identifies which corner pin is the first pin in the numbering scheme, as defined by the choices in the drop-down list.
<i>Prefix</i>	Lets you attach a prefix designation to pin numbers in the selected grid.
<i>Start at</i>	Lets you designate the pin numbering offset to use by defining the pin number for the first pin, as specified in <i>First pin</i> .
<i>Label with letters before numbers</i>	Creates alphanumeric pin numbers in the form of A1, A2, and so on. If you do not choose this option, pin numbers take the form 1A, 1B, and so on. This option affects the pin text only, not the labeling scheme itself, and is enabled only for numbering patterns that contain letters and numbers.
<i>Omit letters as per JEDEC standard</i>	Specifies that the pin numbers of this BGA conform to that standard, omitting the letters I, O, Q, S, X, and Z when generating alpha or alphanumeric pin numbering.

<i>Pad letters with A's</i>	Specifies that the alphabetic portion of pin numbers be of equal length. For example, if there are 30 alpha strings in a symbol using JEDEC standards, naming runs from AA through BK, rather than from A through AK.
<i>Pad numbers with zeroes</i>	Specifies that the numeric portion of pin numbers be of equal length. Leading zeroes are added where needed.
<i>Label unused grid positions</i>	Lets you label grid positions where no pins reside. This option is disabled for some numbering schemes, such as alphanumeric.
<i>Reserve labels for non-staggered positions</i>	Lets you reserve pin numbers for missing positions in a staggered pattern. This option is most useful in conjunction with spiral numbering patterns. It is inactive in non-staggered configurations.
Grids Tab - Apply Changes	
Completes the edits you have currently made. You are prompted to confirm your edits if they cause the editor to renumber, move, or delete existing pins.	
Boundary Tab - Placement Extents	
Edit the following values and the symbol resizes accordingly as long as the new size does not leave any pins outside the extents. Grids are automatically adjusted to remain legal, but no pins change position.	
<i>Lower left X/Y</i>	Edit to resize the lower left corner values.
<i>Upper right X/Y</i>	Edit to resize the upper right corner values.
<i>Select new outline shape</i>	Lets you select a shape or rectangle to use as the new symbol boundary. You can define symbols with notched borders without using the symbol editor tool (.dra).
<i>Text</i>	
<i>Enable text labels for individual pins</i>	Lets you create text on the pin number subclass for each pin in the symbol. The text is displayed unrotated and placed at the specified offset to the owning pin. Its size is specified by the selection chosen in the <i>Text Size</i> drop-down list.
<i>Offset X/Y</i>	Lets you set the distance from the center of the pins to the center of the text for ease of readability.
<i>Text Size</i>	Specifies the text block size of the pin text labels. You can select only from the drop-down list.

<i>Enable border numbers</i>	Lets you create text around the outside border of the symbol. This option is designed to be used only on designs with a single grid.
<i>Offset</i>	Lets you specify the distance border text should be placed from the symbol's boundary box.
<i>Text Size</i>	Specifies the text block size of the boundary text. You can select only from the drop-down list.
Boundary Tab - Name	
<i>Symbol Name</i>	The symbol name to be used for the object you are editing.Lets you create a new name to match the edited symbol, to differentiate it from the library symbol name.

Related Topics:

- [bga editor](#)
- [Final Verification Dialog Box](#)
- [Starting the BGA Editor](#)

New Padstack Information Dialog Box

Use this file browser dialog box to create a new padstack or choose a padstack from the design's library or database (if a valid padstack exists in the database).

<i>New</i>	Lets you create a new padstack in the <i>Specifications</i> frame.
<i>Available Padstack</i>	Lets you select one of the padstack definitions already existing in the current design. You then use this padstack to create pins.
<i>Load from Disk</i>	Enables the <i>Browse</i> button to navigate to a padstack definition in your library of pads (as defined in the PADPATH environment variable). Once selected, you use this padstack to create pins.
<i>Name</i>	Specifies the name of the padstack you create, or, if in another mode, the name of the padstack to be used.
<i>Layer</i>	Specifies the package layer on which to place the package pin.
<i>Circle</i>	Specifies the default condition. When selected, uses a circular pin shape.
<i>Rectangle</i>	When selected, uses a rectangular or square pin shape.
<i>Width</i>	Specifies the width to use. When you change this dimension, dimensions in the height field are adjusted automatically to match.
<i>Height</i>	Specifies the height for the new pins. When you change this dimension, dimensions in the width field are not adjusted, unless you're in circle shape.
<i>Ok</i>	Places the selected padstack into the current design.
<i>Cancel</i>	Closes the dialog box without creating or placing a padstack.

Related Topics:

- [bga editor](#)
- [Component Selection Dialog Box](#)
- [Starting the BGA Editor](#)

Final Verification Dialog Box

The last dialog box in the editor appears after you have integrated your changes and regenerated the symbol. At this point, you have the following options for proceeding.

<i>Display all rats after exiting</i>	When checked, this option displays all ratsnest lines in your design upon completion of your editing session.
<i>Run batch DRC checks</i>	When checked, this option runs a batch check of all DRCS in your design upon completion of your editing session.
<i>Run derive connectivity</i>	When checked, this option ensures that connect lines (clines) get reconnected routed pins. This function is detailed in derive connectivity .
<i>Run purge unused nets</i>	When checked, this option removes any unused nets from your design. This function is detailed in purge unused nets .
<i>View Log</i>	Opens the <code>bga_editor.log</code> file so you can examine the results of your editing session.
<i>Back</i>	Returns you to the Component Editing phase of editing if you need to make changes before ending the session.
<i>OK</i>	Commits the changes to your design and ends the editing session, returning you to the Idle state.

Related Topics:

- [bga editor](#)
- [Component Selection Dialog Box](#)
- [Component Editing Dialog Box](#)

Starting the BGA Editor

Follow these steps to start and run the BGA editor:

1. Create a preliminary BGA symbol using the [bga generator](#) or the [bga text in](#) command. –or– Choose an existing BGA symbol to modify.
2. Run the `bga editor` command to display the BGA Selection dialog box.
3. In the *Action* frame of the dialog box, choose whether to edit the existing component, create a new component, or copy the existing component and edit the copy (leaving the original intact).
4. If you choose the *Create* or *Copy* actions, complete the Component Details field information as described in [Component Selection Dialog Box](#).
5. Click *Next* to accept the currently selected symbol for editing.
6. Edit the object by setting selections and parameters in the various tab pages of the Component Editing dialog box, as described in [Component Editing Dialog Box](#).
7. Click *Next* to move to the Final Verification dialog box. Follow the instructions, as described in [Final Verification Dialog Box](#).

Related Topics:

- [bga editor](#)
- [Component Selection Dialog Box](#)
- [Component Editing Dialog Box](#)
- [New Padstack Information Dialog Box](#)

bga generator

The `bga generator` command displays the BGA Generator wizard, where you can experiment with different package configurations and generate the package easily without using the symbol and padstack editors to create a padstack. For customizing, however, you must use the Padstack or Symbol editors.

You can also use the BGA Generator to create a plating bar, if you do not want to use the automatic plating bar generator.

The `bga generator` command also generates symbol definitions, which includes a Design for Assembly (DFA) boundary. For additional information on meeting DFA requirements, see *Completing the Design* in the user guide.

Access Using

- *Menu Path: Add – Standard Package– BGA Generator*

Related topics:

- [bga editor](#)
- [bga text in](#)
- [BGA Generator - Pin Arrangement Dialog Box](#)
- [BGA Generator - Pin Use Ratios Dialog Box](#)
- [BGA Generator - Padstack Information Dialog Box](#)
- [Padstack for Component Dialog Box](#)
- [BGA Generator - Pin Numbering Dialog Box](#)
- [BGA Generator - Preview Dialog Box](#)
- [Creating a BGA](#)

BGA Generator - General Information Dialog Box

Access Using

- *Menu Path: Add – Standard Package– BGA Generator*

Use these options to specify the BGA package name, placement, and dimensions.

<i>Identifiers</i>	
<i>Name</i>	Specifies the name of the .dra and .psm BGA package symbol. The default symbol name is UNNAMED_BGA when you initially run the BGA Generator wizard. The command subsequently uses settings from the previous session as the default.
<i>Ref Des</i>	Specifies the reference designator for the BGA symbol. The default setting is BGA. The symbol name and reference designator become a logical part in the database just as if you imported a netlist containing them.
<i>Origin</i>	
<i>X Coordinate Y Coordinate</i>	Specifies the X and Y coordinates in the drawing that are used as the center for the BGA symbol. The default setting is 0.0, 0.0. This behaves as if the symbol origin were at the body center and then placed at these X and Y coordinates. The origin must be within the design. The tool validates these coordinates when you invoke the BGA Generator and each time the values change.
<i>Placement</i>	
<i>Mirror placed symbol</i>	<i>If you check this box</i> , when you place the symbol, the tool sets the MIRROR_GEOMETRY flag. As a result, the pin grid created is mirrored through the Y-axis of the symbol instance origin.

Related Topics:

- [BGA Generator - Pin Use Ratios Dialog Box](#)
- [BGA Generator - Padstack Information Dialog Box](#)
- [Padstack for Component Dialog Box](#)
- [BGA Generator - Pin Numbering Dialog Box](#)
- [BGA Generator - Preview Dialog Box](#)
- [Creating a BGA](#)

BGA Generator - Pin Arrangement Dialog Box

Access Using

- *Menu Path: Add – Standard Package– BGA Generator*

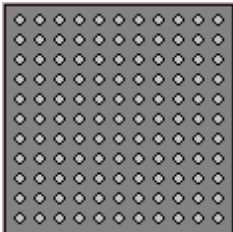
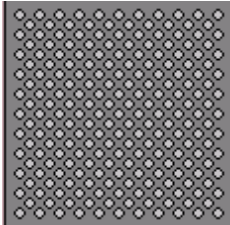
Use the options in this dialog box to specify the pattern for the pins. The graphical display changes dynamically to reflect the type of pattern you choose.

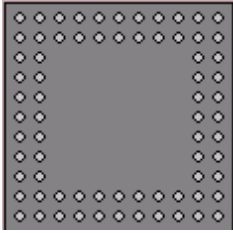
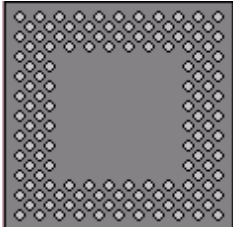
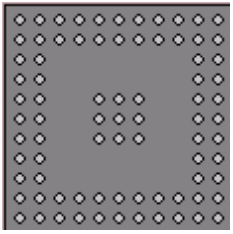
For power and ground pads, you may require larger staggered core pins while using an unstaggered pattern for the outer signal balls. By choosing a perimeter matrix pin arrangement, you can specify separate staggering options for the core and perimeter pins.

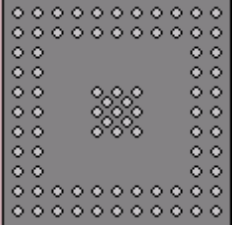
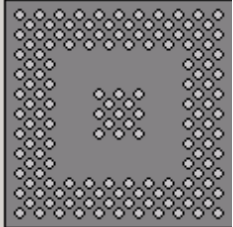
You can fix one of the three parameters: package size (*Width and Height*), *Pin Pitch*, or *Edge Spacing* so that the tool does not recalculate the fixed value when you change other parameters. For example, changing *Pin Pitch* may result in the tool's recalculating the *Edge Spacing* distance. To prevent the change, you can check the *Fix* box in the *Edge Spacing* frame, which results in the package size changing to accommodate the pin pitch. These parameters are also affected by modifying pad dimensions, which are specified later in the wizard.

If you fix one of these parameters and the tool determines that it requires a value change, you receive a pop-up confirmation dialog box showing the original and new values. If you do not accept the change, the modified value that caused this change resets to its previous value. For additional information, see [Rules Used in Recalculation of Die Size, Edge Spacing, and Pin Pitch](#).

<i>Dimensions</i>	
<i>JEDEC Standard BGA</i>	Specifies a standard JEDEC BGA. When you choose this option, the tool limits the values you can use in the <i>Dimensions</i> , <i>Arrangement</i> , and <i>Pin Pitch</i> frames.
<i>Custom BGA</i>	Specifies a custom BGA.
<i>Width Height</i>	Specifies the width and height of the BGA package using positive integers. The default is 1350 x 1350 mil. The dimensions must be less than the drawing size, and the extents of the BGA package must be within the design. If the BGA package boundaries extend beyond the design boundaries, the tool centers the BGA package origin in the design. If the BGA package still fails to fit inside the design boundaries, the tool readjusts the BGA package dimensions to fit within the design boundaries. The tool displays a message at the bottom of the dialog box to indicate the state of the relationship between the design size and the BGA package size.

<i>Fix</i>	<p>If you check this box, the tool preserves the values in the <i>Width</i> and <i>Height</i> fields in future calculations. If the tool determines that changing this field's value is necessary to maintain a proper package, such as when you try and change the pad size for a JEDEC BGA, you are prompted with the original and new values. You can decide whether or not to continue with the change. The tool lets you check only one of the three <i>Fix</i> check boxes. The default setting is that all <i>Fix</i> boxes are unchecked. If you choose <i>JEDEC Standard BGA</i>, the <i>Width</i> and <i>Height</i> values are automatically fixed.</p>
<i>Arrangement</i>	
<i>Columns</i>	Specifies the number of pins in a column. The default is 26 pins.
<i>Rows</i>	<p>Specifies the number of pins in a row. The default is 26 pins. The tool automatically adjusts the <i>Pin Pitch</i> and <i>Edge Spacing</i> values to fit the new pins if you have not checked the <i>Fix</i> box. For wire bond, it adjusts the <i>Pin Pitch</i>; for flip chip, the <i>Edge Spacing</i>. For additional information, see Rules Used in Recalculation of Die Size, Edge Spacing, and Pin Pitch. The total number of pins in the package appears at the right. The tool displays a warning message at the bottom of the dialog box if the number of pins exceeds the boundary of the package.</p>
<i>Full matrix</i>	<p>Specifies a full array of pins. Pins are evenly spaced depending on the values in the <i>Pin Pitch</i>, <i>Columns</i>, and <i>Rows</i> fields. Example of Full Matrix Pin Arrangement</p> 
<i>Stagger full</i>	<p>Creates a staggered pin pattern over the entire BGA symbol by inserting an extra row of pins at a staggered interval in both dimensions. Example of Full Matrix Staggered Pin Arrangement</p> 

<p><i>Perimeter matrix</i></p>	<p>Specifies a perimeter array of pins. You can control the number of rows on the outer perimeter and whether or not you want a core of staggered or unstaggered pins in the center of the package. Example of Unstaggered Perimeter Matrix with No Core Pins</p> 
<p><i>Outer rings</i></p>	<p>Defines the number of perimeter rows. The default is 4 with no corner pins.</p>
<p><i>Stagger outer</i></p>	<p>Staggeres the perimeter pins by inserting an extra row of pins at an staggered interval in both dimensions. Example of Staggered Perimeter Matrix and No Core Pins</p> 
<p><i>Core columns</i> <i>Core rows</i></p>	<p>Specifies the core size. Enter positive integers in each field to indicate the actual number of pins per row and per column, not the total number of rows and columns the core occupies. For example, to have a 2 x 2 rectangular core with <i>core multipliers</i> set to 2 (meaning the pins are twice as far apart), specify 2 rows and 2 columns rather than a 3 x 3 core. Setting these fields to 0 disables the <i>Stagger core</i> field. The default settings for layer, shape, and size of the pads are the same as those for the perimeter padstack. Example of Unstaggered Perimeter Matrix with Unstaggered Core Pins</p> 

<i>Stagger core</i>	<p>Stagger the pattern of the core pins by inserting an extra row of pins at a staggered interval in both dimensions. Example of Unstaggered Perimeter Matrix with Staggered Core Pins</p>  <p>Example of Staggered Perimeter Matrix with Staggered Core Pins</p> 
<i>Pin Spacing</i>	
<i>Pin Pitch</i>	<p>Specifies the horizontal center-to-center distance between pins along the X-axis, and the vertical center-to-center distance between pins along the Y-axis.</p>
<i>Horizontal Vertical</i>	<p>Specifies the horizontal and vertical spacing between pins in the same column or row.</p>
<i>Fix</i>	<p>If you check this box, the tool preserves the <i>Pin Pitch</i> values in future calculations.</p> <p>If the tool determines that changing this field's value is necessary to maintain a proper package, you are prompted with the original and new values. You can decide whether or not to continue with the change.</p> <p>The tool lets you check only one of the three <i>Fix</i> check boxes. The default setting is that all <i>Fix</i> boxes are unchecked.</p> <p>If you choose <i>JEDEC Standard BGA</i>, the <i>Pin Pitch</i> values are automatically fixed.</p>
<i>Core spacing</i>	<p>Specifies the core spacing. Enter a positive integer for the <i>Horizontal</i> and <i>Vertical</i> fields if you chose a <i>Perimeter matrix</i> pin arrangement and defined a core area. A value of 1 indicates that the <i>Pin Pitch</i> remains the same in the core area.</p>
<i>Edge Spacing (Dx and Dy)</i>	<p>Specifies how far from the symbol outline the pins are placed in the X- and Y-axis fields. These fields can be edited only if <i>Custom</i> is selected in the <i>Dimensions</i> section.</p>

<i>Fix</i>	<p>If you check this box, the tool preserves the <i>Edge Spacing</i> values in future calculations.</p> <p>If the tool determines that changing this field's value is necessary to maintain a proper package, you are prompted with the original and new values. You can decide whether or not to continue with the change.</p> <p>The tool lets you check only one of the three <i>Fix</i> check boxes. The default setting is that all <i>Fix</i> boxes are unchecked.</p> <p>If you choose <i>JEDEC Standard BGA</i>, the <i>Edge Spacing</i> values are automatically fixed.</p>
<i>Back</i>	Returns to the BGA Generator - General Information dialog box where you can edit and apply changes to previously defined settings.
<i>Next</i>	Accepts the entries. The BGA Generator - Pin Use Ratios dialog box appears, where you can continue to edit and apply changes.
<i>Cancel</i>	Cancels the operation and closes the dialog box.

Related Topics:

- [bga generator](#)
- [BGA Generator - Padstack Information Dialog Box](#)
- [Padstack for Component Dialog Box](#)
- [BGA Generator - Pin Numbering Dialog Box](#)
- [BGA Generator - Preview Dialog Box](#)
- [Creating a BGA](#)

BGA Generator - Pin Use Ratios Dialog Box

Access Using

- *Menu Path: Add – Standard Package– BGA Generator*

Use this dialog box to specify the ratio of power-to-ground-to-signal pins when you create the perimeter pins. The ratio you supply is used to create a spiral pattern of pin uses to ensure even ratio distribution, and assign power and ground pins to the appropriate nets resident in the design. The original default distribution settings for power:ground:signal are 1:1:4. As is the case with all defaults, if you run the generator more than once, it uses the settings from previous sessions as the current defaults.

Related Topics:

- [bga generator](#)
- [BGA Generator - General Information Dialog Box](#)
- [Padstack for Component Dialog Box](#)
- [BGA Generator - Pin Numbering Dialog Box](#)
- [BGA Generator - Preview Dialog Box](#)
- [Creating a BGA](#)

BGA Generator - Padstack Information Dialog Box

Access Using

- *Menu Path: Add – Standard Package– BGA Generator*

Use these options to specify the padstack definitions for your package.

<i>Method</i>	Specifies the type of padstack that you are using for this BGA symbol. The default setting is <i>New</i> .
<i>New</i>	Defines a new padstack, specifying the dimensions of the BGA pins instead of using an existing padstack from the design or library.
<i>Available padstack</i>	Uses a padstack that already exists in the design. If no padstacks currently exist, you cannot choose this option. When you choose a padstack from the list, the <i>Specifications</i> frame reflects the padstack information. You cannot edit the specifications.
<i>Load from disk</i>	Imports an external padstack definition. Clicking <i>Browse</i> lets you locate a padstack on your disk. The <i>bundle show</i> appears. When you import the padstack, the <i>Specifications</i> frame reflects the padstack information. You cannot edit the specifications.
<i>Specifications</i>	
<i>Name</i>	Specifies the name to use when the tool creates the padstack. For perimeter padstacks, the default name is <code>BGA_PAD</code> ; for core padstacks, <code>BGA_CORE_PAD</code> .
<i>Layer</i>	Defines the conductor layer for the padstack. The default setting is <i>BOT_COND</i> .
<i>Shape</i>	Defines the padstack shape. The default setting is <i>Circle</i> .
<i>Dimensions</i>	
<i>Width Height</i>	Specifies the diameter for the circle, or width and height for the rectangle. The default pad size is 25 x 25 mils.

Related Topics:

- [bga generator](#)
- [BGA Generator - General Information Dialog Box](#)
- [BGA Generator - Pin Arrangement Dialog Box](#)
- [BGA Generator - Pin Numbering Dialog Box](#)
- [BGA Generator - Preview Dialog Box](#)
- [Creating a BGA](#)

Padstack for Component Dialog Box

Access Using

- *Menu Path: Add – Standard Package– BGA Generator*

Use this dialog box to find and choose a padstack easily. All padstacks are listed in alphabetical order.

Rules Used in Recalculation of Die Size, Edge Spacing, and Pin Pitch

When you update the pad size, the tool recalculates the BGA size, *Edge Spacing*, and *Pin Pitch*.

Custom BGA

If you change the pad size on a Custom BGA to a value that is less than or equal to the pin pitch, the tool operates as follows:

1. Changes the *Edge Spacing* values if they are not fixed.
2. If you fixed the *Edge Spacing* values, the tool changes the BGA size.
The tool provides a pop-up confirmation dialog box when it needs to adjust a *fixed* field.

If you change the pad size on a Custom BGA to a value that is greater than the *Pin Pitch*, the tool operates as follows:

- Provides a pop-up confirmation dialog box when it needs to adjust the values for *Pin Pitch* and BGA size (*Width* and *Height*).
If you click *Yes in the dialog box*, the tool adjusts the *Pin Pitch* to the pad size (*Width/Height*) and the minimum DRC spacing on the pad layer. Then it recalculates the BGA size.
If you click *No in the dialog box*, the tool does not make any change. The pad size resets to its original value.

JEDEC Standard BGA

If you change the pad size on a JEDEC Standard BGA to a value that is less than or equal to the *Pin Pitch*, the tool operates as follows:

- Changes the *Edge Spacing* values based on the new *Pin Pitch* and pad size values.

If you change the pad size on a JEDEC Standard BGA to a value that is greater than the *Pin Pitch*, the tool operates as follows:

- Changes the *Edge Spacing* values based on the new *Pin Pitch* and pad size values.

Related Topics:

- [bga generator](#)
- [BGA Generator - General Information Dialog Box](#)
- [BGA Generator - Pin Arrangement Dialog Box](#)
- [BGA Generator - Pin Use Ratios Dialog Box](#)
- [BGA Generator - Preview Dialog Box](#)
- [Creating a BGA](#)

BGA Generator - Pin Numbering Dialog Box

Access Using

- *Menu Path: Add – Standard Package– BGA Generator*

Use these options to specify the numbering scheme for the pins. The graphical display changes dynamically to represent the numbering scheme you choose.

<i>Pin Numbering</i>	
<i>Ordering</i>	Specifies the way you want the pins numbered. The graphical display changes to display the numbering scheme you choose. <i>CW</i> represents clockwise, and <i>CCW</i> represents counterclockwise. The default setting is <i>Number Horiz Letter Vert</i> (number horizontal, letter vertical).
<i>Start at</i>	Defines the position from which to start the pin numbering. The default settings are <i>Top</i> and <i>Left</i> .
Label with letters before numbers	Creates alphanumeric pin numbers in the form of A1, A2, and so on. If you do not choose this option, pin numbers take the form 1A, 1B, and so on. This option affects the pin text only, not the labeling scheme itself. This is the default setting.
<i>JEDEC standard</i>	Specifies that the pin numbers of this BGA conform to that standard, omitting the letters I, O, Q, S, X, and Z when generating alpha or alphanumeric pin numbering. This is the default setting.
<i>Pad letter with A's</i>	Specifies that the alphabetic portion of pin numbers be of equal length. For example, if there are 30 alpha strings in a symbol using JEDEC standards, naming runs from AA through BK, rather than from A through AK.
<i>Pad number with zeros</i>	Specifies that the numeric portion of pin numbers be of equal length. Leading zeroes are added where needed. For example, if a package of 10x10 pins is generated using <i>Number Horiz Letter Vert</i> ordering, these are the resulting numbers: <ul style="list-style-type: none">• A001, A002.....A010• B011, B012.....B020• C021, C022.....C030• J091, J092.....J100

<i>Display Settings</i>	
<i>Display pin numbers</i>	Defines the sides of the package where the pin numbers appear.
<i>Left, Top Right, Bottom</i>	Specifies the package sides on which to display the pin numbers. The default settings are <i>Left</i> and <i>Top</i> .
<i>Text Size</i>	Defines the size of the pin number text. The default setting is 25.0 x 16.0.
<i>Distance from symbol edge</i>	Specifies the distance between the pin number text and the outside of the symbol. The default is 150 mils.

Related Topics:

- [bga generator](#)
- [BGA Generator - General Information Dialog Box](#)
- [BGA Generator - Pin Arrangement Dialog Box](#)
- [BGA Generator - Pin Use Ratios Dialog Box](#)
- [BGA Generator - Padstack Information Dialog Box](#)
- [Creating a BGA](#)

BGA Generator - Preview Dialog Box

Access Using

- *Menu Path: Add – Standard Package– BGA Generator*

After the tool generates the specified package, this dialog box appears. Once you preview the package symbol, do one of the following:

- Click *Finish* to accept the current symbol.
When you click *Finish*, the tool writes the package symbol to the database. If you make a mistake, the only way you can use the package name and reference designator again is to delete the package information from the database. Removing all instances of the symbol does not remove the package name and reference designator from the database.
- Click *Back* to modify settings on previous dialog boxes.
- Click *Cancel* to exit the BGA Generator wizard without creating a symbol.


Related Topics:

- [bga generator](#)
- [BGA Generator - General Information Dialog Box](#)
- [BGA Generator - Pin Arrangement Dialog Box](#)
- [BGA Generator - Pin Use Ratios Dialog Box](#)
- [BGA Generator - Padstack Information Dialog Box](#)
- [Padstack for Component Dialog Box](#)

Creating a BGA

Use the BGA Generator to establish the package outline, padstacks, pin numbering assignments, and pin arrangement for your package.

1. Run the `bga generator` command.
2. Complete the BGA Generator - General Information dialog box. For details, see [BGA Generator - General Information Dialog Box](#).
3. Click *Next*.
4. Complete the BGA Generator - Pin Arrangement dialog box. For details, see [BGA Generator - Pin Arrangement Dialog Box](#).
5. Click *Next*.
6. Complete the BGA Generator - Pin Use Ratios dialog box. For details, see [Creating a BGA](#).
7. Click *Next*.
8. Complete the BGA Generator - Padstack Information dialog box. For details, see [BGA Generator - Padstack Information Dialog Box](#).

 If you create a package with perimeter and core pins, this dialog box appears twice to let you specify different padstacks for the two types of pin. An indicator changes above the *Method* field to reflect which particular padstack you are defining.

9. Click *Next*.
10. Complete the BGA Generator - Pin Numbering dialog box. For details, see [bundle show](#).
11. Click *Next*.

The tool generates the package symbol and displays the BGA Generator - Preview dialog box.

1. Verify that the generated die package meets your design requirements by examining it in the Design Window.
2. If necessary, click *Back* to make any changes to the symbol using the previous dialog boxes.
3. Click *Finish*.

Related Topics:

- [bga generator](#)
- [BGA Generator - General Information Dialog Box](#)
- [BGA Generator - Pin Arrangement Dialog Box](#)
- [BGA Generator - Pin Use Ratios Dialog Box](#)
- [BGA Generator - Padstack Information Dialog Box](#)
- [Padstack for Component Dialog Box](#)
- [BGA Generator - Pin Numbering Dialog Box](#)

bga text in

The `bga text in` command brings up the BGA Text-In Wizard where you can:

- Generate BGA symbols, nets, and properties by importing an ASCII spreadsheet of BGA pin information.
- Place columns of data in a standard format.
- Define pad shape and size for padstacks directly inside the BGA text file.
This allows the design to be portable to sites that do not have the padstack libraries. If the pads are defined in the file, you are not prompted to define them.

To add logic to a BGA package and pins after creating them in another way, use the BGA Text-In Wizard.

The `bga text in` command also generates symbol definitions, which includes a Design for Assembly (DFA) boundary. For additional information about meeting DFA requirements, see *Completing the Design* in the user guide.

Related Topics:

- [bga generator](#)
- [bga editor](#)
- [Importing BGA Pin Data](#)

BGA Text In Dialog Box

Access Using

- *Menu Path: Add – Standard Package – BGA Generator*

BGA Text-In Wizard, Step 1: File Selection Dialog Box

A standard file browser.

BGA Text-In Wizard, Step 2: File Information Dialog Box

Coordinates	Specifies the unit-type of measurement available in the drop-down list.
<i>Absolute</i>	Indicates that the <i>X/Y</i> coordinates for pin locations in the database file are relative to the origin of the design.
<i>Relative</i>	Indicates that the <i>X/Y</i> coordinates for pin locations in the database file are relative to the origin of the symbol.
Delimiters	
<i>Tab</i>	Choose to use a tab to separate columns of data.
<i>Semicolon</i>	Choose to use a semicolon to separate columns of data.
<i>Comma</i>	Choose to use a comma to separate columns of data.
<i>Space</i>	Choose to use a space to separate columns of data.
<i>Other</i>	Choose to use other characters to separate columns of data.
<i>Ignore consecutive delimiters</i>	Choose to treat consecutive delimiters as one delimiter.
<i>Remove trailing delimiters</i>	Choose to remove trailing delimiters from the data.
<i>Units</i>	Specify the type of measurement unit that is represented by your pin data (mils, for example).
<i>Back</i>	Click to return to the previous dialog box.
<i>Next</i>	Click to display the next dialog box.
<i>Cancel</i>	Ignores your input and closes the dialog box.

BGA Text-In Wizard, Step 3: Pin Information Dialog Box

Information contained within this dialog box includes the saved grid parameters for the symbol. The columns in which this information appears depends on the delimiter types (tabs, semicolons, and so on) you selected in the File Information dialog box. Editing grid parameters is *not* recommended.

<i>Ignore Rows</i>	Data in this column is not imported.
--------------------	--------------------------------------

<i>Ignore</i>	Denotes that this line should be ignored (comment line).
<i>Pin Number</i>	Specifies the pin numbers of the pins in the symbol. (Required. You cannot proceed if no column is denoted for pin numbers.)
<i>Pin Name</i>	Specifies the logical pin name that is different from the physical pin number.
<i>Mixed Case Pin Number</i>	Allows you to import and export mixed-case names of the object, for example, from LEF/DEF or OpenAccess.
<i>Padstack:</i>	Specifies the padstack type of each pin in the symbol.
<i>X Coordinate</i>	Specifies the X coordinate of each pin in the symbol. (Required. You cannot proceed to step 3 if no column is denoted.)
<i>Y Coordinate</i>	Specifies the Y coordinate of each pin in the symbol. (Required. You cannot proceed to step 3 if no column is denoted.)
<i>Rotation</i>	Specifies the value in degrees of each pin in the symbol.
<i>Package Pin</i>	Specifies the logical connection for each pin to a corresponding package pin.
<i>Net Name</i>	Specifies the net names assigned to each pin in the symbol.
<i>Mixed Case Net Name</i>	Allows you to import and export mixed-case names of the nets, for example, from LEF/DEF or OpenAccess.
<i>Net Prop Name</i>	Specifies the property names of the nets.
<i>Net Prop Value</i>	Specifies the property values of the nets.
<i>Pin Prop Name</i>	Specifies the property names of the pins.
<i>Pin Prop Value</i>	Specifies the value of the property to assign to this pin.
<i>Back</i>	Click to return to the previous dialog box.
<i>Next</i>	Click to display the next dialog box.
<i>Cancel</i>	Ignores your input and closes the dialog box.

BGA Text-In Wizard, Step 3A: New Padstack Information Dialog Box

Use these options to specify the padstack definitions for your package. If the pads are already defined in the file, the Step 3A screen does not appear.

Method	
<i>New</i>	Click this button to define a new padstack. Fill in all the specification settings at the bottom of the dialog box.
<i>Available Padstack</i>	Click this button to choose a padstack from the design's database. This option is available only if a valid padstack exists in the database. When you choose the padstack from the adjacent list box, the <i>Specifications</i> boxes reflect the padstack information. You cannot edit the padstack specifications.
<i>Load from Disk</i>	Click this button to import an external padstack definition. Use the <i>Browse</i> button to locate the padstack on your disk. When you import the padstack, the <i>Specifications</i> boxes reflect the padstack information. You cannot edit the padstack specifications.
Specifications	
<i>Name</i>	Specifies the padstack name. If you are defining a new padstack, enter the name in this box
<i>Shape</i>	Specifies the padstack shape. Choose either the <i>Circle</i> or a <i>Rectangle</i> button.
<i>Dimensions</i>	Indicates the dimensions of the padstack. Enter values for the <i>Width</i> and <i>Height</i> .
<i>Back</i>	Click to return to the previous dialog box.
<i>Next</i>	Click to display the next dialog box.
<i>Cancel</i>	Ignores your input and closes the dialog box.

BGA Text-In Wizard, Step 4: Package Information Dialog Box

<i>Name</i>	Indicates the name used for the .dra and .psm symbol.
<i>Ref Des</i>	Indicates the reference designator for the BGA symbol. This becomes a logical part in the database just as if a netlist had been imported.
<i>Origin</i>	Enter the X and Y coordinates of the location in the design where you want to place the BGA. The origin must be within the design. These values are checked for validity when you invoke the BGA Text-In Wizard and each time the values change.

<i>Center pins on symbol origin</i>	Use if the BGA does not have an origin at the center. If you enable this option, the origin of the BGA symbol definition is translated to the center of the BGA pins.
<i>Rotation</i>	Specify the rotation of the BGA. The default value is that specified in the text file's header line or 0.000 if not specified.
<i>Pad layer</i>	Specifies the layer on which the padstack is placed. Choose the pad layer from the list to ensure that these components are single-layer only, and that padstacks coming in from the library are moved to the correct layer for a placed instance of the symbol. The pad layer specified in the text file is selected by default. If pad layer is not specified in the text file, <i>SECONDARY</i> is selected.
<i>Dimensions</i>	<p>Choose a method to establish the dimensions of the package:</p> <ul style="list-style-type: none">• <i>Center around origins</i>: Enter positive values for the <i>Width</i> and <i>Height</i> of the BGA package. The default values reflect the size specified in the file header; otherwise, the extent of the pin data. The dimensions must be less than the drawing size and the extents of the BGA package must be within the design. If the BGA package boundaries extend beyond the design boundaries, the origin of the BGA package is placed in the center of the design. If the BGA package still fails to fit inside the design boundaries, the dimensions of the BGA package are readjusted to fit within the design boundaries. The editor displays a message at the bottom of the dialog box to indicate the state of the relationship between the design size and the BGA package size.• <i>Extents</i>: Enter values for the bottom left X and Y coordinates, and the top right X and Y coordinates.
<i>Flip placed symbol</i>	Used if the BGA pin information is pins down/up and needs to be pins up/down. The data is flipped along the Y axis of the BGA symbol, and mirrored.
<i>Reuse device file</i>	Indicates that you use the existing device file in the current working directory. If a device file is not present, a new one is created based on the data being read in.
<i>Back</i>	Click to return to the previous dialog box.
<i>Next</i>	Click <i>Next</i> to proceed to the Final Confirmation dialog box.
<i>Cancel</i>	Ignores your input and closes the dialog box.

BGA Text-In Wizard, Step 5: Final Confirmation Dialog Box

You can use the last screen in the wizard to make changes to the settings you selected in previous screens, cancel the operation without saving, or finish the wizard process.

<i>Run purged unused nets on exit</i>	Purging unused nets lets you remove some or all unused nets left in your design database when you remove or replace design objects or import objects whose names are identical to objects already in your drawing. These nets are not associated with any pins, shapes, or other design objects other than properties, but appear in lists of nets or net reports. This feature is on by default.
<i>Run derive assignment on exit</i>	Lets you check your display for unconnected shapes and incomplete netlists and automatically assign the connections from the existing conductor pattern. This feature is on by default.

Importing BGA Pin Data

 If you are importing BGA information from a spreadsheet, convert the data from your spreadsheet program to ASCII text format. *BGA Text-In Wizard* processes ASCII text files only.


 The BGA Text-In Wizard can also import design information that was previously exported using the BGA Text Out Wizard. For details on that process, see [bga text out.](#))

Perform the following steps to import BGA pin data:

1. Run the `bga text in` command.
2. In the BGA Text-In Wizard, choose that ASCII text file from the file browser.

 Do not enable *Change Directory*.

3. Complete the BGA Text-In Wizard — Delimiters dialog box.
After you choose the delimiters, the BGA Text-In Wizard displays the pin information in discrete columns of information.
4. Complete the BGA Text-In Wizard — Pin Information dialog box.
5. If padstacks are not yet created, complete the BGA Text Wizard — Padstack Information dialog box.

 This dialog box does not appear if the symbol's padstacks have already been defined.

6. Complete the BGA Text-In Wizard — Package Information dialog box.
7. Click *Next* to display the Final Confirmation dialog box.
8. Depending on the state of the BGA creation, click *Finish* to create the BGA component, *Back* to make changes to your settings, or *Cancel* to terminate the wizard without saving the created BGA.

Related Topics:

- [bga text in](#)

bga text out

The `bga text out` command creates a text file of BGA data. Exporting BGA data to a text file provides the following advantages:

- BGA designs can be reused to create new packages from existing designs.
To import BGA data, see [bga text in](#).
- The format is organized in columns of data that can be used by spreadsheet software, for customizing or generating a variety of reports.
- Sorting of data on different criteria lets you organize information the way you want it.

The BGA Wizard presents a series of dialog boxes to guide you through the process of exporting BGA data when you run the program.

Related Topics:

- [Exporting BGA Pin Data](#)

BGA Text Out Dialog Box

Access Using

- *Menu Path: File – Export – BGA Text-Out Wizard*

Export BGA Text-Out Wizard Dialog Box

<i>RefDes</i>	Lets you choose the reference designator of the component you want to export. If your design contains only one valid component, this dialog box is not displayed.
---------------	-------------------------------------------------------------------------------------------------------------------------------------------------------------------

File Selection Dialog Box

A standard browser that lets you choose a file for storing data.

Export BGA Text-Out Wizard Header Information Dialog Box

Lets you choose the headers you want to include in the exported data file by clicking on associated buttons. Header data is automatically displayed from the design data. By default, all headers are included. Two new check boxes on the first page allow you to control whether the pad definitions are exported to the file and to control if the grids are defined. Grids default to on, while padstacks default to off.

Export BGA Text-Out Wizard Pin Information Dialog Box

*Column
header
buttons*

Above each column, a heading displays the type of information in the column. To change the order of columns or to assign a blank button an information type, right-click on a column header and choose the type of data you want displayed in that column. Available header types are:

- *Remove*: Removes a column from the Export BGA output. Once you remove a column, you can get it back only by canceling the operation and starting it over.
- *Rotation*: Specifies the rotation value in degrees of each pin in the BGA.
- *Ref Des*: Specifies the reference designator of the BGA whose pins are listed in the corresponding Package Pin column.
- *Padstack*: Specifies the padstack type of each pin.
- *Pin Number*: Specifies the pin numbers.
- *Pin Name*: Specifies the logical pin name that is different from the physical pin number.
- *Package Pin*: Specifies the logical connection for each pin in the symbol to a corresponding package pin.
- *Net Name*: Specifies the net names that are assigned to each pin in the BGA. If the net does not already exist for a pin, create it in this column.
- *Mixed-Case Net Name*: Allows you to import and export mixed-case names of the nets, for example, from LEF/DEF or OpenAccess.
- *Die Pin Name*: Specifies the logical name of the pin. Use this instead of Pin Number, as physical pin numbers tend to change.
- *Net Prop Name*: Specifies the property names of the nets. Each name should be specified with the *Net Property Value* data type.
- *Net Prop Value*: Specifies the property values of the nets. Each value should be specified with the *Net Property Name* data type.
- *Pin Prop Name*: Specifies the property names of the pins. Each name should be specified with the *Pin Property Value* data type.

	<ul style="list-style-type: none">• <i>Pin Prop Value</i>: The property names of the pins. Each name should be specified with the <i>Pin Property Value</i> data type. <i>X</i>: The x coordinate of each pin in the BGA. <i>Y</i>: The y coordinate of each pin in the BGA.• <i>Swap code</i>: Preserves pin-swapping information.• <i>Include Column Headers</i>: Specifies whether the column headers are included in the output. (This does not affect the file headers specified on the previous dialog box.)
<i>Duplicate Pin Information</i>	Lets you display or hide repetitive information. For example, there may be a single net with many package pins assigned to it. The net name in this case can be displayed in just the first occurrence or in each subsequent occurrence.
<i>Mirror coordination in y-axis</i>	Click to display mirrored coordinate values for die pins.
<i>Preview</i>	Click to display the output before it is written to a file
<i>Sort</i>	Click to sort the package information by up to three criteria in ascending or descending order.

Exporting BGA Pin Data

Follow these steps to export BGA pin data:

1. In the Export BGA Wizard dialog box, choose the reference designator of the component you want to export, then click *OK*. If your design contains only one valid component, this dialog box is not displayed.
A standard file browser is displayed.
2. Name the file in which the data is to be stored, then click *Save*.
The Export BGA Wizard Header Info dialog box appears
3. Choose the headers that you want included in the exported data file, then click *Next*.
The Export BGA Wizard Pin Info dialog box appears.
4. Specify pin information according to the description in Export BGA Text-Out Wizard Pin Information Dialog Box.

Click when the columns are organized the way you want to write it to a file.

Related Topics:

- [bga text out](#)

blank_waived_drcs

blank waived drcs

The `blank waived drcs` command lets you suppress waived DRC error markers from displaying on the board. This command is the opposite of the `show waived drcs` command.

For more information on waiving DRCs, see [waive drc](#), [show waived drcs](#), and [restore waived drc](#), and for information about waiving design rule check errors, see *Creating Design Rules* in the user guide.

Access Using

- *Menu Path: Display – Waive DRCs – Blank*

Concealing Waived DRC Error Markers in the Design

To hide waived DRC error markers from your design:

1. Run the `blank waived drcs` command.

The waived DRC error markers disappear from the board.

bmpflush

An internal Cadence engineering command.

bmscheck

An internal Cadence engineering command.

board keepout

The `board keepout` command displays the Keepout dialog box, where you define keepout areas to isolate sections within the board outline where component placement is not allowed. You can create, modify, or delete keepout areas.

This command allows you to define areas of the board without having to use one of the add shape commands.

Related Topics:

- [Opening the Keepout Dialog Box](#)
- [Creating a Keepout](#)
- [Editing a Keepout Area](#)
- [Creating a New Segment within an Existing Segment](#)
- [Moving a Keepout](#)
- [Deleting a Keepout](#)

Keepout Dialog Box

Access Using

- *Menu Path: Setup – Outlines – Keepout*

Use this dialog box for defining, modifying, moving, and deleting areas within the board outline where component placement is not allowed.

<i>Command Operations</i>	Indicates what action you want to perform. The default is <i>Create</i> .
<i>Side of Board</i>	Indicates where you want to create a new keepout or where an existing keepout you to edit, move, or delete is located. The default is <i>Top</i> .
<i>Create Options</i>	Specifies the way to create a new keepout. The default is <i>Draw Rectangle</i> .
<i>Draw Rectangle</i>	Enables you to freely create and size a rectangle.
<i>Place Rectangle</i>	Enables you to create a rectangle according to dimensions you specify. When selected, the <i>Wdt</i> and <i>Hgt</i> fields appear to accept your width and height dimensions.
<i>Draw Polygon</i>	Enables you to freely create and size a polygon.

Related Topics:

- [Creating a Keepout](#)
- [Editing a Keepout Area](#)
- [Creating a New Segment within an Existing Segment](#)
- [Moving a Keepout](#)
- [Deleting a Keepout](#)

Opening the Keepout Dialog Box

Perform the following steps to open the keepout dialog box:

1. Run the `board keepout` command.
2. In the Keepout dialog box, choose the task you want to perform from the *Command Operations* section.
3. In the *Side of Board* section, choose the location of the new or existing keepout.
4. Continue with Step 2 for the task you are performing.

Related Topics:

- [board keepout](#)
- [Editing a Keepout Area](#)
- [Creating a New Segment within an Existing Segment](#)
- [Moving a Keepout](#)
- [Deleting a Keepout](#)

Creating a Keepout

1. Follow the instructions in [Opening the Keepout Dialog Box](#).
2. In the *Create Options* section of the Keepout dialog box, choose the type of keepout you want to create:

Create Option	Steps
<i>Draw Rectangle</i>	<ol style="list-style-type: none">a. Click one set of coordinates in the design.b. Click a different set of coordinates. A rectangular keepout is created.
<i>Place Rectangle</i>	<ol style="list-style-type: none">a. Enter the values you need in the <i>Wdt</i> (width) and <i>Hgt</i> (height) fields. If the units you enter are other than mil, the value in mil is calculated and substituted.b. Move the cursor to the design. The bottom left corner of an outline of the keepout attaches itself to the cursor.c. Click a coordinate in the design. A rectangular keepout with a fixed height and width is created.
<i>Draw Polygon</i>	<ol style="list-style-type: none">a. Click three or more coordinates in the design.b. To close the polygon, do one of the following: Click the original starting point. This point is easier to observe when you zoom in on the design. –or– Click <i>OK</i> in the Keepout dialog box. –or– Click <i>Edit</i>, <i>Move</i>, or <i>Delete</i> in the Keepout dialog box.

Related Topics:

- [board keepout](#)
- [Keepout Dialog Box](#)
- [Creating a New Segment within an Existing Segment](#)
- [Moving a Keepout](#)
- [Deleting a Keepout](#)

Editing a Keepout Area

To edit a keepout area, follow these steps:

1. Follow the instructions in [Opening the Keepout Dialog Box](#).
2. Click a keepout in the design. The keepout is highlighted, handles (squares) appear on the corners, and mid-points of every line segment appear.
3. Click any handle on the keepout. The handle attaches itself to the cursor.
4. Drag the handle to the target coordinates. Continuous line segments are automatically merged.

Related Topics:

- [board keepout](#)
- [Keepout Dialog Box](#)
- [Opening the Keepout Dialog Box](#)
- [Moving a Keepout](#)
- [Deleting a Keepout](#)

Moving a Keepout

To move a keepout, perform the following steps:

1. Follow the instructions in [Opening the Keepout Dialog Box](#).
2. Click a keepout in the design. An outline of the keepout attaches to the cursor.
3. Click the target coordinates. The keepout moves to the new location.

Related Topics:

- [board keepout](#)
- [Keepout Dialog Box](#)
- [Opening the Keepout Dialog Box](#)
- [Creating a Keepout](#)
- [Editing a Keepout Area](#)

Deleting a Keepout

To delete a keepout from your layout, follow these steps:

1. Follow the instructions in [Opening the Keepout Dialog Box](#).
2. Click a keepout in the design. The keepout is highlighted.
3. Click *Yes* when asked to confirm the deletion. The keepout is deleted.

Related Topics:

- [board keepout](#)
- [Keepout Dialog Box](#)
- [Opening the Keepout Dialog Box](#)
- [Creating a Keepout](#)
- [Editing a Keepout Area](#)
- [Creating a New Segment within an Existing Segment](#)

board outline

Displays the Design Outline dialog box, where you create a new board outline or modify, move, or delete an existing one.

Related Topics:

- [Opening the Design Outline Dialog Box](#)
- [Creating a Board Outline](#)
- [Editing a Board Outline](#)
- [Creating a New Segment within an Existing Segment](#)
- [Moving a Board Outline](#)
- [Deleting a Board Outline](#)

Design Outline Dialog Box

Access Using

- *Menu Path: Setup – Outlines – Design Outline*

Use this dialog box to create a new board outline or modify, move, or delete an existing one. Creating a board outline automatically generates package and route keepins. Modifying or moving a board outline automatically regenerates those keepins.

<i>Command Operations</i>	Indicates what action you want to perform. When an outline exists, the default is <i>Edit</i> . Otherwise, it is <i>Create</i> .
<i>Design Edge Clearance</i>	Defines the space between the board outline and package and route keepin boundaries. This field appears only when you choose <i>Create</i> or <i>Edit</i> . You can modify the default clearance value by changing the Non-Etch grid spacing values in the <i>Define Grid</i> dialog box. Change these values and you get a different default spacing value.
<i>Create Options</i>	Specifies the way to create a new keepout. The default is <i>Draw Rectangle</i> .
<i>Draw Rectangle</i>	Enables you to freely create and size a rectangle.
<i>Place Rectangle</i>	Enables you to create a rectangle according to dimensions you specify. When selected, the <i>Wdt</i> and <i>Hgt</i> fields appear to accept your width and height dimensions.
<i>Draw Polygon</i>	Enables you to freely create and size a polygon.

Related Topics:

- [Creating a Board Outline](#)
- [Editing a Board Outline](#)
- [Creating a New Segment within an Existing Segment](#)
- [Moving a Board Outline](#)
- [Deleting a Board Outline](#)

Opening the Design Outline Dialog Box

To open the design outline dialog box, follow these steps:

1. Run the `board outline` command.
2. In the Design Outline dialog box, choose the task you want to perform from the *Command Operations* section.
3. Continue with Step 2 for the task you are performing.

Related Topics:

- [board outline](#)
- [Editing a Board Outline](#)
- [Creating a New Segment within an Existing Segment](#)
- [Moving a Board Outline](#)
- [Deleting a Board Outline](#)

Creating a Board Outline

Perform these steps to create a board outline:

1. Follow the instructions in [Opening the Design Outline Dialog Box](#).
2. In the *Create Options* section of the Keepout dialog box, choose the type of keepout you want to create:

Create Option	Steps
<i>Draw Rectangle</i>	<ol style="list-style-type: none">a. Click one set of coordinates in the design.b. Click a different set of coordinates. A rectangular outline is created.
<i>Place Rectangle</i>	<ol style="list-style-type: none">a. Enter the values you need in the <i>Width</i> and <i>Height</i> fields. If the units you enter are other than mil, the value in mil is calculated and substituted.b. Move the cursor to the design window. The bottom left corner of the outline attaches itself to the cursor.c. Click a coordinate in the design. A rectangular outline with a fixed height and width is created.
<i>Draw Polygon</i>	<ol style="list-style-type: none">a. Click three or more coordinates in the design.b. To close the polygon, do one of the following: Click the original starting point. This point is easier to observe when you zoom in on the design. –or– Click <i>OK</i> in the Outline dialog box. –or– Click <i>Edit</i>, <i>Move</i>, or <i>Delete</i> in the Outline dialog box.

Related Topics:

- [board outline](#)
- [Design Outline Dialog Box](#)
- [Creating a New Segment within an Existing Segment](#)
- [Moving a Board Outline](#)
- [Deleting a Board Outline](#)

Editing a Board Outline

Follow the instructions in [Opening the Design Outline Dialog Box](#).

In the design, the board outline is highlighted and handles (squares) appear on the corners and midpoints of every line segment.

1. Click any handle on the board outline. The handle attaches to the cursor.
2. Drag the handle to the target coordinates. Continuous line segments are automatically merged.
3. Enter the value you need in the *Board Edge Clearance* field. If the units are other than mil, the value in mil is calculated and substituted.

Creating a New Segment within an Existing Segment

Follow these steps to create new segments within existing segments in your design:

1. Follow the instructions in [Opening the Keepout Dialog Box](#).
2. Click two points on the existing segment. The new segment attaches itself to the cursor.
3. Drag the new segment to the target coordinates.

Related Topics:

- [board keepout](#)
- [Keepout Dialog Box](#)
- [Opening the Keepout Dialog Box](#)
- [Creating a Keepout](#)
- [Deleting a Keepout](#)

Moving a Board Outline

To move a board outline, follow these steps:

1. Follow the instructions in [Opening the Design Outline Dialog Box](#).

In the design, an outline of the board attaches to the cursor at the lower left corner.

1. Click the target coordinates. The board moves to the new location.

Related Topics:

- [board outline](#)
- [Design Outline Dialog Box](#)
- [Opening the Design Outline Dialog Box](#)
- [Creating a Board Outline](#)
- [Editing a Board Outline](#)

Deleting a Board Outline

Perform these steps to delete a board outline:

1. Follow the instructions in [Opening the Design Outline Dialog Box](#).
2. Click Yes when asked to confirm the deletion. The board outline, package keepin, and route keepin are deleted.

Related Topics:

- [board outline](#)
- [Design Outline Dialog Box](#)
- [Opening the Design Outline Dialog Box](#)
- [Creating a Board Outline](#)
- [Editing a Board Outline](#)
- [Creating a New Segment within an Existing Segment](#)

boardoutline import

The `boardoutline import` command displays the Import Board File Browser dialog box. The file browser enables easy selection of source boards for use in a current design. Once a board is selected, the Import Board dialog box is displayed. This dialog box allows the selective reuse of existing board design data. Source board parameters such as electrical rule constraints, rooms, stack-up, and so on, can be selectively accessed as a basis for a new design. The directory path and file name appear at the top of the form.

Related Topics:

- [Choosing a Source Board](#)
- [Importing Board Geometry Information](#)
- [Importing Electrical Rules](#)
- [Importing Placed I/O Components](#)
- [Importing Rooms](#)

Import Board File Dialog Box

Access Using

- *Menu Path: File – Import – Board*

Import Board Dialog Box

Use this dialog box to selectively access source board parameters such as electrical rule constraints, rooms, and stack-up as a basis for a design. The file path for the board is displayed at the top of the dialog box.

Board Geometry Tab	
<i>Board Cross Section</i>	Overwrites any existing cross section data.
<i>Board Outline</i>	Imports the board outline and package/route keepin information.
<i>Keepouts</i>	Imports board keepout information.
Electrical Rules Tab	
<i>Exclude</i>	Displays source board rules (rules that are not imported).
<i>Include</i>	Displays new board rules (rules that are imported).
<i>Move All</i>	Moves all the rules to the opposite box.
Placed I/O Components Tab	
<i>Exclude</i>	Displays the placed connector components that are not imported.
<i>Include</i>	Displays the placed connector components that are imported.
<i>Move All</i>	Moves all the placed components to the opposite box.
Rooms Tab	
<i>Exclude</i>	Displays the rooms that are not imported.
<i>Include</i>	Displays the rooms that are imported.
<i>Move All</i>	Moves all the rooms to the opposite box.

Conflicts Dialog Box

Use this dialog box to make choices when conflicts between existing data and imported data occur while importing a board.

<i>Write over data</i>	Writes over existing conflicting data.
<i>Reject conflicting data</i>	Leaves existing conflicting data intact and discards imported data.
<i>Query for each item</i>	Presents a confirm window for each conflicting item, presenting a choice of keeping or deleting existing data.

Related Topics:

- [Importing Board Geometry Information](#)
- [Importing Electrical Rules](#)
- [Importing Placed I/O Components](#)
- [Importing Rooms](#)

Conflicts Dialog Box

Use this dialog box to make choices when conflicts between existing data and imported data occur while importing a board.

<i>Write over data</i>	Writes over existing conflicting data.
<i>Reject conflicting data</i>	Leaves existing conflicting data intact and discards imported data.
<i>Query for each item</i>	Presents a confirm window for each conflicting item, presenting a choice of keeping or deleting existing data.

Choosing a Source Board

Follow these steps to choose a source board:

1. Run `boardoutline import`.
The BoardOutline Import file browser dialog box appears.
2. Choose a `.brd` name in the File Name list box and click *OK*.
The Import Board dialog box appears.

Related Topics:

- [boardoutline import](#)
- [Importing Electrical Rules](#)
- [Importing Placed I/O Components](#)
- [Importing Rooms](#)

Importing Board Geometry Information

Follow these steps to import board geometry information:

1. Click the Board Geometry tab.
2. Choose *Board Cross Section* to import cross section data from the source board.
3. Choose *Board Outline* to import the board outline and package/route keepin information from the source board.
4. Choose *Keepouts* to import board keepout information from the source board.

Related Topics:

- [boardoutline import](#)
- [Import Board File Dialog Box](#)
- [Importing Placed I/O Components](#)
- [Importing Rooms](#)

Importing Electrical Rules

Source board rules (rules that are not imported) appear in the Exclude list box. New board rules (rules that are imported) appear in the Include list box.

1. Click a rule in the Exclude list box to move it to the Include list box.
2. Click a rule in the Include list box to move it to the Exclude list box.
3. Click ALL in either direction to move all the rules to the opposite box.

Related Topics:

- [boardoutline import](#)
- [Import Board File Dialog Box](#)
- [Choosing a Source Board](#)
- [Importing Rooms](#)

Importing Placed I/O Components

Perform these steps to import placed I/O components:

1. Choose the placed input/output components to import from the source board.
2. Click a component in the Exclude list box to move it to the Include list box.
3. Click a component in the Include list box to move it to the Exclude list box.
4. Click ALL in either direction to move all the components to the opposite box.

Related Topics:

- [boardoutline import](#)
- [Import Board File Dialog Box](#)
- [Choosing a Source Board](#)
- [Importing Board Geometry Information](#)

Importing Rooms

To import rooms to your layout, perform the following steps:

1. Choose the rooms to import from the source board.
2. Click a room in the Exclude list box to move it to the Include list box.
3. Click a room in the Include list box to move it to the Exclude list box.
4. Click ALL in either direction to move all the rooms to the opposite box.

If conflicts exist between the current board and the imported information, the [Conflicts Dialog Box](#) appears.

Related Topics:

- [boardoutline import](#)
- [Import Board File Dialog Box](#)
- [Choosing a Source Board](#)
- [Importing Board Geometry Information](#)
- [Importing Electrical Rules](#)

board plane

The `board plane` command displays the Plane Outline dialog box for creating a new plane outline or modifying, moving, or deleting an existing outline.

Plane Outline Dialog Box

Access Using

- *Menu Path: Setup - Outlines - Plane Outline*

Use this dialog box to create a plane outline. You can also edit move or delete an existing plane outline.

Command Operations

<i>Create</i>	Enables you to create a new plane outline. Enter the shape data, then choose an option in the Create Options area of the dialog box.
<i>Edit</i>	Enables you to edit an existing plane outline. Choose the plane layer, then click the edit handles provided on the outline (in the Floorplanner view) to edit the shape.
<i>Move</i>	Enables you to move an existing plane outline. Click anywhere on the plane (in the Floorplanner view) to attach it to your cursor and move it relative to its lower left corner.
<i>Delete</i>	Deletes a plane outline.

Shape Data

<i>Layer</i>	Specifies a target layer for the plane.
<i>Net</i>	Assigns a net to the plane. Type in a net name or click <i>Browse</i> to choose a net from a list.
<i>Browse</i>	Displays the Select a Net Browser.
<i>Voltage</i>	Specifies a plane voltage level.

Create Options

<i>Draw Rectangle</i>	Enables you to freely create and size a rectangle.
<i>Place Rectangle</i>	Enables you to create a rectangle according to dimensions you specify. When selected, two type-in fields appear to accept your dimensions in mils.
<i>Draw Polygon</i>	Enables you to freely create and size a polygon.
<i>Copy From Plane</i>	Enables you to copy and use an outline from an existing plane. After choosing this option, click the down-arrow to display a list of planes to choose from.
<i>Copy Board Outline</i>	Enables you to copy and use the board outline. Choose a plane layer, then click <i>Apply</i> or <i>OK</i> .

bond wire length

The `bond wire length` command displays the Bonding Wire Length Report.

bond wire location

The `bond wire location` command displays the Bonding Wire Location Report.

bondwire text in

The `bondwire text in` command brings up the Bond Wire Text-In Wizard where you can:

- Define wire bond connections in the design by importing an ASCII spreadsheet of bond wire information.
- Update the mapping of pins to fingers.

Related Topics:

- [Importing Bond Wire Data](#)

Bond Wire Text In Dialog Box

Access Using

- *Menu Path: Route – Wire Bond – Bond Wire Text Import*

Bond Wire Text-In Wizard, Step 1: File Selection Dialog Box

A standard file browser that allows you to select the text file with bond wire information.

Wire Bond Text-In Wizard, Step 2: File Information Dialog Box

Coordinates	Specifies the unit-type of measurement available in the drop-down list.
Delimiters	
<i>Tab</i>	Choose to use a tab to separate columns of data.
<i>Semicolon</i>	Choose to use a semicolon to separate columns of data.
<i>Comma</i>	Choose to use a comma to separate columns of data.
<i>Space</i>	Choose to use a space to separate columns of data.
<i>Other</i>	Choose to use other characters to separate columns of data.
<i>Ignore consecutive delimiters</i>	Choose to treat consecutive delimiters as one delimiter.
<i>Remove trailing delimiters</i>	Choose to remove trailing delimiters from the data.
<i>Back</i>	Click to return to the previous dialog box.
<i>Next</i>	Click to display the next dialog box.
<i>Cancel</i>	Ignores your input and closes the dialog box.

Bond Wire Text-In Wizard, Step 3: Pin Information Dialog Box

Information contained within this dialog box includes the saved parameters for the wires and optionally, fingers. The columns in which this information appears depends on the delimiter types (tabs, semicolons, and so on) you selected in the File Information dialog box. Editing grid parameters is *not* recommended.

<i>Ignore Rows</i>	Data in this column is not imported.
<i>Ignore</i>	Denotes that this line should be ignored (comment line).
<i>Start X</i>	Specifies the X coordinate of the wire start location. This is a required field.
<i>Start Y</i>	Specifies the Y coordinate of the wire start location. This is a required field.
<i>Start Layer</i>	Specifies the name of the layer from where the wire starts. This is a required field.
<i>End X</i>	Specifies the X coordinate of the wire end location. This is a required field.
<i>End Y</i>	Specifies the Y coordinate of the wire end location. This is a required field.
<i>End Layer</i>	Specifies the name of the layer where the wire ends. This is a required field.
<i>Profile</i>	Specifies the wire profile. If not specified, the default profile for the design is used.
<i>Finger X Coord</i>	Specifies the X coordinate of the finger. Required if finger needs to be created.
<i>Finger Y Coord</i>	Specifies the Y coordinate of the finger. Required if finger needs to be created.
<i>Finger Rotation</i>	Specifies finger rotation. Required if finger needs to be created.
<i>Finger Padstack</i>	Specifies the finger padstack. Required if finger needs to be created.
<i>Back</i>	Click to return to the previous dialog box.
<i>Next</i>	Click to display the next dialog box.
<i>Cancel</i>	Ignores your input and closes the dialog box.

Bond Wire Text-In Wizard, Step 3A: New Padstack Information Dialog Box

Use these options to specify the padstack definitions. If the pads are already defined in the file, the Step 3A screen does not appear.


Method	
<i>New</i>	Click this button to define a new padstack. Fill in all the specification settings at the bottom of the dialog box.
<i>Available Padstack</i>	Click this button to choose a padstack from the design's database. This option is available only if a valid padstack exists in the database. When you choose the padstack from the adjacent list box, the <i>Specifications</i> boxes reflect the padstack information. You cannot edit the padstack specifications.
<i>Load from Disk</i>	Click this button to import an external padstack definition. Use the <i>Browse</i> button to locate the padstack on your disk. When you import the padstack, the <i>Specifications</i> boxes reflect the padstack information. You cannot edit the padstack specifications.
Specifications	
<i>Name</i>	Specifies the padstack name. If you are defining a new padstack, enter the name in this box
<i>Shape</i>	Specifies the padstack shape. Choose either the <i>Circle</i> or a <i>Rectangle</i> button.
<i>Dimensions</i>	Indicates the dimensions of the padstack. Enter values for the <i>Width</i> and <i>Height</i> .
<i>Back</i>	Click to return to the previous dialog box.
<i>Next</i>	Click to display the next dialog box.
<i>Cancel</i>	Ignores your input and closes the dialog box.

Bond Wire Text-In Wizard, Step 4: Final Confirmation Dialog Box

You can use the last screen in the wizard to make changes to the settings you selected in previous screens, cancel the operation without saving, or finish the wizard process.

<i>Run purged unused nets on exit</i>	Purging unused nets lets you remove some or all unused nets left in your design database when you remove or replace design objects or import objects whose names are identical to objects already in your drawing. These nets are not associated with any pins, shapes, or other design objects other than properties, but appear in lists of nets or net reports. This feature is on by default.
<i>Run derive assignment on exit</i>	Lets you check your display for unconnected shapes and incomplete netlists and automatically assign the connections from the existing conductor pattern. This feature is on by default.

Importing Bond Wire Data

 If you are importing bond wire information from a spreadsheet, convert the data from your spreadsheet program to ASCII text format. *Bond Wire Text-In Wizard* processes ASCII text files only.

To import bond wire data, follow these steps:

1. Run the `bondwire text in` command.
2. In the Bond Wire Text-In Wizard, choose that ASCII text file from the file browser.

 Do not enable *Change Directory*.

3. Complete the Bond Wire Text-In Wizard — Delimiters dialog box. For details, see [Wire Bond Text-In Wizard, Step 2: File Information Dialog Box](#).
After you choose the delimiters, the Bond Wire Text-In Wizard displays the wire and optionally, finger, information in discrete columns.
4. Complete the Bond Wire Text-In Wizard — Pin Information dialog box. For details, see [Bond Wire Text-In Wizard, Step 3: Pin Information Dialog Box](#).
5. Click *Next* to display the Final Confirmation dialog box.
6. Depending on the state of the Bond Wire creation, click *Finish* to create the bond wires, and, optional fingers, *Back* to make changes to your settings, or *Cancel* to terminate the wizard without saving the created wires or fingers.

Related Topics:

- [bondwire text in](#)

bpa

The `bpa` command attaches a unique string identifier to each bondpad in your design. The strings must take the form of an alphanumeric ID that ends with an integer, such as BF-1, BF-2, and so forth. With this command, you can document and communicate connectivity from die pin to finger to package I/O. You can also change existing bondpads when you make design changes.

This command works only on the BOND_PAD property, which is automatically generated when you add wire bonds to your design with the [wirebond select](#) command.

Related Topics:

- [Adding Text to BondPads](#)

BPA Command: Options Panel

Access Using

- Menu Path: *Manufacture – Documentation – Bond Finger Text*

<i>Remove existing finger labels</i>	Check this box to disable the label configuration fields. Select the necessary fingers and the BOND_PAD property value becomes an empty string.
<i>Allow BondPad Re-Assignment</i>	<p>Lets you replace the text value of the BOND_PAD property. If not selected, the value of the property is not affected. When you choose a bondpad with this option turned off, a message similar to the following appears in the console window:</p> <pre>Unchanged BONDPAD = BF13 at (location coordinates)</pre> <p>When you choose a bondpad with this option turned on, a message similar to the following appears in the console window: Changed BONDPAD from BF13 to BF14 at (location coordinates)</p>
<i>Single label for merged fingers</i>	Check this box to assign a single finger name to all fingers that are covered by a merge finger shape.
<i>Beginning Ref#</i>	Enter an alphanumeric value that ends in an integer—for example, A3 or xyz/16. The default is <i>BF1</i> .
<i>Increment By</i>	Enter an integer. The default is <i>1</i> .
<i>Sorting</i>	
<i>Sort by bondpad location</i>	The default sorting option, this control sorts the bondpad ID by comparing its position to other bondpads in the design.
<i>Sort by die pin location</i>	This option uses the first bondwire it locates to find its associated die pin, then sorts IDs based on the locations of die pins relative to each other. If a die pin for a particular item is not found, the associated bondpad's location is used.
<i>Horizontal Vertical</i>	Choose a sorting option for each field. This option is useful in accounting for slight misalignments of bondpads in your design. The defaults are <i>Left to Right</i> and <i>Top to Bottom</i> .

Adding Text to BondPads

Follow these steps to add text to bondpads:

1. Run the `bpa` command.

The Options panel displays the bondpad options and the Find panel is set to *Vias*. (Bondpads are represented as vias in the tool.)

1. Complete the Options panel. For details, see [BPA Command: Options Panel](#).
2. Choose the bondpads to reassign. You can do this individually or by choosing one of the pop-up menu selections (Temp Group or Window Select).
3. Choose Done (or Complete and Done if selecting by Temp Group). The new text is assigned to the BOND_PAD properties.
4. Verify the outcome of your action using the [show element](#) command.

Related Topics:

- [bpa](#)

brd export

Database conversion between a .brd design and an .mcm design is not supported in this release. If you have questions, contact Customer Support.

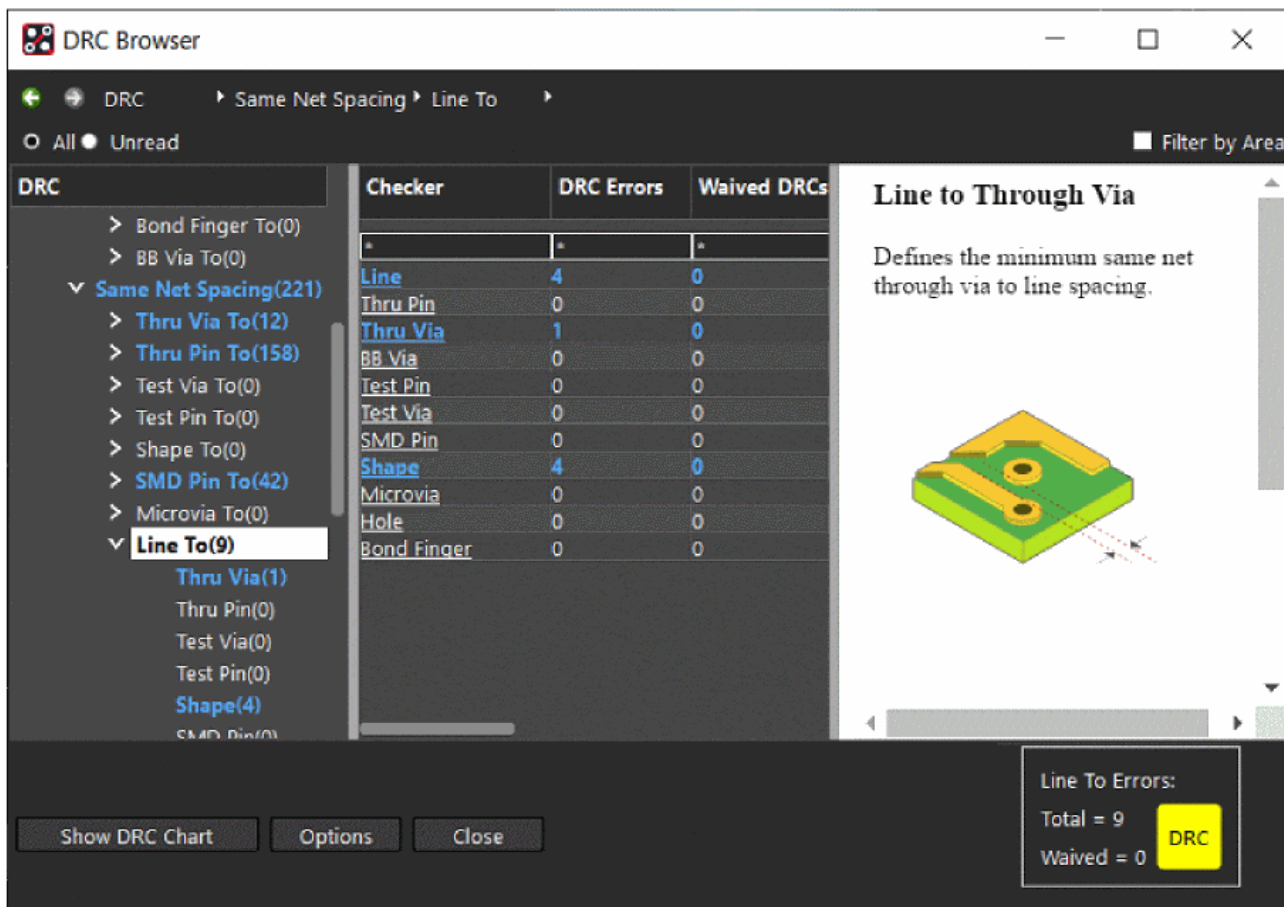
brd import

Database conversion between a .brd design and an .mcm design is not supported in this release. If you have questions, contact Customer Support.

browse drcs

The `browse drcs` command opens DRC Browser, an advanced utility that helps PCB designers to locate, review and address DRC errors in a design. The DRC Browser displays all DRCs created by the DRC system including external DRCs. The browser provides real-time feedback on the type and number of errors and dynamically updates this data as issues are corrected or introduced while editing the design.

For each DRC type, the browser window displays description, image, DRC code, and other details. The DRC Browser UI contains various navigation, sorting, and filtering capabilities that helps to focus on resolving design issues by DRC violation types and areas.



All errors are organized in the navigation list and the state of each DRC is represented in pre-defined color scheme. The navigation list displays DRC location, value, subclass, and so on in a spreadsheet format and supports all basic operation of spreadsheet, such as column resizing, column reordering,

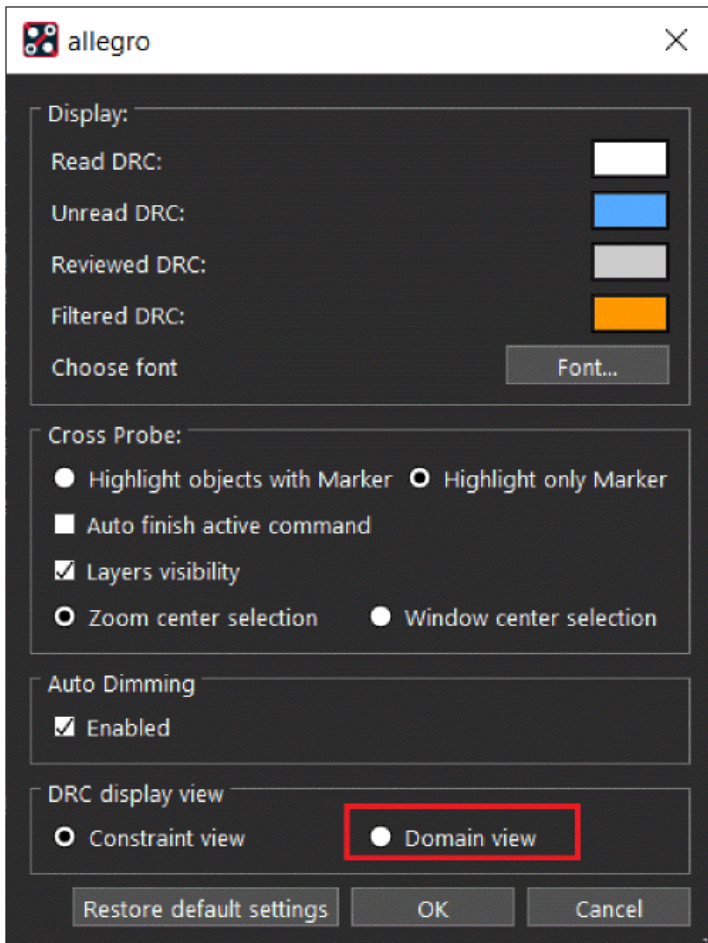
sorting, filtering, zoom in and out using mouse controls. Selecting a DRC row locates the error in the canvas and highlights its DRC marker keeping the rest of the design objects in dim mode. The status of the error changes from *Unread* to *Read*. Using right-click options you can mark the state as *Review* for further review as a reminder. When choose to waive a DRC, you can add a note in the Waive DRC Comments window that opens.

The screenshot shows the 'DRC Browser' window with a table of DRC errors. The table has four columns: 'DRC Location', 'DRC Subclass', 'Actual Value', and 'Required Value'. A context menu is open over the table, and a 'Waive DRC Comments' dialog is also visible. A 'Shape Errors' summary box shows 4 total errors and 0 waived.

DRC Location	DRC Subclass	Actual Value	Required Value
(145759.8 171400.0)	Top	0 UM	150 UM
(145759.8 165000.0)	Top	0 UM	150 UM
(145759.8 167400.0)	Top	0 UM	150 UM
(145759.8 173800.0)	Top	0 UM	150 UM

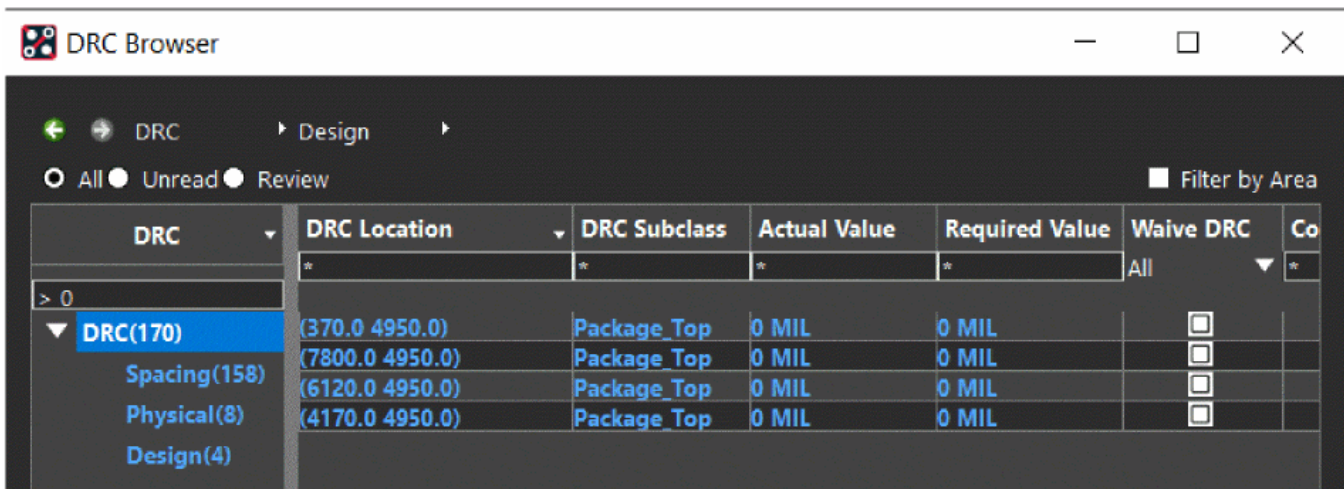
Shape Errors:
Total = 4
Waived = 0

To change the default view of displaying DRC errors, click *Options*. The window that opens provides controls to change the display of DRC, crossprobing modes, and option to restore the default settings. By default the browser zooms window to fit objects associated with DRC error. To maintain the current zoom level, enable *Window center selection* option.



By default, DRCs are displayed by constraint rule. The *Constraint view* option allows navigation through the hierarchal constraint tree, and the DRCs are displayed when navigation reaches the lowest tier of the constraints.

To view DRCs by constraint domain, set the *DRC display view* to *Domain view*. Enabling this option displays a flat list of DRCs under each constraint domain. This setting remains valid for the current session only.



DRC	DRC Location	DRC Subclass	Actual Value	Required Value	Waive DRC	Co
> 0	*	*	*	*	All	*
▼ DRC(170)	(370.0 4950.0)	Package_Top	0 MIL	0 MIL	<input type="checkbox"/>	
Spacing(158)	(7800.0 4950.0)	Package_Top	0 MIL	0 MIL	<input type="checkbox"/>	
Physical(8)	(6120.0 4950.0)	Package_Top	0 MIL	0 MIL	<input type="checkbox"/>	
Design(4)	(4170.0 4950.0)	Package_Top	0 MIL	0 MIL	<input type="checkbox"/>	

Correcting the design error automatically removes the DRC from the list in the DRC Browser window. You can also analyze the volume of DRCs through graphical representation in the form of bar and pie charts. These charts show number of DRC errors of a type in different colors for the selected domain to aid you in prioritizing the fixes based on the severity of DRC violation.



Related Topics:

- [Filtering in DRC Browser](#)
- [DRC Browser Dialog Box](#)
- [Waiving DRC by Group Select](#)

Navigating in DRC Browser

You can navigate through the DRCs in DRC Browser using the following:

- **DRC Tree:** You can expand and collapse nodes and branches of the DRC domains, down to the list of DRC locations. You can also expand or collapse the navigation tree at different nodes using right-click options *Expand All* or *Collapse All*. Each node displays the number of DRCs exist in the design.
- **Breadcrumb:** You can select an item in breadcrumb navigation path to quickly navigate to a previous location. Navigation arrows can be used to move along the most recent DRC domain path.
- **List Navigation:** You can select entries in from the list and navigate down to the list of DRC locations.
- **DRC Chart Navigation:** You can navigate through the different DRC domains by selecting a segment of the chart.

The screenshot shows the DRC Browser window with the following components:

- Breadcrumbs:** A path at the top showing the navigation sequence: DRC > Same Net Spacing > Thru Pin To > Thru Pin. A red arrow points to this path.
- DRC Tree:** A tree view on the left showing the hierarchy of DRC domains. The 'Thru Pin(156)' node is selected, and a red arrow points to it.
- DRC Locations Table:** A table with columns: DRC Location, DRC Subclass, Actual Value, and Required Value. It lists 156 DRC locations. A red arrow points to the table.
- Thru Pin Errors Summary:** A box at the bottom right showing 'Total = 156' and 'Waived = 0' with a yellow 'DRC' button.

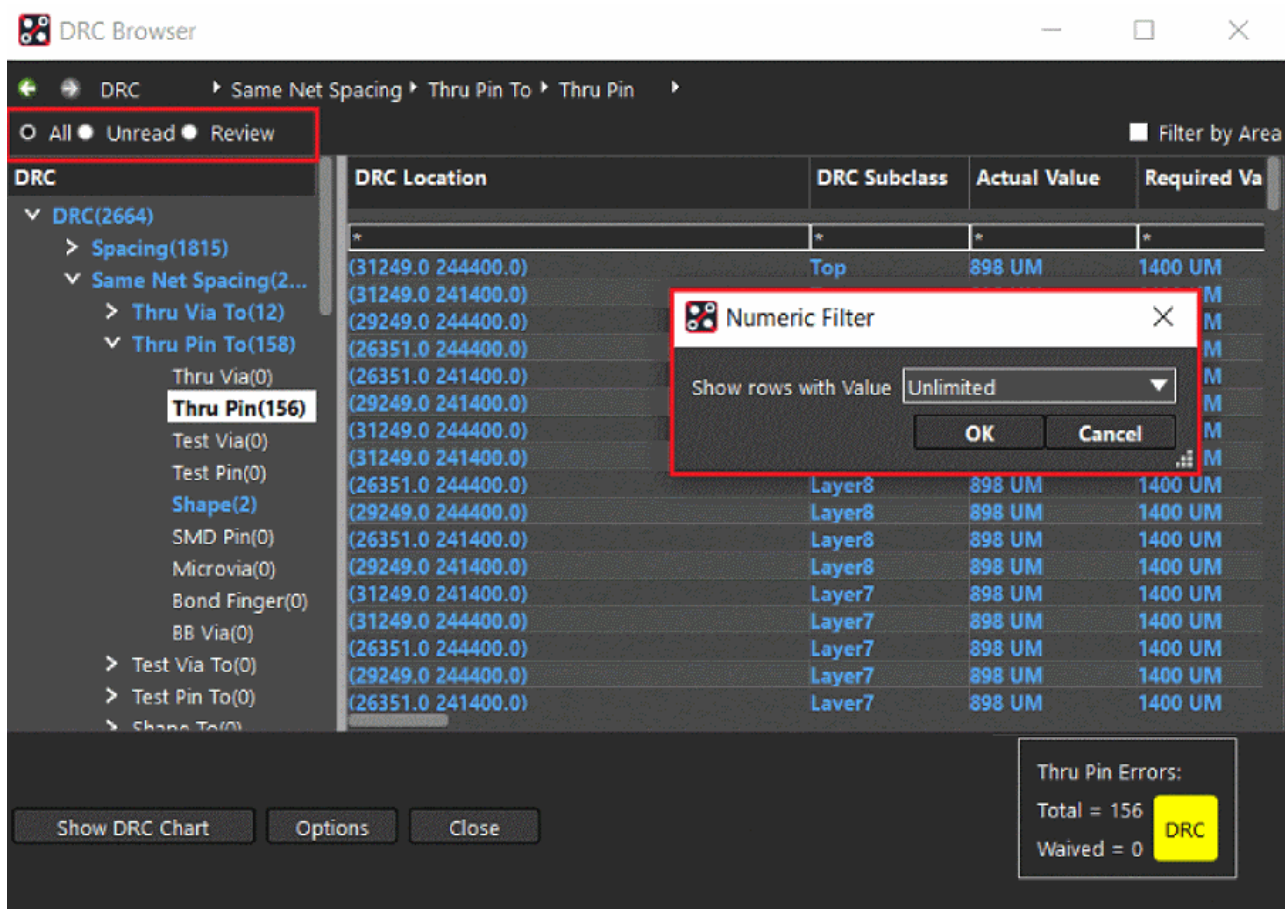
DRC Location	DRC Subclass	Actual Value	Required Value
(31249.0 241400.0)	L14-pgnd	898 UM	1400 UM
(31249.0 241400.0)	Layer19	898 UM	1400 UM
(31249.0 241400.0)	Bottom	898 UM	1400 UM
(31249.0 241400.0)	L13-p1_8V	898 UM	1400 UM
(31249.0 244400.0)	L18-p1_2V	898 UM	1400 UM
(31249.0 244400.0)	L21-p2_5V	898 UM	1400 UM
(31249.0 241400.0)	Layer11	898 UM	1400 UM
(31249.0 241400.0)	Layer15	898 UM	1400 UM
(31249.0 244400.0)	Bottom	898 UM	1400 UM
(31249.0 241400.0)	L17-0_75V	898 UM	1400 UM
(31249.0 244400.0)	Layer19	898 UM	1400 UM
(31249.0 244400.0)	L9-0_9V	898 UM	1400 UM
(31249.0 244400.0)	L6-p3_3V	898 UM	1400 UM
(31249.0 244400.0)	Layer20	898 UM	1400 UM
(31249.0 241400.0)	Layer20	898 UM	1400 UM
(31249.0 244400.0)	Layer15	898 UM	1400 UM
(31249.0 241400.0)	L25-pand	898 UM	1400 UM

Related Topics:

- [browse drcs](#)
- [DRC Browser Dialog Box](#)
- [Waiving DRC by Group Select](#)

Filtering in DRC Browser

For reviewing DRCs, the browser UI has controls to filter and sort the DRCs. You can filter the DRCs by state using radio buttons: *All*, *Unread* or *Review*. Numeric filter is also available with each column to narrow down the list of DRCs. To sort the list of DRC locations from low to high value by selecting the column header.



DRCs can also be filtered by drawing a window into the location of a selected DRCs in the canvas by enabling the *Filter by Area* checkbox. You can window select a design area and DRC Browser automatically updates the DRC tree and displays filtered DRCs for each domain that are occurred in the selected area. The *Clear Area* button clears all the selection from the design canvas and restores the default view in DRC Browser.

DRC Browser

DRC Physical Ts Allowed

All Unread Review

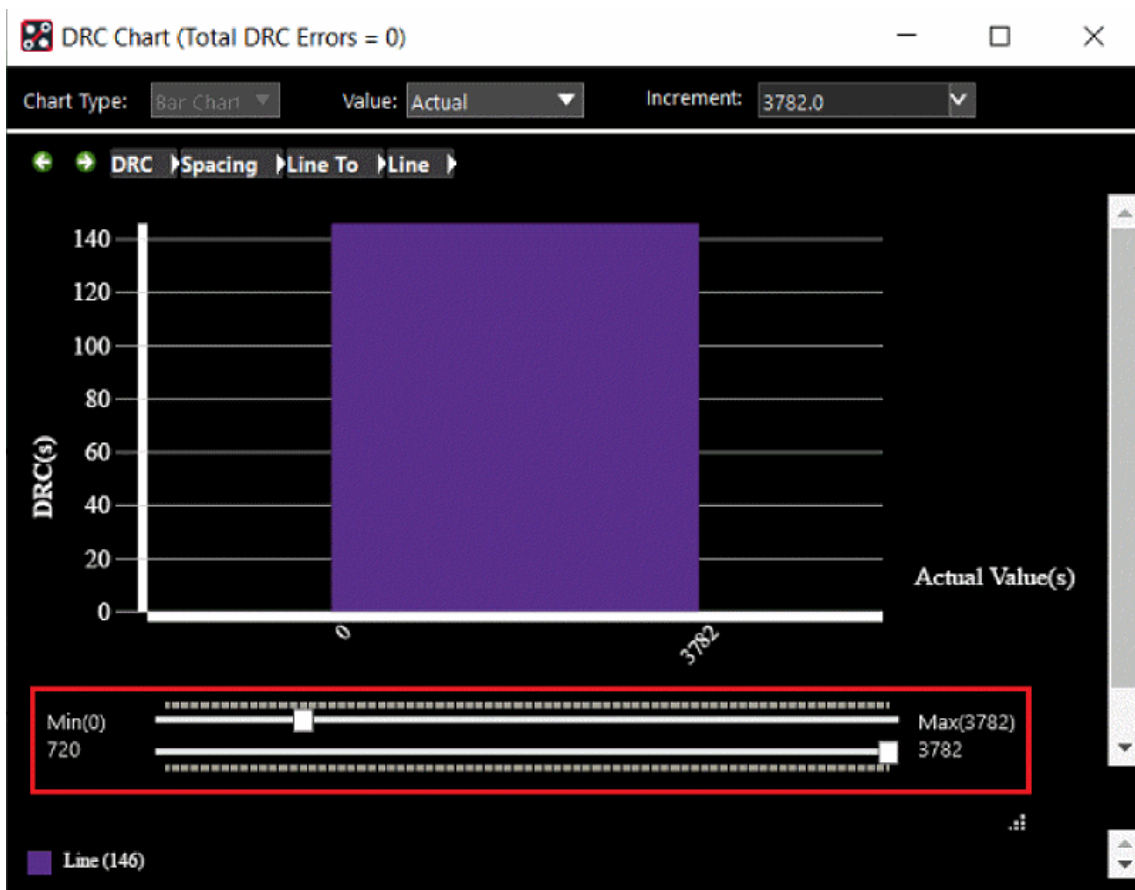
Clear Area ☒ Filter by Area

DRC	DRC Location	DRC Subclass	Actual Value	Require
DRC(137/2664)				
> Spacing(1/1815)				
> Same Net Spacing(8/221)				
> Physical(128/491)				
Vialist DRC(0/0)				
Ts Allowed(73/238)				
Pad-pad direct conne...				
Minimum Neck Wid...	(156200.0 128400.0)	Top	ANYWHERE	PINS_VI
Min neck width(0/0)	(262600.0 166400.0)	Top	ANYWHERE	PINS_VI
Min blind/buried via s...	(257800.0 166400.0)	Top	ANYWHERE	PINS_VI
Maximum Neck Len...	(310500.0 174300.0)	Top	ANYWHERE	PINS_VI
Max line width(0/0)	(310500.0 149400.0)	Top	ANYWHERE	PINS_VI
Max blind/buried via ...	(310500.0 157400.0)	Top	ANYWHERE	PINS_VI
DiffPair Minimum G...	(262275.6 146100.0)	Top	ANYWHERE	PINS_VI
External(0/0)	(310500.0 153400.0)	Top	ANYWHERE	PINS_VI
> Electrical(0/4)	(310500.0 144600.0)	Top	ANYWHERE	PINS_VI
> Design(0/123)	(262600.0 143200.0)	Top	ANYWHERE	PINS_VI
	(262275.6 122900.0)	Top	ANYWHERE	PINS_VI
	(257800.0 143200.0)	Top	ANYWHERE	PINS_VI
	(253800.0 143200.0)	Top	ANYWHERE	PINS_VI
	(253800.0 166400.0)	Top	ANYWHERE	PINS_VI
	(249800.0 143200.0)	Top	ANYWHERE	PINS_VI
	(255597.0 148000.0)	Top	ANYWHERE	PINS_VI
	(272900.0 107100.0)	Top	ANYWHERE	PINS_VI

Show DRC Chart Options Close

Ts Allowed Errors:
Total = 238
Waived = 0 **DRC**

Filtering can also be done in DRC Chart. When DRC Chart is displayed at the DRC list level, sliding bars can be used to filter the DRC list for values between maximum and minimum values for DRC list display.



Related Topics:

- [browse drcs](#)
- [Navigating in DRC Browser](#)
- [Waiving DRC by Group Select](#)

DRC Browser Dialog Box

Access Using

- *Menu Path: Tools – DRC Browser*

<i>DRC</i>	Displays bread crumb navigation to move along the most recent DRCs
<i>All</i>	Displays all the DRCs of a particular type that is selected in the DRC tree
<i>Unread</i>	Displays unread DRCs of a particular type that is selected in the DRC tree
<i>Review</i>	Displays all the DRCs marked for review of a particular type that is selected in the DRC tree
<i>Filter by Area</i>	Choose to filter DRCs by window select an area in design canvas
<i>Clear Area</i>	Choose to clear all the selections from the design canvas
<i>DRC</i>	Displays total number of DRCs at each node of the DRC tree. In
<i>Show DRC Chart</i>	Opens DRC Chart window
<i>Options</i>	Opens All Layer window to change the display options
<i>DRC Errors</i>	Displays total number of DRCs and waived DRC for the node selected in the DRC tree

Related Topics:

- [browse drcs](#)
- [Navigating in DRC Browser](#)
- [Filtering in DRC Browser](#)

Waiving DRC by Group Select

To waive a group of DRCs by selecting multiple DRCs, by either dragging the mouse if the rows are adjacent or using the `Control` key to select multiple non-adjacent rows. You can follow the same method to unwaive DRCs.

Waive DRC Using Mouse

1. Click to select the first row.
2. Drag the cursor to the last item to be waived.
All items between the first and last row are selected.
3. Hover the cursor over one of the selected rows, right-click, and choose *Waive*.
4. Enter a comment in the Waive DRC Comments pop-up window.
5. Click *OK*.
The highlight is removed from the selected rows and the Waive DRC comment you entered is assigned to all the selected rows.

Waive DRC Using Control Key

1. Press the `Ctrl` key and click one or more rows to select.
You can drag to select multiple adjacent rows.
2. Hover the cursor over one of the selected rows, right-click, and choose *Waive*.
3. Enter a comment in the Waive DRC Comments pop-up window.
4. Click *OK*.
The highlight is removed from the selected rows and the Waive DRC comment you entered is assigned to all the selected rows.

Related Topics:

- [browse drcs](#)
- [Navigating in DRC Browser](#)
- [Filtering in DRC Browser](#)
- [DRC Browser Dialog Box](#)

build_pe_script

The `build_pe_script` batch command uses individual via pattern scripts and extract command files to generate a master script which adds blind and buried via patterns on an MCM/Hybrid design. The tool also supports these patterns being defined and added as a function of automatic routing.

Syntax

```
build_pe_script [<drawing>] [<pattern_1>] [<pattern_2>]...
```

[<drawing>]	Name of the drawing to which the via patterns are to be added.	
<pattern_n>	Names of the via patterns being created. For each via pattern, the following files must be in the current directory:	
	<pattern>_cmd.txt	The extract command file that extracts the x, y location of the pads to which the pattern is to be added.
	<pattern>_pe.scr	Script that defines the Connect commands for creating the pattern.

The output of the `build_pe_script` command is a script called `drawing_pes.scr` . Executing this script adds all the defined via patterns to the drawing.

bundle blank

The `bundle blank` command hides the display of bundles associated with one or more selected design objects.

Access Using

- *Menu Path: Display – Blank Bundles – Selected*
- Right Mouse Button Option: *Blank Bundle*

Related Topics:

- [bundle blank_all](#)
- [bundle blank_unselected](#)
- [Object Selection Shortcuts](#)

Hiding Rat Bundles Associated with Selected Objects

You can hide rat bundles associated with selected objects by following these steps:

1. Select one or more objects associated with the route plan (bundle, rat, component, symbol, pin, net, c-line, c-line segment, etc.).

✔ Design density may make object selection difficult. You can limit the find criteria to just one specific object type by right-clicking in the Design window, then choosing *Super filter – <object_type>* from the menu.

For tips on multi-object selection, see the [Object Selection Shortcuts](#) table.

The selected objects highlight and also appear in the *WorldView* window.

2. With your cursor on a selected object, right-click and choose *Blank Bundle* from the menu.
The rat bundles associated with the selected objects are hidden.
3. Repeat steps 1 and 2 to hide rat bundles associated with other objects as needed.

Object Selection Shortcuts

To ...	Press and hold this key ...	and use this mouse action ...
Add individual objects to the selection set.	Shift	Left click on the object.
Add groups of objects to the selection set.	Shift	Depress the left mouse button and drag a window around the objects.
Remove individual objects from the selection set.	Ctrl	Left click on the object.
Remove groups of objects from the selection set.	Ctrl	Depress the left mouse button and drag a window around the objects.

To ...	Use this mouse action ...	press these keys and repeat
Toggle the selection of stacked objects in the design.	Hover your mouse cursor over an area where the objects overlap.	Ctrl +Tab
Toggle the selection of associated objects. For example, a net, a bundle, and a plan line of a connection.	Hover your cursor over one of the objects.	Tab

Related Topics:

- [bundle blank](#)

bundle blank_all

The `bundle blank_all` command hides the display of all rat bundles in the design.

Access Using

- *Menu Path: Display – Blank Bundles – All*

Related Topics:

- [bundle blank](#)
- [bundle blank_unselected](#)

Hiding All Rat Bundles

Perform this step to hide all rat bundles:

1. Choose *Display – Blank Bundles – All* from the menu bar.
All rat bundles are hidden.

bundle blank_unselected

The `bundle blank_unselected` command hides the display of all rat bundles in the design that are not currently selected.

Access Using

- *Menu Path: Display – Blank Bundles – Unselected*
- *Right Mouse Button Option: Blank Unselected Bundles*

Related Topics:

- [bundle blank](#)
- [bundle blank_all](#)

Hiding Unselected Rat Bundles

You can hide unselected rat bundles by performing these steps:

1. In IFP application mode, select one or more bundles to remain displayed.

✔ Design density may make bundle selection difficult. You can limit the find criteria to just bundles by right-clicking in the Design window, then choosing *Super filter – Ratbundle* from the menu.


For tips on multi-object selection, see the [Object Selection Shortcuts](#) table.
The selected rats highlight and also appear in the *WorldView* window.

2. With your cursor on a selected rat, right-click and choose *Blank Unselected Bundles* from the menu.
All bundles except for selected bundles in the design are hidden.

bundle create

The `bundle create` command lets you create a new bundle of rats from a selection of unbundled rats. Collectively, rat bundles provide guidance to the GRE route engine and influence the general flow of the interconnect solution. You can also create and manage ratsnest bundles in [Constraint Manager](#).

Access Using

- *Menu Path: FlowPlan – Create Bundle*
- *Right Mouse Button Option: Create Bundle*
- *Toolbar Icon:* 

Related Topics:

- [bundle edit](#)
- [bundle split](#)
- [bundle properties](#)
- [bundle delete](#)

Creating A New Bundle of Rats

Follow these steps to create a new bundle of rats:

1. In IFP application mode, select one or more rats to bundle.

✔ Design density may make rat selection difficult. You can limit the find criteria to just rats by right-clicking in the Design window, then choosing *Super filter – Ratsnest* from the menu.

For tips on multi-object selection, see the [Object Selection Shortcuts](#) table.
The selected rats highlight and also appear in the *WorldView* window.

2. With your cursor on a selected rat, right-click and choose *Create Bundle* from the menu.
A new bundle containing the selected rats as members is created (appears in the Design window as a fat line) and an auto-generated name is assigned.


✔ You can change the name as well as other characteristics of the bundle using the `bundle properties` command.

3. Repeat steps 1 and 2 to create additional rat bundles as needed.

bundle delete

The `bundle delete` command lets you delete one or more bundles leaving their rat members in an unbundled state.

Access Using

- *Menu Path: FlowPlan – Delete Bundle*
- Right Mouse Button Option: *Delete Bundle*
- Toolbar Icon: 

Related Topics:

- [bundle edit](#)
- [bundle split](#)
- [bundle properties](#)
- [bundle create](#)
- [Deleting All Rat Bundles in the Design](#)

Deleting Selected Rat Bundles

To delete selected rat bundles, follow these steps:

1. In IFP application mode, select one or more rat bundles to delete.

✔ Design density may make bundle selection difficult. You can limit the find criteria to just bundles by right-clicking in the Design window, then choosing *Super filter – Ratbundle* from the menu.

For tips on multi-object selection, see the [Object Selection Shortcuts](#) table.
The selected bundles highlight and also appear in the *WorldView* window.

2. With your cursor on a selected bundle, right-click and choose *Delete Bundle* from the menu.
The selected bundles are removed leaving their rat members in an unbundled state.
3. Repeat steps 1 and 2 to delete additional rat bundles as needed.

Deleting All Rat Bundles in the Design

Perform these steps to delete all rat bundles in the design:

1. Ensure that nothing in the canvas is selected. In IFP application mode, right-click in the canvas background and choose *Selection set – Clear all selections* from the menu.
2. Click on the `bundle delete` icon in the FlowPlan toolbar.
All route bundles in the design are deleted.

Related Topics:

- [bundle edit](#)
- [bundle delete](#)

bundle edit

The `bundle edit` command lets you add or remove rats from a single bundle. You can also create and manage ratsnest bundles in [Constraint Manager](#).

Access Using

- *Menu Path: FlowPlan – Edit Bundle*
- *Right Mouse Button Option: Edit Bundle*
- *Toolbar Icon:* 

Related Topics:

- [bundle delete](#)
- [bundle split](#)
- [bundle properties](#)
- [bundle create](#)

Adding or Remove Rats From a Bundle

You can add or remove rats from bundles by following these steps:

1. In IFP application mode, hover your cursor over a bundle you want to edit.

✔ Design density may make bundle selection difficult. You can limit the find criteria to just bundles by right-clicking in the Design window, then choosing *Super filter – Ratbundle* from the menu.

The bundle highlights.

2. Right-click and choose *Edit Bundle* from the menu.

The bundle is active and awaiting the addition or removal of rat members.

3. Click on individual rats in the Design window or drag a window around a portion of several rats to add to the bundle.

- or -

Press and hold the Ctrl key and click on individual rat rake lines near the bundle pins or drag a window around several rat rake lines to remove rats from the bundle.

The rats are added or removed and the bundle display updates accordingly.

4. Repeat step 3 to add or remove other rats as needed.

- or -

Right-click and choose *Done* from the menu.

bundle import

The `bundle import` command displays a dialog box that lets you select and import rat bundles from another design. You can also use this command to restore the bundles of ECO-affected designs.

Once a source design is selected, a list of bundles available for import is presented. You have an option to import the entire list of bundles or you can choose to select bundles individually. Upon completion of the import, a summary is displayed in the status bar of the dialog box that conveys whether or not selected bundles were imported properly. You can click the *View log* button at the bottom of the dialog box to display an import log file that shows additional details.

Related Topics:

- [Importing Bundles From Another Design](#)
- [Saving and Restoring Bundles of ECO-affected Designs](#)

Bundle Import Dialog Box

Access Using

- *Menu Path: File – Import – Bundle*

<i>Import from:</i>	Specifies the path of the source design from which bundles are imported. You can also click on the adjacent icon to browse for and select a source design to import from.
<i>Replace all bundles with all bundles in import design</i>	When enabled (checked), specifies that all bundles in the source design replace all existing bundles in the target design. When disabled, specifies manual selection of individual bundles.
<i>Import bundle's flow as well as member rats</i>	When enabled (checked), imports the flow of selected bundles in the source design along with their rat members.
<i>Import bundle's plan data as well as member rats</i>	When enabled (checked), imports the associated plan data of selected bundles in the source design along with their rat members.
<i>Bundle name filter:</i>	Enables you to use text strings and Allegro supported wildcard characters to filter the names of the bundles in the <i>Available Bundles</i> list.
<i>Undo</i>	Reverses the results of bundle import operation and re-populates the <i>Selected Bundles</i> pane in the dialog box.
<i>Viewlog</i>	Displays the <code>bundle import</code> log file showing detailed events of the import operation.

Related Topics:

- [Saving and Restoring Bundles of ECO-affected Designs](#)

Importing Bundles From Another Design

Follow these steps to import bundles from another design:

1. Run the `bundle import` command.
The Bundle Import dialog box appears.
2. In the *Import from:* text box, enter the path of the source design containing bundles that you wish to import.
- or -
Click the adjacent icon to browse and select the source design.
The names of rat bundles in the source design populate the *Available bundles* pane in the dialog box.
3. If you wish to replace all of the existing bundles in your design with all the bundles in the *Available bundles* pane, enable (check) the *Replace all bundles with all bundles in import design* option in the dialog box.
The names of all bundles in the source design appear in the *Selected bundles* pane and are greyed out.
- or -
Click on bundle names to select individual bundles from the *Available bundles* pane.
The selected bundles move to the right into the *Selected bundles* pane.
4. If you wish to import the bundle flows along with the rat members of the bundles listed in the *Selected bundles* pane, enable (check) the *Import bundle's flow as well as member rats* option in the dialog box.
5. If you wish to import the plan data associated with the bundles listed in the *Selected bundles* pane as well as their rat members, enable (check) the *Import bundle's plan data as well as member rats* option in the dialog box.
6. Click *Apply* to begin the bundle import process.
The bundle names disappear from the *Selected bundles* pane, an import summary is displayed in the status bar at the bottom of the dialog box, and the *Viewlog* and *Undo* buttons are enabled.
7. If you wish to display the details of the bundle import log file, click on the *Viewlog* button.
8. Upon reviewing the log file, if you wish to reverse the results of the bundle import operation, click the *Undo* button in the dialog box. You can also choose *Edit – Undo* from the main menu.
The bundle import is undone and the names of the bundles previously imported re-appear in the *Selected bundles* pane.
9. Repeat steps 2 through 5 to import additional rat bundles into your design.
- or -

Click *OK* to dismiss the Bundle Import dialog box.

Related Topics:

- [bundle import](#)

Saving and Restoring Bundles of ECO-affected Designs

You can save and restore bundles of ECO-affected designs by performing these steps:

1. Open the design associated with the ECO.
2. Identify the ECO-affected bundles and delete all associated plan data (if any).
3. Choose *File – Save As* from the main menu to create a backup copy of the design for later use.
Use a unique name such as `<boardname>_bundle_restore.brd`
4. Delete the ECO-affected bundles identified in step 2.
5. Load in the new netlist or package as required by the ECO.
6. Save the design with the changes.
7. Restore the bundles deleted in step 4. Choose *File – Import – Bundle* from the main menu.
The Bundle Import dialog box appears.
8. In the *Import from:* text box, enter the path of the backup design that you created in step 3.
- or -
Click the adjacent icon to browse and select the backup design.
9. Enable (check) the following options in the dialog box:
Replace all bundles with all bundles in import design
Import bundle's flow as well as member rats
The names of all bundles in the backup design appear in the *Selected bundles* pane and are greyed out.
10. Click the *Apply* button to begin the bundle restoration.
The bundle names disappear from the *Selected bundles* pane, an import summary is displayed in the status bar at the bottom of the dialog box, and the *Undo* and *Viewlog* button are enabled.
11. If you wish to display the details of the bundle import log file, click on the *Viewlog* button.
12. If after reviewing the log file you wish to reverse the results of the bundle restoration, click the *Undo* button in the dialog box. You can also choose *Edit – Undo* from the main menu.
The bundle restoration is undone and the names of the bundles previously restored re-appear in the *Selected bundles* pane.
- or -
Click *OK* to accept the restoration results and dismiss the Bundle Import dialog box.

Related Topics:

- [bundle import](#)
- [Bundle Import Dialog Box](#)

bundle properties


The `bundle properties` command displays a dialog box that lets you control the routing behavior, bundle characteristics (such as name), and initial visibility settings for one or more selected bundles. You can also create and manage ratsnest bundles in [Constraint Manager](#).

Related Topics:


- [bundle delete](#)
- [bundle split](#)
- [bundle edit](#)
- [bundle create](#)
- [Editing the Properties of a Single Bundle](#)
- [Editing the Properties of Multi-selected Bundles](#)
- [Removing Bundle Property Overrides](#)


Edit Bundle Property Dialog Box

Access Using

- Menu Path: *FlowPlan – Bundle Properties*
- Right Mouse Button Option: *Bundle Properties*
- Toolbar Icon: 



General Tab

<i>Bundle Name</i>	The current name of the active bundle. You can enter a new bundle name adhering to the following rules: <ul style="list-style-type: none">• Allegro group name character and length rules• Name must be unique (not the same as another bundle)• No leading or trailing spaces (removed automatically)	
<i>Bundle Ownership</i>	Specifies whether you or the system has control over the rats that belong to the bundle. Options are:	
	<i>System</i>	The GRE route engine has control over the bundle and it may change or delete it when an <code>autobundle</code> command runs. This is the default for new bundles created by automatic bundling.
	<i>User</i>	You retain control over the bundle. An <code>autobundle</code> command cannot change or delete the bundle. The bundle is protected from system changes. This is the default for bundles created manually.
<i>Flow's x/y Guidance</i>	Specifies whether or not the GRE route engine is guided by the bundle flow path. Options are:	
	<i>Off</i>	The bundle flow path does not guide the GRE route engine. This is the default setting for new bundles.
	<i>Guide Router</i>	<div>The bundle flow path guides the GRE route engine.<div> When you edit the x/y path of the bundle's flow, this setting is automatically enabled.</div></div>


<i>Display Controls</i>		
<i>Bundle</i>	Specifies the visibility of the bundle in the Design window. This controls the display of flow lines, flow vias and rake lines.	
<i>Plan</i>	Specifies the visibility of the route plan associated with the bundle.	
<i>Ratsnests</i>	<p>Specifies the visibility of the ratsnest lines associated with the bundle.</p> <div style="border: 1px solid #fde9d9; padding: 10px; margin-top: 10px;">  If the bundle rats are not currently all on or all off, the button is left in the off position. This is possible due to the fact that ratsnest display can be controlled by a bundle or a net. </div>	




Routing Controls Tab (IFP only)




<i>Elongation Control</i>		
<i>Trombone</i>	Specifies a trombone pattern for elongation. This is the default pattern.	
<i>Accordian</i>	Specifies an accordian pattern for elongation.	
<i>Sawtooth</i>	Specifies a sawtooth pattern for elongation.	
<i>Gap</i>	Specifies the gap between adjacent wraps of regular patterns. The default is three times the trace width.	
<i>Min Amplitude</i>	Specifies the minimum amplitude for any wrap of a regular pattern. The default is three times the trace width.	
<i>Max Amplitude</i>	Specifies the maximum amplitude for any wrap of a regular pattern. The default is <i>No limit</i> .	
<i>Corner Type</i>	Specifies the corner type for the elongation when a bend is made.	
<i>Corner Size</i>	Specifies the minimum corner length bases on the trace width. The default is equal to the trace width.	
<i>Spacing</i>		

<i>Within Bundle</i>	Specifies whether the GRE route engine should be forced to route the rats within the bundle at the minimum spacing constraint value between themselves or try to increase the spacing within the allowable range.		
	<i>Default</i>	GRE route engine is unconstrained with regard to packing or unpacking.	
	<i>Min DRC plus</i>	GRE route engine is forced to meet minimum DRC spacing plus an additional spacing amount. Options are:	
		<i>Min</i>	<p>The minimum amount of additional spacing GRE route engine attempts to meet. This value may be any positive real number that is less than the <i>Max</i> value.</p> <div>  Setting this value to zero has a packing action that keeps the bundle packed at its minimum c-line to c-line constraint. </div>
		<i>Max</i>	<p>The maximum amount of additional space GRE route engine tries to meet if it can. If this value is met, no further attempts are made. This value may be any positive real number equal to or greater than the <i>Min</i> value.</p>
Clear Override	<p>Removes a routing control property override specified for the bundle.</p> <div>  For each override you want to clear, you must first click this button, followed by a click on the override item (blue color) in the tab. </div>		



Bundle Layer Tab

<i>Layer Transitions</i>	
<i>Max Transitions (IFP only)</i>	<p>Specifies the maximum number of transitions that are allowed per bundle member.</p> <div>  This value does not include pin escapes. Regardless of the value used, GRE route engine will not violate the MAX_VIA_COUNT constraint. </div> <p>Options are:</p>

	<i>Unlimited</i>	An unlimited number of transitions is allowed.
	<i>Limited to</i>	Maximum number of transitions allowed. This value must be a positive integer.
<i>Layer Matching (IFP only)</i>	Specifies whether bundle members are to route on the same layer. Options are:	
	<i>Off</i>	Bundle members are allowed to route on different layers.
	<i>Route on same layer</i>	<p>Bundle members must route on the same layer. If a transition does occur, then all members must transition together.</p> <div>  This option is disabled if there is no single layer on which all bundle members can be routed. </div>
<i>Enable</i>	<p>Shows whether a layer is enabled (checked) or disabled for the bundle.</p> <div>  The bundle derives its default layer usage from the Allegro constraint system. This property lets you further refine the bundle routing solution to a subset of layers that are allowed by the constraint system </div> <p>Options are:</p>	
	<i>Layer Check box</i>	<p>Enables or disables the named layer from being used to route the bundle. The default is enabled (checked) for any layer allowed by the constraint system.</p> <div>  This control is unavailable when the layer is not allowed by the combined LayerSet constraints on the members of the bundle. Also, a warning message may appear when you disable a layer. This happens in cases where disabling the layer usage is not recommended by the system. You are given alternate choices. </div>
<i>Direction</i>	Shows the preferred routing direction for a layer. The cells cannot be edited.	

<i>LayerSetGroup</i>	<p>Shows layer membership in one or more layer set groups. The cells cannot be edited.</p> <div> If none of the layers belong to layer set groups, the columns are hidden.</div>
<i>One Layer On</i>	<p>Enables a single layer that you select, and disables all others.</p> <div> You must click this button first, followed by a click on the layer that you want enabled.</div>
<i>Clear Override</i>	<p>Removes a layer property override specified for the bundle.</p> <div> For each override you want to clear, you must click this button first, followed by a click on an override item (blue color) in the tab.</div>

Flow Line Tab (IFP only)

<i>Remove Flow Line Layer Usage</i>	Resets all layers available for routing the bundle (as defined in the Allegro constraint system) back to the enabled (checked) state.	
<i>Enable</i>	<p>This column shows whether a layer is enabled (checked) or disabled for the routes of the selected bundle flow line.</p> <div style="border: 1px solid #fde725; padding: 10px; margin: 10px 0;">  The flow line layer usage is limited to a subset of the layers enabled for the bundle (see the Bundle Layer tab). </div> <p>Options are:</p>	
	<i>Layer Check box</i>	<p>Enables or disables the named layer from being used to route the selected bundle flow line. The default is enabled (checked) for any layer allowed for the bundle itself.</p> <div style="border: 1px solid #fde725; padding: 10px; margin: 10px 0;">  This control may be unavailable when the layer is not allowed by the combined layerset constraints for all members of the bundle. Also, a warning message may appear when you disable a layer. This happens in cases where disabling the layer usage is not recommended by the system. You are given alternate choices. </div>
<i>Direction</i>	Shows the routing bias direction for the layer (as set in the global layer controls).	
<i>Bundle</i>	Shows whether the bundle is allowed or forbidden to route on the layer.	
<i>LayerSet Group</i>	Shows whether the layer belongs to a layerset group.	

Related Topics:

- [Editing the Properties of Multi-selected Bundles](#)
- [Removing Bundle Property Overrides](#)

Editing the Properties of a Single Bundle

Perform these steps to edit the properties of a bundle:

1. In IFP application mode, hover your cursor over a rat bundle whose properties you want to edit.

✔ Design density may make bundle selection difficult. You can limit the find criteria to just bundles by right-clicking in the Design window, then choosing *Super filter – Ratbundle* from the menu.

The bundle highlights.

2. Right-click and choose *Bundle Properties* from the menu.
The Edit Bundle Property dialog box appears.
3. Click on the appropriate tab to access the properties you want to edit.
4. Change the bundle property values and settings as needed.
If necessary, click *Help* in the dialog box to access property descriptions.

⚠ Any property with a black color, changes to a blue color when you modify it indicating an override of its associated global parameter value. Existing overrides in blue may be cleared using the *Clear Override* button at the bottom of the tab. See the procedure [To remove property overrides](#) for further details.

5. Repeat steps 3 and 4 to edit other bundle properties as required.
6. Click *OK* to update the bundle property values and dismiss the dialog box.

Related Topics:

- [bundle properties](#)
- [Removing Bundle Property Overrides](#)

Editing the Properties of Multi-selected Bundles

To edit the properties of multi-selected bundles, perform these steps:

1. In IFP application mode, select one or more rat bundles whose properties you want to edit.

✔ Design density may make bundle selection difficult. You can limit the find criteria to just bundles by right-clicking in the Design window, then choosing *Super filter – Ratbundle* from the menu.

For tips on multi-object selection, see the [Object Selection Shortcuts](#) table.
The selected bundles highlight and also appear in the *WorldView* window.

2. Hover your cursor over one of the selected bundles, right-click and choose *Bundle Properties* from the menu.

The Edit Bundle Property dialog box appears.

3. Click on the appropriate tab to access the properties you want to edit.

✔ The color of the property label in the tab indicates the status of its value relative to all the bundles in the selection set as described in the following table.

This property color ...	means ...
Black	All bundles are using the global parameter value.
Blue	All bundles are using the same override value.
Brown	Bundles are using mixed values for this property.

4. Change the bundle property values and settings as needed.
If necessary, click *Help* in the dialog box to access property descriptions.

⚠ Any property with a black color, changes to a blue color when you modify it indicating an override of its associated global parameter value. Existing overrides in blue may be cleared using the *Clear Override* button at the bottom of the tab. See the procedure [To remove property overrides](#): for further details.

5. Repeat steps 3 and 4 to edit other bundle properties as required.
6. Click *OK* to update the bundle property values for all the selected bundles and dismiss the dialog box.

Related Topics:

- [bundle properties](#)
- [Edit Bundle Property Dialog Box](#)

Removing Bundle Property Overrides

You can remove the bundle property overrides by following these steps:

1. In IFP application mode, select one or more rat bundles containing property overrides that you want to remove.

✔ Design density may make bundle selection difficult. You can limit the find criteria to just bundles by right-clicking in the Design window, then choosing *Super filter – Ratbundle* from the menu.

For tips on multi-object selection, see the [Object Selection Shortcuts](#) table.
The selected bundles highlight and also appear in the *WorldView* window.

2. Hover your cursor over one of the selected bundles, right-click and choose *Bundle Properties* from the menu.
The Edit Bundle Property dialog box appears.
3. Click on the appropriate tab to access property overrides (blue and brown colors) that you want to remove.
4. Click the *Clear Override* button at the bottom of the tab, then click on the property override item that you want to remove.
The property value changes back to match its associated global parameter value and the property color of the item changes to black.
5. Repeat steps 3 and 4 to remove additional property overrides on other tabs as needed.
6. Click *OK* to update the property values for all selected bundles and dismiss the dialog box.

Related Topics:

- [bundle properties](#)
- [Edit Bundle Property Dialog Box](#)
- [Editing the Properties of a Single Bundle](#)

bundle restore

An internal Cadence engineering command.

bundle show

The `bundle show` command displays rat bundles associated with one or more selected design objects.

Access Using

- *Menu Path: Display – Show Bundles – Selected*
- Right Mouse Button Option: *Show Bundle*
- Toolbar Icon

Related Topics:

- [bundle show_all](#)
- [bundle show_unplanned](#)
- [Displaying Selected Rat Bundles](#)

Displaying Rat Bundles Associated with Selected Objects

Follow these steps to display rat bundles associated with selected objects:

1. In IFP application mode, select one or more objects associated with the route plan (plan lines, rats, components, symbols, pins, nets, c-lines, c-line segments, etc.).

✔ Design density may make object selection difficult. You can limit the find criteria to just one specific object type by right-clicking in the Design window, then choosing *Super filter – <object_type>* from the menu.

For tips on multi-object selection, see the [Object Selection Shortcuts](#) table.
The selected objects highlight and also appear in the *WorldView* window.

2. With your cursor on a selected object, right-click and choose *Show Bundle* from the menu.
The rat bundles associated with the selected objects appear.
3. Repeat steps 2 and 3 to display rat bundles associated with other objects as needed.

Displaying Selected Rat Bundles

To display selected rat bundles, perform these steps:

1. Right-click in the Design window and choose *Selection set – Object Browser* from the menu.
The Find by Name or Property dialog box appears.
2. At the top of the dialog box, set *Object type* to Group, then select the name of one or more bundles you want to display from the *Available objects* window and move them to the *Selected objects* window.
3. Click OK to dismiss the dialog box.
The selected bundles appear in the *WorldView* window.
4. Choose *Display – Show Bundles – Selected*.
The selected bundles appear in the canvas.

Related Topics:

- [bundle show](#)

bundle show_all

The `bundle show_all` command displays all rat bundles in the plan.

Access Using

- *Menu Path: Display – Show Bundles – All*
- *Right Mouse Button Option: Show All – Bundles*

Related Topics:

- [bundle show](#)
- [bundle show_unplanned](#)

Displaying All Rat Bundles

Perform this step to display all rat bundles:

1. Choose *Display – Show Bundles – All* from the menu bar.
All rat bundles appear.

bundle show_unplanned

The `bundle show_unplanned` command displays rat bundles in the design that have not been planned by the GRE route engine. All other bundles currently visible and already planned are hidden. An unplanned bundle is one that contains at least one rat that has no planning data.

 This command operates on the entire design and not on pre-selected objects.

Access Using

- *Menu Path: Display – Show Bundles – Unplanned*
- *Right Mouse Button Option: Show All – Unplanned Bundles*

Related Topics:

- [bundle show](#)
- [bundle show_all](#)

Displaying Only Rat Bundles that are Not Planned

To display only unplanned rat bundles, follow these steps:

1. In IFP application mode, ensure that nothing in the design is selected by clicking the right mouse button in the canvas background and choosing *Selection set – Clear all selections* from the menu.
2. Click the right mouse button in the canvas background and choose *Show All – Unplanned Bundles* from the menu.


All bundles not planned by the GRE route engine appear and all other bundles are hidden.

bundle split

The `bundle split` command lets you split a single rat bundle into two or more individual bundles. It creates new bundles with auto-generated names and preserves the name of the original bundle. The properties of the source bundle are propagated to each destination bundle. Optionally, you can choose modify the flow of the destination bundles to match the flow of the source bundle and re-locate the bundles within the design.

 This command does not operate on multi-selected source bundles.

Access Using

- *Menu Path: FlowPlan – Split Bundle*
- Right-click Command: *Split Bundle*
- Toolbar Icon: 
- Related pop-up options:
 - *Rename Bundle*: Lets you rename the destination bundle.
 - *Copy Flow*: Modifies the flow of the destination bundle to match the source bundle.
 - *Next Destination Bundle*: Starts a new destination bundle to receive rats from the source bundle.

Related Topics:

- [bundle delete](#)
- [bundle properties](#)
- [bundle edit](#)
- [bundle create](#)
- [Splitting a Rat bundle into Multiple Bundles](#)

Splitting a Rat Bundle Into Two Bundles

You can split a rat bundle into two bundles by performing these steps:

1. In IFP application mode, hover your cursor over a bundle you want to split.

The selected bundle highlights.

✔ Design density may make bundle selection difficult. You can limit the find criteria to just bundles by right-clicking in the Design window, then choosing *Super filter – Ratbundle* from the menu.

2. Right-click and choose *Split Bundle* from the menu.

A destination bundle is created with an auto-generated name and is awaiting rat members from the source bundle. If desired, you can right-click and choose *Rename Bundle* to change the name.

3. Click on rat rake lines near the bundle pins to move individual rats from the source bundle to the active destination bundle.

- or -

Drag a window around a group of rat rake lines to move several rats from the source bundle to the active destination bundle.

The selected rats are added to the active destination bundle and the bundle display updates in the design canvas.

✔ You can also move rats back to the source bundle by clicking on their rake lines in the destination bundle.

4. Continue to select other rats to move between the source and active destination bundle until you are satisfied with the bundle configurations.

5. Optionally, right-click and choose *Copy Flow* from the menu to modify the flow of the destination bundle to match the flow of the source bundle.


The destination bundle with its updated flow attaches to your cursor. Move and track the destination bundle using your mouse, then click to place it within the design.

i When a flow is copied, its width and layers are automatically adjusted to be compatible with the destination bundle.

6. Right-click and choose *Done* from the menu to end the command.

Splitting a Rat bundle into Multiple Bundles

To split a rat bundle into multiple bundles, perform the following steps:

1. Follow steps 1 through 5 in the previous procedure to create the first destination bundle, then proceed to step 2.
 2. Right-click and choose *Next Destination Bundle* from the menu.
Another destination bundle is created with an auto-generated name and is awaiting rat members from the source bundle. If desired, you can right-click and choose *Rename Bundle* to change the name.
 3. Click on rat rake lines near the bundle pins to move individual rats from the source bundle to the active destination bundle.
- or -
Drag a window around a group of rat rake lines to move several rats from the source bundle to the active destination bundle.
The selected rats are added to the active destination bundle and the bundle display updates in the design canvas.
You can also move rats back to the source bundle by clicking on their rake lines in the active destination bundle.
 4. Optionally, right-click and choose *Copy Flow* from the menu to modify the flow of the destination bundle to match the flow of the source bundle.
The destination bundle attaches to your cursor. Move and track the destination bundle using your mouse, then click to place it within the design.
-  When a flow is copied, its width and layers are automatically adjusted to be compatible with the destination bundle.
5. Repeat steps 2 and 4 to create additional destination bundles as needed.
- or -
Right-click and choose *Done* from the menu to end the command.


Related Topics:

- [bundle delete](#)
- [bundle split](#)

bundle toggle

The `bundle toggle` command lets you reverse the display state of rat bundles associated with one or more selected objects. When objects are selected, the command determines the current visibility state of the associated bundles and reverses it. When no objects are selected the command works globally on the entire design where all bundles currently displayed are hidden. If all bundles are currently hidden, they appear.

Access Using

- Toolbar Icon: 

Related Topics:

- [bundle show](#)
- [bundle show_all](#)
- [bundle blank](#)
- [bundle blank_all](#)
- [Toggling the Display of All Rat Bundles in the Design](#)

Toggling the Display of Rat Bundles Associated with Selected Objects

Follow these steps to toggle the display of rat bundles associated with selected objects:

1. Select one or more objects associated with the route plan (bundle, rat, component, symbol, pin, net, etc.).

✔ Design density may make object selection difficult. You can limit the find criteria to just one specific object type by right-clicking in the Design window, then choosing *Super filter – <object_type>* from the menu.

For tips on multi-object selection, see the [Object Selection Shortcuts](#) table.
The selected objects highlight and also appear in the *WorldView* window.

2. Click on the `bundle toggle` icon in the FlowPlan toolbar.
The visibility of the associated bundles is reversed.
3. Repeat steps 1 and 2 to toggle the display of bundles associated with other objects as needed.

Toggling the Display of All Rat Bundles in the Design

To toggle the display of all rat bundles in your design, perform these steps:

1. Ensure that nothing in the canvas is selected. In IFP application mode, right-click in the canvas background and choose *Selection set – Clear all selections* from the menu.
2. Click on the `bundle toggle` icon in the FlowPlan toolbar.
All route bundles currently displayed in the design are hidden.
- or -
If all bundles are currently hidden, they appear.

Related Topics:

- [bundle delete](#)
- [bundle toggle](#)

button

The `button` command re-assigns an action to mouse buttons. Currently the only supported action is the mouse wheel. This command works only in Cadence tools on Windows, not at the operating-system level. The default mouse wheel behavior is zoom in and out, which is set in the global `env` file. To make button assignments permanent, define and save buttons in a local or site environment file that remain in effect at every login until you change the environment file.

To delete a button and the action assigned to it, use the `unbutton` command.

Syntax

```
button [modifier] | [wheel] | [wheel_up] | [wheel_down] | [action to execute]
```

modifier	Create buttons with or without <code>Shift</code> and <code>Control</code> keys or a combination of both. Modifiers are <code>S</code> (<code>Shift</code> key), <code>C</code> (<code>Control</code> key), and <code>SC</code> (<code>Shift</code> and <code>Control</code>) and are case sensitive. (optional)
wheel	Specifies upward or downward mouse wheel movement if the <code>wheel_up</code> and <code>wheel_down</code> arguments are unspecified.
wheel_up	Specifies an upward mouse wheel movement. Defining this argument suppresses the upward mouse movement of the <code>wheel</code> argument.
wheel_down	Specifies a downward mouse wheel movement. Defining this argument suppresses the downward mouse movement of the <code>wheel</code> argument.
action to execute	Specifies the action to execute when the mouse rolls up or down.

If you enter `button` at the console window prompt without arguments, the Defined Mouse Buttons window lists all assigned button actions.

If you enter `button` and one argument at the console window prompt, the Defined Mouse Buttons window lists only what is assigned to that button.


System-set variables

The tool automatically sets the following environment variables, whose values are updated dynamically to reflect their current state whenever you roll the mouse wheel. You cannot enter values for these variables.

<code>_wheelcnt</code>	System-set variable that specifies the degree to which the mouse wheel rolls in detents (or audible clicks). Value is an integer between -4 and -1 for downward mouse wheel rolls; between 1 and 4 for upward mouse wheel rolls. Zero is not a valid value.
<code>sx1, sx2</code>	System-set variable that specifies the current coordinates of the mouse wheel's position in design units.

Examples

1. Zoom in and out default operation. This zoom centers on the current cursor location.

 When you roll the mouse wheel up or down, the tool dynamically references and substitutes the value of the `_wheelcnt` environment variable to reflect its current state. The single quotation marks that enclose `_wheelcnt` ensure that variable substitution does not occur when you assign an action to a button.

```
button wheel zoom in '$_wheelcnt'
```

```
button wheel zoom in '$_wheelcnt'
```

2. Change the active subclass when you press the `Shift` key and roll the mouse wheel up:

```
button Swheel_up subclass -+
```

3. Change to an alternative subclass when you press the `Shift` key and roll the mouse wheel down while using the `.add connect` command:

```
button Swheel_down altsubclass -+
```

4. Access the `add connect` or `slide` command when you press the `Control` key and roll the mouse wheel up or down, respectively:

```
button Cwheel_up add connect
```

```
button Cwheel_down slide
```

bw doc

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

The `bw doc` command is used to define the bond wire groupings. Ensures final designs are manufactured accurately by keeping groupings up to date as changes are made to the bond shell, whether for an ECO of the die, a new revision of the substrate, or even the design of an alternate design variant.

Running the command opens the Bond Wire Documentation Prep dialog box where you can add the needed layers. You can control the wires on a layer by wire profile, by the starting layer or die, and/or the ending layer or die. As you tweak the combination of filters for each layer, you will see how many wires map to that layer. Top of the dialog box displays how many wires are not yet mapped to any layer.

Click `Auto generate all in-use combinations` to pre-populate the chart with the different combinations. All the bond wires are copied to the appropriate layers. The individual layers can be added to your artwork films (which will also allow you to export these to a PDF document), but can also be referenced in your DXF or GDSII conversion files.

