

F 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	9
F Commands	9
fabmaster out	11
fanout_by_pick	12
Routing Short Pin Escape Wires from Pins to Vias	13
fdcheck	13
feedback	14
Export Logic Dialog Box	15
Exporting Native (Cadence) Logic	17
Exporting Third-party (Other) Logic	18
file_property	19
File Properties Dialog Box	20
Locking Database Files	25
Unlocking Database Files	28
file_register	29
Registered File Extensions Dialog Box	30
file_unregister	31
Running File Register Command	32
filemgr	33
fill_ipf	34
Syntax	34
Running fill_ipf Command	36
film area	37
film param	38
Artwork Control Dialog Box	39
Creating Film Records for a Gerber Data File	48
Suppressing the Shapefill Algorithm in Negative Artwork	50
film res	52
	52

Thick/Thin Film Resistor Generator Controls Dialog Box	54
Running the film_res command	55
Reviewing the film_res.log File	56
find_by_name	57
	57
Find by Name/Property Dialog Box	58
Running the Find By Name Command	59
find_by_query	60
	60
Find by Query Dialog Box	60
Find Setting Dialog Box	63
Procedure for Editing Filter Settings	65
Procedure for Running a Query	66
find_control	67
findfilter	68
findprop	69
	69
findprop Dialog Boxes	70
Procedures	70
fix	71
	71
Restricting Elements to Prevent Modification	72
flash_convert	73
flipdesign	74
flow copy	75
Copying the Flow of a Bundle	76
flow create	76
	76
Flow Create Options Dialog Box	78
Running the Flow Create Command	79
flow default	80
Restoring the default flow to selected bundles	81
flow move	81
Moving a flow	82
flow rat layer control	83
	83
Editing Bundle Flow	84

Running the Rat Layer Control Command	85
flow sequence	86
Running the Flow Sequence Command	87
flow slide	88
	88
Sliding a Flow Line Segment	89
Sliding a Flow Line Vertex	90
Sliding a Flow Line Via	91
Flow Line Editing Shortcuts	92
flow vertex	93
	93
Inserting a vertex into a flow line segment	94
Moving an Existing Flow Line Vertex	95
flow vertex delete	96
	96
Deleting a Flow Line Vertex	97
flow vertex insert	98
	98
Inserting a Vertex into a Flow Line	99
flow vertex move	100
	100
Moving a flow line vertex	101
flow via delete	102
	102
Deleting a Flow Line Via	103
flow via insert	104
	104
Inserting a flow via into a flow line segment	105
flow via move	106
	106
Moving an Existing Via in a Flow Line	107
form	108
form_next	109
	109
form_prev	110
	110
front	111

fse_arc_tangent	112
	112
Arc Tangent Command: Options Panel	113
Drawing an arc tangent between two points from two different segments	114
fse_break_delete	115
	115
Break and Delete Command: Options Panel	116
Removing extra lines and segments from a shape boundary	117
fse_edge_move	118
	118
Edge Move Command: Options Panel	119
Moving a shape edge	120
fse_edge_spread	121
	121
Edge Spread Command: Options Panel	122
Spreading a shape edge	123
fse_edge_stretch	124
	124
Edge Stretch Command: Options Panel	125
Stretching a shape edge	126
fse_end_connect	128
	128
Line End Connect Command: Options Panel	129
End connecting the ends of two lines	130
fse_seg_tangent	131
	131
Tangent Segment Command: Options Panel	132
Drawing a tangent line or arc from a designated start point	133
fse_shape_chamfer	134
	134
Shape Chamfer Command: Options Panel	135
Converting all corners of a shape to arcs or miters	136
fse_shape_logicop	137
	137
Shape Logicop Command: Options Panel	138
Creating a boolean shape from two groups of overlapping shapes	139
fse_shape_scale	140

	140
Shape Scale Command: Options Panel	141
Creating a scaled copy of a shape	142
fse_shape_zcopy	143
	143
Shape Z-Copy Command: Options Panel	144
Creating scaled copies of a shape on ETCH subclasses	146
Creating scaled copies of a shape on non-ETCH subclasses	148
fse_shape_zdelete	149
	149
Shape Z-Delete Command: Options Panel	150
Deleting Z-copied shapes from selected classes and subclasses	151
fse_vertex_convert	152
	152
Vertex Convert Command: Options Panel	153
Converting a corner of a shape to an arc or a miter	154
fse_vertex_insert	155
	155
Vertex Insert Command: Options Panel	156
Inserting a vertex in the boundary edge of a shape	157
fse_vertex_move	159
	159
Vertex Move Command: Options Panel	160
Moving a vertex	161
fsp auto pinswap	162
Running the Auto Pinswap Command	163
fsp load database	164
fsp manual pinswap	165
Running the Manual Pinswap	165
fsp synchronize	166
func	166
	166
func Command Dialog Boxes	167
Displaying Information	168
Editing Object Properties	169
funckey	170
	170

Creating a Function Alias	172
---------------------------	-----

F Commands

fabmaster out	fanout_by_pick	fdcheck
feedback	file_property	file_register
file_unregister	filemgr	fill_ipf
film area	film param	film res
find_by_name	find_by_query	find_control
findfilter	findprop	fix
flash_convert	flipdesign	flow copy
flow create	flow default	flow move
flow rat layer control	flow sequence	flow slide
flow vertex	flow vertex delete	flow vertex insert
flow vertex move	flow via delete	flow via insert
flow via move	form	form_next
form_prev	front	fse_arc_tangent
fse_break_delete	fse_edge_move	fse_edge_spread

F Commands
F Commands

fse_edge_stretch	fse_end_connect	fse_seg_tangent
fse_shape_chamfer	fse_shape_logicop	fse_shape_scale
fse_shape_zcopy	fse_shape_zdelete	fse_vertex_convert
fse_vertex_insert	fse_vertex_move	fsp auto pinswap
fsp load database	fsp manual pinswap	fsp synchronize
func	funckey	

fabmaster out

The `fabmaster out` command generates Allegro to fabmaster file in a text format.

This command extracts the database using the extract view file `fabmaster.txt` located in the installation hierarchy at `<installation_hierarchy\share\pcb\text\views`.

fanout_by_pick

Routes short pin escape wires from pins to vias. Lets you control pin and via sharing, specify the layer depth, control the escape direction, and set a temporary via grid for this command to use.

Access using:

- Menu path:
 - *Route – Fanout By Pick* (Allegro X PCB Editor and Allegro X SI)
 - *Route – Router – Fanout By Pick* (APD with *SiP Layout* option)

Routing Short Pin Escape Wires from Pins to Vias

1. Run the `fanout_by_pick` command.
2. Right-click to display the pop-up menu and choose *Setup*.
The Automatic Router Parameters dialog box appears with the Fanout tab selected.
3. Make your selections. For additional information, see the fanout tab of the [Automatic Router dialog box](#).
4. Click *OK* to save the changes and dismiss the dialog box.
5. Choose a segment or group of segments.
6. Choose one of the options from the pop-up menu, as described below.

<i>Done</i>	Terminates the command, saving any routing performed while the command was active.
<i>Oops</i>	Removes the results of the last route.
<i>Cancel</i>	Terminates the command without saving any routing.
<i>Temp Group</i>	Enables you to route groups of connections.
<i>Complete</i>	Completes the selection of the items to group.
<i>Setup</i>	Opens the Automatic Router Parameter dialog box.
<i>Results</i>	Opens the routing results form to display the results of the current routing session.

fdcheck

The `fdcheck` command run on Linux and checks for available file descriptors. If none exist, the command issues a warning and attempts to make some available. If file descriptors are available, the command returns *OK* to the status line in the command console. Use `fdcheck` if you suspect that you have run out of file descriptors; for example, if you are unable to open a shell window.

feedback

Exports logic information from a design to another file or program. This command displays the Export Logic dialog box

Related Topics

- [Exporting Native \(Cadence\) Logic](#)
- [Exporting Third-party \(Other\) Logic](#)

Export Logic Dialog Box

Access Using

- Menu Path: *File – Export – Logic*

Use this dialog box to backannotate board logic and design logic changes to a Cadence or third-party schematic

<i>Cadence Tab</i>	
<i>Logic type</i>	Selects the type of logic you want to feed back.
<i>Export using Constraint Manager enabled flow</i>	Exports the ecets in the current design back to the schematic.
<i>Database branding</i>	Identifies the logic format of the file to be loaded.
<i>Export to directory:</i>	Indicates the location of the original logic file for Design Entry HDL or System Connectivity Manager logic type. Use the Browse button to specify a different directory. Note: The default for HDL based logic is the location of the last active project. For SCALD, the default is the current working directory displayed as a period.
<i>Export Cadence</i>	Runs feedback.
<i>Other Tab</i>	
<i>Comparison Design</i>	Indicates the location of the original Third Party logic file. Use the Browse button to navigate to the file.
<i>Include Spare TF-Functions</i>	Determines whether spare gates are to be included in the backannotation file. The options are: <i>On</i> : Indicates spare gates are to be included. <i>Off</i> : Indicates spare gates are not to be included. <i>Export Other</i> : Runs feedback.

Related Topics

- [Exporting Third-party \(Other\) Logic](#)

Exporting Native (Cadence) Logic


You can export logic to Cadence (native) front-end tools, or to third-party (other) tools.

1. Choose *File – Export – Logic*.

The Export Logic dialog box appears.

2. Use the *Browse* button to navigate to the location where the transfer files are to be located.
The default path is your current working directory, displayed in the *Export to directory* field as a period (unless another location was specified and applied in a previous session).
3. Checking *Export using Constraint Manager enabled flow* allows you to backannotate constraints to Design Entry HDL and System Connectivity Manager. It is selected by default if you imported logic that contained constraints.
4. Click *Export Cadence*.

The tool creates the output files in the location indicated. A log file, `genfeed.log`, is also created, which you can view by using *File – Viewlog*, as well as a `boardname.baf` file from the active board/substrate. This file contains reference designator assignments (after gate/pin swap, or reference designator rename), and a was/is list of all pins for each part, indicating changes that may have occurred during gate and pin swapping.

 If you run Design Sync/Import Physical from Project Manager or Design Entry HDL and System Connectivity Manager, you can generate feedback files (*Export – Logic*), package the design, and backannotate the schematic. Design Sync can be used to view changes before backannotation.

Related Topics

- [feedback](#)

Exporting Third-party (Other) Logic

1. Choose *File – Export – Logic*.
The Export Logic dialog box appears.
2. Click the *Other* tab to denote third-party (non-Cadence) logic.
3. In the Comparison design field, enter or browse for the name of the original design file (before gate/pin swapping or reference designator rename).
4. Choose *Include spare TFunctions*.
Check this option to have spare gates to be included in the output file. Spare gates appear at the end of the backannotation file.
5. Click *Export Other*.
If you are backannotating from Windows, The tool creates the output files in the current working directory. A log file, backan.log, is also created, which you can view via viewlog.


Related Topics

- [feedback](#)
- [Export Logic Dialog Box](#)

file_property

Lets you set an optional password-protected database lock from the File Properties dialog box. Doing so marks your file as read-only in the database (as opposed to on your platform's operating system). This ensures that your design is not accidentally over-written by you or an unauthorized user when attempting to save without saving as a different file name.

As an added level of security, you can also specify a NTP Time Server when locking a database with an expiration duration. The name of the server is stored in the design and is used to obtain the current time when opening the design.

 You cannot open a design if the NTP Server is inaccessible. It is recommended to check if the NTP Server is accessible to users before locking the database.

For additional information, see Protecting Files with Edit Locks in the *Allegro User Guide: Getting Started with Physical Design*.

For information on how to perform these actions in batch mode, see the `dbdoctor` command in the *Allegro PCB and Package Physical Layout Command Reference*.

Related Topics


- [Locking Database Files](#)
- [Unlocking Database Files](#)

File Properties Dialog Box

Access Using

Menu Path: File – Properties

Use this dialog box to secure your design file with a read-only database lock.

<i>Locking</i>		
<i>File</i>		
<i>Lock duration</i>	Locks the database.	
	<i>Password</i>	Sets a password. Allows a maximum of 20 legal alphanumeric characters. Illegal characters are: spaces, backslashes (\), and dashes(-). Passwords are case-sensitive and cannot be changed without first unlocking the database file.
	<i>Expiration duration (days)</i>	<p>Locks the database 14, 90, 180 and 365 days. Setting to None locks the database for an unlimited period of time. You can also enter the amount of days, with a minimum value of 1 day.</p> <div>  You cannot open any locked database once the <i>Expiration Duration</i> has expired. </div>
<i>Lock type</i>		

	<i>View(No Save and Export)</i>	<p>Locks the database for saving and exporting design data such as techfiles, libraries, and modules. Use this option to share database for viewing only. All the export commands are grouped under five categories and are enabled by default. Selecting an option disables the export command belong to that group.</p> <ul style="list-style-type: none">• Manufacturing• Database• Logic• Constraints• MACD/ECAD <p>Refer to Table#160;6-1, for list of supported export options. If enabled, the database file name is automatically updated to <code><design_name>_view_locked</code> and becomes active database.</p>
	<i>Export (No Export)</i>	<p>Locks the database for exporting design data, such as techfiles, libraries, and modules. All the export commands are grouped under five categories and are enabled by default. Selecting an option disables the export command belong to that group.</p> <ul style="list-style-type: none">• Manufacturing• Database• Logic• Constraints• MACD/ECAD <p>Refer to Table#160;6-1, for list of supported export options. If enabled, the database file name is automatically updated to <code><design_name>_export_locked</code> and becomes active database.</p>
	<i>Write (No Save)</i>	<p>Locks the database for saving design data. If enabled, the database file name is automatically updated to <code><design_name>_write_locked</code> and becomes active database.</p>
<i>Unlock</i>	<p>Unlocks the database. If locked with a password, requires the password option to unlock.</p>	
<i>NTP time service option</i>		

	<i>Use a NTP server to verify time</i>	Choose to use a NTP server to verify time.
	<i>Server</i>	Specify the server. The default NTP server is <i>0.pool.ntp.org</i> .
	<i>Test</i>	Choose to test NTP Server accessibility. If the test is successful, confirmer displays the network time. If the test is unsuccessful, the confirmer states that the network time cannot be obtained using the specified server. You can test the NTP Server accessibility by enabling the <i>Lock Design</i> , <i>Use a NTP server to verify time</i> options and entering the <i>Server</i> name on an unlocked design.
<i>Info</i>		
	<i>By</i>	Displays the name of the user who locked the database.
	<i>System</i>	Displays the name of the system on which it was locked.
	<i>When</i>	Displays the time when database was locked.
<i>Comment</i>	Optional. Provides new, or updates existing, user comments.	
<i>Tiering</i>		
	<i>Update to Tier</i>	Choose to update the current design capability to match the current product and options selected.

Table -1 Valid Export Options

<i>Manufacture</i>			
	<i>File</i>	<i>Export</i>	<i>IPC 356</i>
	<i>File</i>	<i>Export</i>	<i>IPC 2581</i>
	<i>File</i>	<i>Export</i>	<i>ODB++ inside</i>
	<i>Manufacturing</i>	<i>Artwork</i>	
	<i>Manufacturing</i>	<i>Stream Out(GDS II)</i>	

	<i>Manufacturing</i>	<i>DFx Check</i>	
	<i>Manufacturing</i>	<i>NC</i>	<i>Drill Legend</i>
	<i>Manufacturing</i>	<i>NC</i>	<i>NC Drill</i>
	<i>Manufacturing</i>	<i>NC</i>	<i>NC Route</i>
	<i>Manufacturing</i>	<i>Variants</i>	<i>Create Assembly Drawing</i>
	<i>Manufacturing</i>	<i>Variants</i>	<i>Create Bill of Materials</i>
<i>Database</i>			
	<i>File</i>	<i>Export</i>	<i>PDF</i>
	<i>File</i>	<i>Export</i>	<i>Router</i>
	<i>File</i>	<i>Export</i>	<i>Sub-Drawing</i>
	<i>File</i>	<i>Export</i>	<i>Parameters</i>
	<i>File</i>	<i>Export</i>	<i>Libraries</i>
	<i>File</i>	<i>Export</i>	<i>Annotations</i>
	<i>File</i>	<i>Export</i>	<i>Placement</i>
	<i>File</i>	<i>Export</i>	<i>Downrev design</i>
	<i>File</i>	<i>Export</i>	<i>Strip design</i>
	<i>Tools</i>	<i>Reports</i>	
<i>Logic</i>			
	<i>File</i>	<i>Export</i>	<i>Logic/Netlist</i>
	<i>File</i>	<i>Export</i>	<i>Netlist w/Properties</i>
	<i>File</i>	<i>Export</i>	<i>Symbol Spreadsheet</i>
<i>Constraints</i>			
	<i>File</i>	<i>Export</i>	<i>Techfile</i>
	<i>File</i>	<i>Export</i>	<i>Pin delay</i>

<i>MACD/ECAD</i>			
	<i>File</i>	<i>Export</i>	<i>IPF</i>
	<i>File</i>	<i>Export</i>	<i>DXF</i>
	<i>File</i>	<i>Export</i>	<i>IDF</i>
	<i>File</i>	<i>Export</i>	<i>IDX</i>

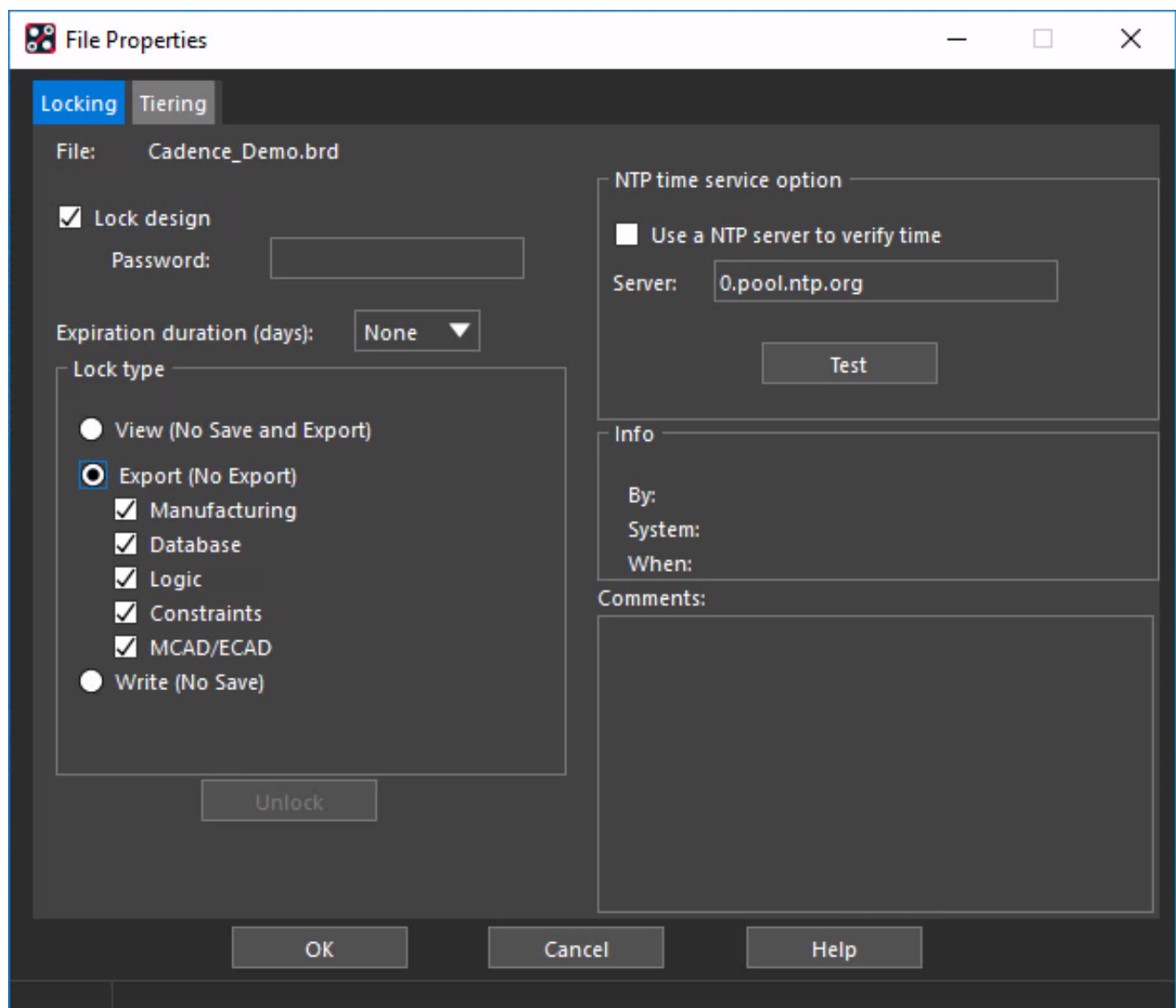
Related Topics

- [Unlocking Database Files](#)

Locking Database Files

To protect your design, you can lock specific aspects of the database so they cannot be changed by others. This enables you to control your design data by choosing which Export options you want to lock. Once locked, that particular Save and Export functionality is disabled if you share your design with others.

1. Choose *File – Save* to ensure any unsaved design work has been saved.
2. Choose *File – Properties*.
The File Properties dialog box opens, and you can choose the *Locking* tab.



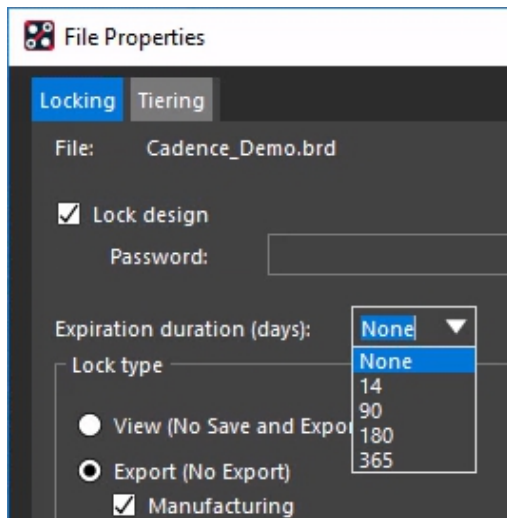
3. Check *Lock design*.

The fields and check box option become active.

4. Enter a password. It may contain a maximum of 20 alphanumeric characters. Invalid characters are: spaces, backslashes (\), and dashes (-). Passwords are case-sensitive.

❗ It is extremely important that you keep a record of any passwords used to lock databases. Cadence does not support the recovery of databases in a locked state due to forgotten passwords.

5. Perform any of these optional actions:
 - a. Choose *Expiration duration (days)* from the drop-down list.



- b. Check one of the lock modes. This option prohibits the export of design data on *View* lock and *Export* lock databases. For *Write* lock databases only prohibits database saves but allows the export of design data.
6. Choose options for locking the database and click *OK*.

A dialog box opens and prompts you to confirm the password. The locked database is saved and becomes active database. The database file name gets automatically updated and assign a prefix based on the selected lock mode as follows:

 - View Lock: <design_name>_view_locked
 - Export Lock: <design_name>_export_locked
 - Write Lock: <design_name>_write_locked

The original database remains unchanged and available on the disk.

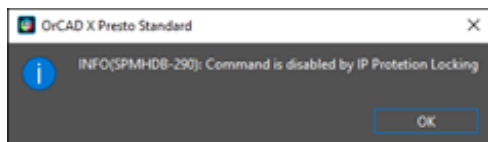
7. Check *Use a NTP server to verify time*.

The *Server* field become active.

8. Specify the server name.

9. Enter additional comments.

When locked, other users will be able to *view* your design, but receive an error message like the following, when attempting to alter and save, or to export protected functionality, depending on your lock mode settings.



Related Topics

- [file_property](#)

Unlocking Database Files

Save the database to a new name and remove the appended lock mode prefix that was added during database locking.

1. Choose *File – Properties*.

The File Properties dialog box opens.

2. Perform the appropriate action:

To unlock a database without password protection:

- a. Click *Unlock*.

A confirmer appears stating that database has been locked.

- b. Click *OK* or *Cancel*.

The dialog box closes and the database is now unlocked.

- c. Choose *File – Save*, to save the design to a new name.

To unlock a database with password protection:

- a. Click *Unlock*.

The password window opens.

- b. Enter the password, and click *OK*.

If you enter an incorrect password, an error message is displayed. Click *OK* to re-enter the password.

If the password is correct, the password window closes and a confirmer appears stating that database has been locked.

- c. Click *OK* or *Cancel*.

The dialog box closes.

- d. Choose *File – Save*, to save the design to a new name.

Related Topics

- [file_property](#)
- [File Properties Dialog Box](#)

file_register

Displays a list of registered file extensions.

Registered File Extensions Dialog Box

This text display dialog box lists registered file extensions.

<i>File – Save As</i>	Saves the information in a text file. When you see this command, you are prompted for a file name and the program appends the <code>.txt</code> extension.
<i>Close</i>	Dismisses the window.

file_unregister

Removes registered files of the specified extension.

Running File Register Command

1. Type `file_unregister <.extension>` at the console window prompt.

filemgr

Displays your current working directory.

fill_ipf

Batch command on UNIX workstations that fills lines segment by segment by generating an even number of passes for each segment. The first two passes fill the external contour of the segment and round its ends in the same way lines are drawn by a photoplotter using a circular aperture. This technique produces lines with good definition of the corners, especially when the line thickness requires many passes of the pen.

After the first two passes, other passes are generated, if required, to fill the internal space of the segments. Arcs with non-zero width are filled with multiple arc passes the same way line segments are filled, except that the ends of the arcs are not rounded.

For additional information, see Plotting in the *Allegro User Guide: Preparing Manufacturing Data*.

Prerequisites

Before executing the command:

- Create the `plot_ipf` file with the classes and subclasses to be plotted using `create plot`.
- Ensure the command text file, `fill_ipf.cmd`, which controls processing, exists in the directory where you run `fill_ipf`.

Syntax

```
fill_ipf [input_IPF] [output_IPF] [-s scale_factor]
```

input_IPF	Specifies the name of the IPF file generated by the editor (the <code>.plt</code> extension is automatically added).
output_IPF	Specifies the name of the IPF file that is generated as a result of executing <code>fill_ipf</code> (the <code>.plt</code> extension is automatically added).
-s scale_factor	Is an optional scale factor; the default is 1.0.

Related Topics

- [Running fill_ipf Command](#)

Running fill_ipf Command

1. Run the `fill_ipf` command from an operating-system prompt, after you create the IPF file with `create plot`.
2. Run `allegro_plot` on the output generated during `fill_ipf` processing.
3. Specify a scale factor of 1 and no fill options.

Related Topics

- [fill_ipf](#)

film area

Displays the Film Area Geometry Report.

Access Using

Menu Path: *Tools – Reports – Film Area*

```
film area [n] [-f <fil name...>] [output file name]
```

n	Skips the calculations and displays just the film data.
f	Specifies a list of legal film names to process.
output file name	Specifies the name of the output file. If you do not provide a name, the data is displayed in a text view window.

film param

Displays the *Artwork Control Form* dialog box, from which you can set film options and generate photoplot film files, load gerber data, and create artwork. You can also set general artwork parameters and edit aperture wheels.

When dynamic shapes are out-of-date, *Dynamic Shapes Need Updating...* appears on the *Artwork Control Form* dialog box.

If you attempt to use the *Create Artwork* button on the *Artwork Control Form* dialog box, an error message appears: "Dynamic Shapes are out of date, please update them." Click *Dynamic Shapes Need Updating...* to open the *Status* tab of the *Status* dialog box, which becomes active, blocking any use of the *Artwork Control Form* dialog box until you update dynamic shapes and/or DRCs before proceeding with artwork.

Related Topics

- [Creating Film Records for a Gerber Data File](#)
- [Suppressing the Shapefill Algorithm in Negative Artwork](#)

Artwork Control Dialog Box


Access Usingenvd

Menu Path: *Manufacture – Artwork*

The *Artwork Control Form* comprises two tabs: *Film Control* and *General Parameters*.

Film Control Tab

This tab lists film layers with check boxes to the left of the names. Film control records define the manufacturing (artwork) files created and the classes and subclasses that each manufacturing file includes. By default, a film control record exists for each ETCH subclass (layer) of the board. Each of these records has the ETCH, PIN, and VIA classes included in the film control record for the corresponding etch subclass.

 Wire bonds (the entire pattern) can be added to films as clines by adding the WIRE subclass to the Artwork Control Form. To do so, choose *Manufacture – Artwork*. In the Artwork Control Form, under available films, right-click on any of the listed subclasses and choose *Add*. In the Subclass Selection dialog box, check *WIRE* under *CONDUCTOR*. DXF Export supports selection by wire profile, and is the preferred method for wire bond profile documentation.

<i>Available Films</i>	Choose a record by clicking the check box. Expand or collapse the layers in a record by clicking the plus or minus sign (+ or -) to the left of the record. If the database does not contain any film control records, the tool creates a default film record for each ETCH class. This record consists of etch, pins, and vias.
<i>Select All</i>	Lets you choose all the available films.
<i>Add</i>	Click to launch the <i>Select Film File to Add</i> dialog box from which you can choose a previously created film record.
<i>Replace</i>	Click to substitute another film record for the currently selected record.
<i>Check Database Before Artwork</i>	Click the check box to verify the integrity of a drawing database prior to generating artwork by invoking the Dbdoctor database-checking program. If Dbdoctor detects errors in the database, the tool does not generate artwork.

<i>Create Artwork</i>	Generates artwork for each film record you checked. A message appears on the dialog box informing you of successful completion of the process. If an error occurs, the message informs you to look at the output file.
-----------------------	--

Film Options

<i>Film name</i>	Displays the name of the film record to be edited. You cannot edit this field. You must change the name in the <i>Available Films</i> section on the Film Control tab of the Artwork Control Form dialog box by first highlighting the film name and then clicking it. You can also perform the renaming task within a script.
<i>PDF Sequence</i>	Specifies the sequence number of films in the PDF output. You can override the order of films in PDF output. If two films are assigned a same sequence number, they are sorted in the alphabetical order.
<i>Rotation</i>	Specifies the rotation of the plotted film image. A drop-down list displays the angle of rotation. Choices are 0, 90, 180, and 270. The default is 0.
<i>Offset X Y</i>	Specifies the x and y offset to add to each photoplot coordinate. If you enter positive x and y offsets, all photoplotted lines shift in the positive direction on the film. The default is 0.
<i>Undefined line width</i>	Determines the width of any line that is undefined.
<i>Shape bounding box</i>	Applies to negative film. Vector artwork uses this value to extend the shape fill for a negative layer beyond the board outline by that value. Raster artwork for negative layers adds fill from the edge of the shapes to the photoplot outline. If the shape bounding box value is positive, the fill extends that distance beyond the photoplot outline. When no photoplot outline exists, raster format draws fill up to the board geometry outline as it does with vector formats. If the shape bounding box value is positive, the fill extends that distance beyond the board geometry outline.
<i>Plot mode</i>	Specifies whether the photoplot output is positive or negative. The default is positive.
<i>Film mirrored</i>	Specifies whether the photoplot output is to be mirrored. The default is not mirrored.

<i>Full contact thermal-reliefs</i>	Applies to negative film. When you choose this option, a pin or via that is connected to a shape uses no flash, which causes a solid mass of copper to cover the pad. If you do not choose this option, a pin or via connected to a shape uses a thermal-relief flash. The default is no selection.
<i>Suppress unconnected pads</i>	Specifies that the pads of pins and vias that have no connection to a connect line or shape in a Gerber data file are not plotted. This option applies only to internal layers and to pins whose padstack has the suppression of unconnected internal pads enabled. Selecting this option also suppresses donut antipads in raster-based negative artwork. When disabled, for negative plane layers, donut pads generate based on the regular and antipad definitions in the padstacks. In this case, padstacks must be set up so that the regular pad is smaller than the antipad so an annular ring, suitable for manufacturing, is formed. Caution: If the value of regular pads is not less than the antipad value in the padstack, the donut pad will be missing in the artwork file. Enabling the <i>Dynamic unused pads suppression</i> option, available by running <i>Setup – Cross Section</i> (xsection command) enables this option for all films; suppression occurs as required for just those films needing it, despite all films displaying as checked. Otherwise this option is greyed out. With the <i>Dynamic unused pads suppression</i> option disabled, this option's functionality remains unchanged. Unconnected outer pads of the vias on internal layers are never suppressed.
<i>Draw missing pad apertures</i>	Choose this option to allow vector artwork to use line apertures to outline and fill pads with no matching aperture in the <code>art_aper.txt</code> file. Not selecting this option means that such pads are not drawn. (A warning is written to the <code>photoplot.log</code> file if this is the case). The default is no selection.
<i>Use aperture rotation</i>	Specifies whether to use the aperture rotation. The default is no selection.
<i>Suppress shape fill</i>	Available for Gerber 6x00 and Gerber 4x00 only. Specifies that the area outside the shapes and all voids is not to be filled on a negative film. You must replace the filled areas with separation lines before running the <code>artwork</code> command. This option is useful for negative nested shapes. See the section, Suppressing the Shapefill Algorithm in Negative Artwork .
<i>Vector-based pad behavior</i>	Specifies that raster artwork uses vector-based (Gerber) behavior to determine which type of pad to flash.

<i>Draw holes only</i>	Choose this option to draw holes in artwork. This option is only enabled when pins and/or vias and no conductor layers are set up in the film record. For positive photoplot this option generates raster artwork output as shapes that are equal to the size of holes. For negative photoplot this option generates raster artwork output as shapes with voids which are equal to the size of the holes.
------------------------	---

General Parameters Tab

The General Parameters tab shows different parameters and defaults for each photoplotter model type, depending on which type you choose. The selection that you make in the Device Type section in the upper left-hand corner determines the available controls for that plotter and displays only those options in the dialog box.

<i>Device Type</i>	Lets you specify the photoplotter model (Gerber 6x00, Gerber 4x00, Gerber RS274X, Barco DPF, or McDonald Dettwiler (MDA)) for which the tool writes artwork data files. Choose one photoplotter at a time. To display parameters that apply to a different photoplotter mode, click on another model.
<i>Film Size Limits</i>	Enables you to specify the dimensions of the film used by the photoplotter. This parameter prevents the creation of plot commands with dimensions that are larger than the actual film in the plotter when you run the <code>artwork</code> command. If the editor finds any elements that plots outside the boundaries given by Max X and Max Y, it writes the data to the artwork file anyway, and also writes a warning to the log file.
<i>Coordinate Type</i>	Applies only to Gerber 6x00 and Gerber 4x00 device types. Enables you to specify whether the photoplot coordinates are the absolute distance from the drawing origin (<i>Absolute</i>) or the relative distance from the last coordinate (<i>Incremental</i>).
<i>Error Action</i>	Specifies the action when an error is found —such as an undefined aperture— while processing the artwork files. The choices are: <i>Abort Film</i> : Discards the data about the film file in error, but continues processing any other films still on the list. <i>Abort All</i> : Aborts the entire process; no additional artwork files are created. In either selection, errors are written to the log file and its action recorded.

Format	<p>Lets you specify the number of integer places and the number of decimal places in the output coordinate fields. The two format fields are: <i>Integer Places</i>: Specify a number between 0 and 5. <i>Decimal Places</i>: Specify a number between 0 and 5. The format refers to either English (inch) or metric (millimeter) and should be set based on the output unit. For example:</p> <table><tr><th>Database Unit</th><th>Accuracy</th><th>=</th><th>Format</th></tr><tr><td>MILS</td><td>0</td><td>=</td><td>2.3</td></tr><tr><td>MILS</td><td>1</td><td>=</td><td>2.4</td></tr><tr><td>MILS</td><td>2</td><td>=</td><td>2.5</td></tr><tr><td>MILS</td><td>3</td><td>=</td><td>2.6 not available</td></tr><tr><td>MILS</td><td>4</td><td>=</td><td>2.7 not available</td></tr><tr><td>INCH</td><td>3</td><td>=</td><td>2.3</td></tr><tr><td>INCH</td><td>4</td><td>=</td><td>2.4</td></tr></table> <p>Roundoff occurs to artwork output data generated from boards created at accuracies higher than the number 0.5 decimal equivalent. Cadence recommends designing data at or below the accuracy level the fab vendor supports.</p>	Database Unit	Accuracy	=	Format	MILS	0	=	2.3	MILS	1	=	2.4	MILS	2	=	2.5	MILS	3	=	2.6 not available	MILS	4	=	2.7 not available	INCH	3	=	2.3	INCH	4	=	2.4
Database Unit	Accuracy	=	Format																														
MILS	0	=	2.3																														
MILS	1	=	2.4																														
MILS	2	=	2.5																														
MILS	3	=	2.6 not available																														
MILS	4	=	2.7 not available																														
INCH	3	=	2.3																														
INCH	4	=	2.4																														
Output Options	These miscellaneous options do not apply to Gerber RS-274X, Barco DPF, or MDA device types.																																
Optimize Data	Applies only to Gerber 6x00 and Gerber 4x00 device types. Sorts coordinates to minimize photohead travel time. Laser plotters optimize the data at plot time, making this step unnecessary for artwork. This is the default setting.																																
Use ‘G’ Codes	Applies only to the Gerber 4x00 device type. Specifies G codes in the Gerber data. Gerber data uses G codes to describe an upcoming process, for example, prepare to receive x, y coordinates, prepare to choose aperture, or prepare to flash aperture. Gerber 4x00 photoplotters support G codes.																																
Suppress	Does not apply to the Barco DPF device type. Controls whether the tool writes leading or trailing zeroes or equal coordinates in the Gerber data file.																																
Leading Zeroes	Suppresses the writing of leading zeroes for coordinates in the Gerber data file. This is the default setting.																																
Trailing Zeroes	Suppresses writing trailing zeroes for coordinates in the Gerber data file. Note: You can suppress either leading or trailing zeroes, or you can suppress neither leading or trailing zeroes, but you cannot suppress both leading and trailing zeroes.																																

<i>Equal Coordinates</i>	Suppresses the writing of duplicate coordinates in the Gerber data file. This is the default setting. Gerber photoplotters are modal, which means that they retain old values until they read a new one of the same type. This means that coordinates with the same value do not need to be written more than once. This option reduces the size of the Gerber data file. This is the default setting.
<i>Output Units</i>	Applies only to Gerber RS274X device types. Lets you specify the output units as inches, millimeters, or mils.
<i>Max Apertures per Wheel</i>	Applies only to Gerber 6x00 and Gerber 4x00 device types. Lets you specify the maximum number of apertures that the photoplotter wheel uses. Enter a value between 1 and 99. Photoplotter wheels have a maximum number of apertures. If your layout uses more than the number specified in <i>Max Apertures Per Wheel</i> , the tool writes a warning to the log file.
<i>Global Film Filename Affixes</i>	
<i>Prefix</i>	<p>Adds a user-defined, case-sensitive string before generated film filenames on a board-level basis, allowing a maximum 512-character filename, such as a part or revision number, which may be useful for larger boards with many layers and numerous artwork films as a result. For example, if a board has a project number of CDS1234, adding a prefix of CDS1234_ creates artwork in the following format: CDS1234_TOP.art CDS1234_BOTTOM.art CDS1234_SMASKT.art CDS1234_SMASKB.art Names must be legal filenames and cannot contain directory names. Although Allegro permits filename affixes of 512 characters, many operating systems limit filenames to 256 characters (including extensions). Consequently, Cadence recommends film filenames (affixes plus filename plus extensions) be less than 256 characters. Note: You can also change the default file extension of .art for artwork film filenames by setting the <code>ext_artwork</code> environment variable in the User Preferences Editor, available by choosing <i>Setup – User Preferences</i> (envded command).</p>

Suffix	<p>Appends a user-defined, case-sensitive string after generated film filenames on a board-level basis, allowing a maximum 512-character filename, such as a part or revision number, which may be useful for larger boards with many layers and numerous artwork films as a result. For example, if a board has a revision number of Rev-3, adding a suffix of <code>_Rev-3</code> creates artwork in the following format: <code>TOP_Rev-3.art</code> <code>BOTTOM_Rev-3.art</code> <code>SMASKT_Rev-3.art</code> <code>SMASKB_Rev-3.art</code>. Names must be legal filenames and cannot contain directory names. Although Allegro permits filename affixes of 512 characters, many operating systems limit filenames to 256 characters (including extensions). Consequently, Cadence recommends film filenames (affixes plus filename plus extensions) be less than 256 characters. Note: You can also change the default file extension of <code>.art</code> for artwork film filenames by setting the <code>ext_artwork</code> environment variable.</p>
<i>Continue with Undefined Apertures</i>	<p>Available for Gerber RS-274X, Barco DPF, and MDA device types. Check to continue to generate the Gerber data file when a definition for a flash aperture in the padstack is missing. Messages about the undefined apertures are written to the log file. If you do not check this box, the process stops when an aperture definition is not found. <i>On:</i> Continues to generate the Gerber data file and write messages about undefined apertures in the log file <i>Off:</i> Stops generating the Gerber data file when it cannot find an aperture.</p>
<i>Scale Factor for Output</i>	<p>Value causes all entries in the artwork file to be scaled vertically and horizontally. If you use the default of 1.0000, no scaling occurs. If you enter a different value, the artwork output is scaled and a recommended aperture table is added to the <code>photoplot.log</code> file. For example, a value of 0.5 reduces each artwork entry by 50 percent; a value of 2.0000 increases each entry by 100 percent. The field accepts a total of eight characters, including the decimal point. The maximum number of decimal places is four. Add the recommendation in the <code>photoplot.log</code> file to the aperture table that accompanies the artwork file to manufacturing. This assures that the scaled apertures have the correct width and size.</p>
<i>OK</i>	Saves the settings and closes the dialog box.
<i>Cancel</i>	Closes the dialog box.
<i>Apertures</i>	Click to display the <i>Edit Aperture Wheels</i> dialog box.
<i>Viewlog</i>	<p>Displays the <code>photoplot_out.log</code> that contains messages generated after you create artwork, and is available only after you have done so. This log limits the 0 width line warnings to a maximum of 2 unless you set the <code>artwork_allwarnings</code> environment variable in your local <code>.env</code> file.</p>

Film Record Pop-up Menu

The Film Record pop-up menu appears when you right-click a film record. It includes the following options:

<i>Display for Visibility</i>	Displays the visibility updates in the <i>Color</i> form for class/subclass. This menu disables all subclasses, and then enables the subclasses that you select in the film record.
<i>Display for Artwork Check</i>	This menu performs the same function as <i>Display for Visibility</i> . In addition it changes the settings in the Design Parameter Editor dialog box for the Enhanced display modes in Display tab: Disables modes for the thermal pads and holes, if they were enabled. Enables the Filled pads and Connect line endcaps, if they were disabled.
<i>Add</i>	Opens a dialog box and adds a new film record after the selected film. The list of classes and subclasses contains those that appear in the current design window.
<i>Cut</i>	Deletes the selected film layer. Your design must always contain at least one film layer.
<i>Undo Cut</i>	<i>Undoes the cut action that you just performed.</i>
<i>Copy</i>	Adds a copy of the selected layer directly beneath the layer. The copy is named "Copy_of_".
<i>Save</i>	Saves changes made to a layer during the current session. When you reload the board, the film record is re-created.
<i>Save All Checked</i>	Saves multiple films to an external file.
<i>Match Display</i>	Deletes all class and subclass items from the film and replaces them with the list of classes and subclasses that appear in the current window.
<i>Select All</i>	Lets you choose all the available films.
<i>Deselect All</i>	Deselects the films you have chosen.

Layer Pop-up Menu

The Layer pop-up menu appears when you right-click a class/subclass in a film record. This pop-up box has the following options:

<i>Add</i>	Displays the Subclass Selection window. You can choose one or more subclasses to add to the selected film record.
<i>Cut</i>	Deletes the class or subclass.

Related Topics

- [Suppressing the Shapefill Algorithm in Negative Artwork](#)

Creating Film Records for a Gerber Data File

To produce artwork data files, the editor reads the film control records that you create in a layout. It reads these records to determine the following:

- The number of artwork files to produce
- The names it assigns to the artwork data files
- The classes and subclasses to include in each artwork data fill

1. Run the `color192` command or *Display – Color/Visibility* to display the Color dialog box.
2. In Board Geometry, Package Geometry, Manufacturing, Stack-Up, Components, and Areas, turn off all the classes and subclasses and then choose the classes and subclasses that you want included in the Gerber data.
3. Run the `film param` command or *Manufacture – Artwork*.
When the Artwork Control dialog box initially opens, it reads the cross-section and auto-generates one film record for each etch subclass. The record consists of etch, pins, and vias. Once you click *OK* in this dialog box, the editor does not automatically update the list again.
4. To add a new record, right-click one of the film records listed in the *Available Films* list.
5. Choose *Add* from the pop-up menu.
6. In the *New Film* field of the dialog box that appears, enter a new film name for the Gerber data file and then click *OK*.
7. Repeat steps 4 to 6 for any other film records that you want to create.
You can manipulate the film records and layers by right-clicking the record or layer and choosing options from the pop-up box.
8. Complete the Film Control tab of the Artwork Control Form dialog box.
9. Choose the General Parameters tab and set the photoplotter model type and associated parameters.
10. When you have completed setting all the parameters in both the Film Options tab and the General Parameters tab of the Artwork Control dialog box, do one of the following:
 - Click *Add* to add a previously created film record text file.
 - Click *Create Artwork* to generate artwork for the films you have selected.
 - Click *OK* to close the form.

Related Topics

- [film param](#)

Suppressing the Shapefill Algorithm in Negative Artwork

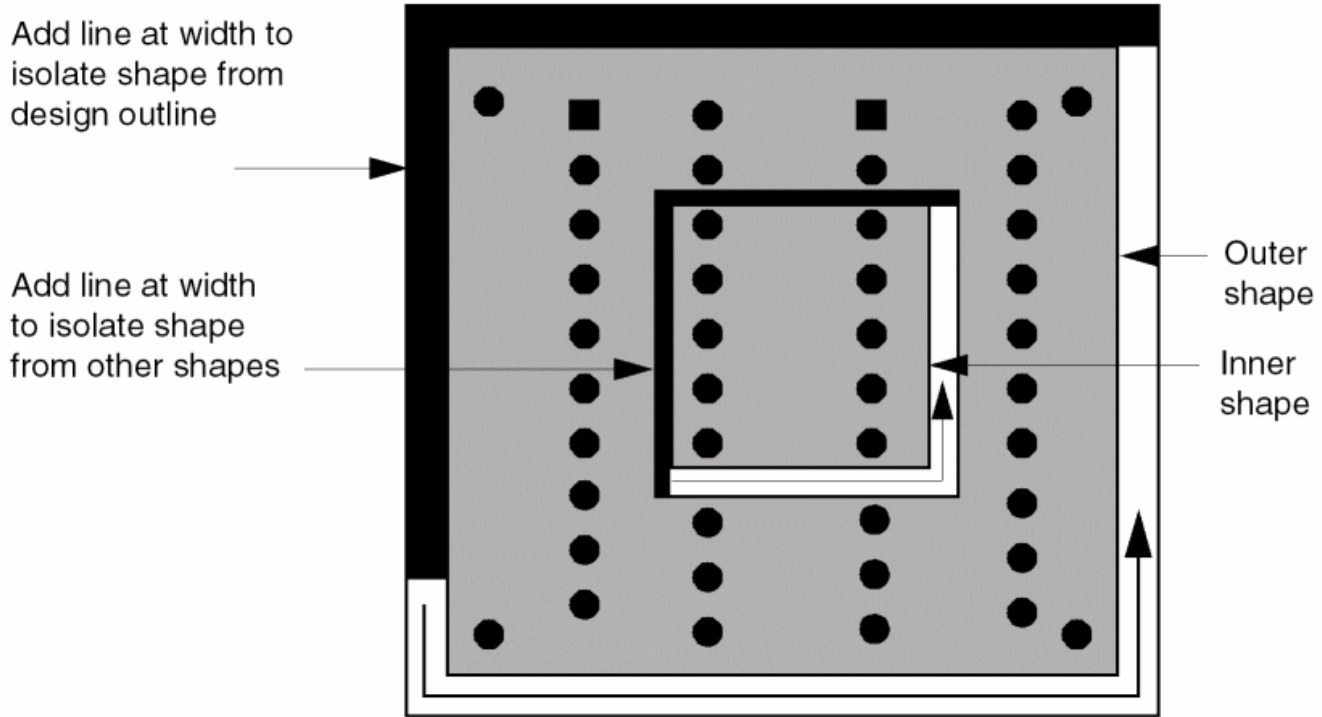
When you suppress the shapefill algorithm that fills the background and voids, replace the filled areas with separashows the separation lines before you run the `artwork` command.

1. Create a new subclass for the separation lines in any non-ETCH class or use the ANTI-ETCH subclasses.
2. Draw the separation lines.
These lines must separate each plane from the design outline and from other planes.
3. Add the subclass for the separation lines to the film control record for the layer.
4. Choose *Manufacture – Artwork* or run the `film param` command and check the *Suppress Shape Fill* option for the film record in the Film Options tab of the Artwork Control Form dialog box.

When you create artwork, the only graphical elements in the artwork data file are pins that the photoplotter flashes as thermal reliefs or antipads and the separation lines.

The following figure shows the separation lines that must be added around shapes in a design when you suppress the shapefill algorithm

Figure -1 Suppressing the Shapefill Algorithm



Related Topics

- [film param](#)
- [Artwork Control Dialog Box](#)

film res

Runs the Thick/Thin Film Resistor Synthesizer. The Resistor Synthesizer reads the film_res.rcf file (film resistor command file) and generates the thick- or thin-film resistors accordingly. You can specify an alternate command file to be used (instead of the film_res.rcf file) by the Resistor Synthesizer in the Thick/Thin Film Resistor Generator Control dialog box. Running the resistor generates the film_res.log file.

For additional information, see Paste Resistor Symbols in the *Allegro X Advanced Package Designer* *User Guide: Placing the Elements*.

Prerequisites

Before you run the paste resistor command you need to:

- have the correct Input files for running the Thick/Thin-Film Resistor Synthesizer
- specify the corresponding input controls in the film resistor control file
- have certain Package output files in the schematic directory if you are using Design Entry HDL or System Connectivity Manager

The following table shows the input files required by the Resistor Synthesizer:

Input File Prerequisites

Input File	Film Resistor Command Directive
Film resistor control file	Not applicable
Part properties tables (if using Design Entry HDL or System Connectivity Manager)	part_table_file
Resistor specification file (if not using Design Entry HDL or System Connectivity Manager)	resistor_specs

Scale factor file (if not using the default scale factors)	scale_factor_file
--	-------------------

You must also be sure that your film resistor control file contains the directives that specify the type of processing, output, and resistor and ink controls you want. Some of the directives that you specify in the control file may require additional information to be defined before running the `film_res` command. For example, you may have to attach certain properties to resistors, and create a dummy padstack.

Design Entry HDL or System Connectivity Manager Prerequisites

If you use Design Entry HDL or System Connectivity Manager you must also have the following files (as created by the Compiler and Packager-XL) in the schematic directory:

- `pstxnet.dat`
- `pstxpri.dat`
- `pstchip.dat`

Note: The locations of these files varies, depending on which logic import/export mode (Design Entry HDL or System Connectivity Manager) you are using.

The Resistor Synthesizer reads these files to generate a resistor of a certain shape (usually larger than is needed so that you or the `trim_check` directive can trim the resistor for manufacturing purposes).

Also, if you are generating thin-film resistor symbols, be sure that resistor instances in the schematic have the `RES_TYPE=THIN` property attached. The Resistor Synthesizer assumes by default that resistor instances in the schematic are thick film resistors.

Related Topics

- [Running the film_res command](#)
- [Reviewing the film_res.log File](#)

Thick/Thin Film Resistor Generator Controls Dialog Box

Access Using

Menu Path: *File – Import – Paste Resistor*

Use this dialog box to generate thick- or thin-film resistor symbols.

<i>Resistor Control File</i>	Indicates the name of the file to be used by the Resistor Generator program. The tool now uses <code>.rcf</code> files that are not in microns and do not match the design units of the active drawing into which they are being imported.
<i>Text Block for Symbol text</i>	Indicates the size of the text for the generated symbols.
<i>Browse</i>	Displays an Open browser window so you can enter the Resistor Control File name.
<i>OK</i>	Starts the Resistor Generator program.
<i>Close</i>	Closes the Thick/Thin Film Resistor Generator Controls dialog box without running the Resistor Generator program.

Related Topics

- [Reviewing the film_res.log File](#)

Running the film res command

1. Run `film res`.

If there are unsaved changes in the current design you are prompted to save the design. If you click *Yes*, the current design is saved in `designname_tmp.mcm`, in your working directory.

2. Enter the name of the resistor control file (`film_res.rcf`) in the *Resistor Control File* field. You can click the *Browse* button to locate the correct file.

3. Choose the size for the text display in the Text Block for *Symbol* text field.

4. Click *OK*.

A warning message is displayed if the resistor control file you specify cannot be located. If there are errors in the control file you need to correct them and rerun the *film res* command. Errors are listed in the `film_res.log` file. You can view the current log file by running the `viewlog` command or by choosing *File – File Viewer* in the menu bar and selecting the `film_res.log` file.

The resistor symbols, padstacks, design cross section data is generated, depending on the output directives specified in your film resistor control file.

5. Run `netin param` to update the design with the new netlist.
Cadence recommends this process as alternate symbols may be generated for the resistors.
6. Place the generated resistor symbols on your design using the standard commands.

Related Topics

- [film res](#)

Reviewing the film_res.log File

The film_res.log file lists the details of the thick/thin-film resistor generation process. For example, the log lists:

- Packager-XL files used (if you use Design Entry HDL or System Connectivity Manager).
- The generated resistors.
- A summary of the command directives used.
- Design information, such as number of components, number of nets, and number of pins.
- Any errors or warnings.

You can view the log file by choosing the *File – Viewlog* command.

Related Topics

- [film res](#)
- [Thick/Thin Film Resistor Generator Controls Dialog Box](#)

find_by_name

The `find_by_name` command works in conjunction with an active command and is used when you want to find a design element by name or property.

Related Topics

- [Running the Find By Name Command](#)

Find by Name/Property Dialog Box


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

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

Double clicking an object in either the Available Object list or the Selected Object list results in the object being moved to the other column.

When you click the *Apply* button, the command:


- Selects the elements to be acted upon by the active command; for example, `property edit`.

 Different commands result in specific behaviors. For example, if you are running `property edit`, the Edit Properties and Show Properties dialog boxes are displayed; if you running move, the selected elements are selected for editing.

- Displays the location of the element(s) in the *WorldView* area of the UI
- Highlights the selected element(s) in the design area of the UI.

Running the Find By Name Command

1. In an active command, run `find_by_name`.
The Find by Name/Property dialog box appears.
2. Choose the object type from the *Object Type* drop-down list.
The list of available object types in your design appears in the list box.

 To filter the list items, type in one or more characters before or after the asterisk (*) in the *Name Filter* field.

3. Click *Apply* to complete the active command and keep the dialog box displayed or *OK* to complete the active command and return to an idle state.

Related Topics

- [find_by_name](#)

find_by_query

The `find_by_query` command lets you find objects that meet a set of pre-defined search criteria. This command filters different types of objects (nets, clines, shapes, voids, and so on) and provides accurate search results in a tabular format.

The command is invoked by clicking the *Find by Query* button, present at the bottom of the *Find* pane.

Once you select the objects, you can also access relevant application mode commands using pop-up menus.

For more information, see [Finding Objects by Query](#) in the Allegro User Guide: Getting Started with Physical Design.

Related Topics

- [Find Setting Dialog Box](#)
- [Procedure for Editing Filter Settings](#)
- [Procedure for Running a Query](#)

Find by Query Dialog Box

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

<i>Objects</i>	Displays a list of different types of objects. For example, components, clines, text shapes, symbols, and so on.
<i>Configure</i>	Controls the display of objects in the <i>Objects</i> tab. De-selecting a check box will stop showing that object in the <i>Objects</i> tab.

<i>Fields</i>	Displays selected objects and their attributes. To add objects to the <i>Fields</i> , either double-click or drag and drop them from the <i>Objects</i> tab. To remove objects from the <i>Fields</i> , use drag and drop method.
<i>Filters</i>	Display filtered objects and set the values of their attributes. To add objects to the <i>Filters</i> , either double-click use the arrow button or drag and drop them from the <i>Fields</i> tab. You can remove objects from the <i>Filters</i> by using the arrow button or the drag and drop method. The attributes of the objects are added with default value "*" . You can double-click to specify the values by setting values in the <i>Filter Setting</i> dialog box.
->	Add the selected object from the <i>Fields</i> to the <i>Filters</i> section.
<-	Removes the selected object from the <i>Filters</i> section.
AND	Controls the filter operations. If enabled, the filter objects are added under node of AND. By default, this option is enabled.
OR	Controls the filter operations. If enabled, the filter objects are added under node of OR.
Matching Objects	Displays the total number of matching objects depending on the criterion defined in the <i>Filters</i> section and display the results in a table format. A default list of attributes for each type of object is always displayed in the table. For example, <i>Text</i> objects attributes <i>Class Name</i> , <i>Subclass Name</i> , and <i>Text Contents</i> are always displayed in the table.
Defer Selection	Suspends the selection of objects when clicked in the <i>Matching Objects</i> table for canvas selection or as an input to a command until the <i>Select</i> button is clicked.
Select	Available when <i>Defer Selection</i> is enabled. Redirects selection of objects from the <i>Matching Objects</i> table for canvas selection or to a command as an input.
Display Search Result in Table	Click to display the <i>Matching Objects</i> results in a table format.
Configure Search Result Table	Click to control the display of object attributes for the object types in the <i>Matching Objects</i> result table.

Save Query	Saves the current query settings in a file (.qfnd). You can set <code>miscpath</code> env variable in the <i>User Preferences Editor</i> dialog box to specify the location for saving queries.
Load Query	Loads previously saved query files.
Clear Query	Removes the settings for current query.
Rerun Query	Updates the <i>Matching Objects</i> results for the current query.
Export Result	Exports the query results in an XML or CSV format.
Close	Displays <i>Close Dialog</i> for saving query.
Cancel	Closes the dialog box without saving any changes.

Related Topics

- [Procedure for Editing Filter Settings](#)
- [Procedure for Running a Query](#)


Find Setting Dialog Box

Use this dialog box to set up filter settings.

<i>Operator</i>	Specifies the logical expression. By default, <i>equal to</i> operator is set.
<i>Value</i>	Specify the value for the object attribute.
<i>Selection</i>	Displays the available and selected values for an object attribute.
<i>Available Values</i>	Displays a list of all the available values for the selected object attribute in the design. You can also filter the available values.
->	Adds all the selected values from the <i>Available Values</i> to the <i>Selected Values</i> list.
<-	Removes all the selected values from the <i>Selected Values</i> list.
>>	Adds all the available values into the selected values list.
<<	Removes all the available values from the selected values list.
Selected Values	Displays a list of all the selected values.
OK	Applies the changes and closes the dialog box.
Cancel	Closes the dialog box without applying any changes.

When you click the search result in the *Matching Objects* table, the command:

- Selects the objects to be acted upon by the active command; for example, `property edit`.

 Different commands result in specific behaviors. For example, if you are running `property edit`, the *Edit Properties* and *Show Properties* dialog boxes are displayed; if you running `move`, the selected objects are selected for editing.

- Displays the location of the object(s) in the *WorldView* area of the UI
- Highlights the selected object(s) in the design area of the UI.

In the result table, select any row right-click and choose *Select All* to highlight all the objects at a time.

Related Topics

- [find_by_query](#)
- [Procedure for Running a Query](#)

Procedure for Editing Filter Settings

1. Double-click the object row in the *Filters* section.
The *Filter Setting* dialog box appears.
2. In the *Operator* field, choose a logical expression from the drop-down list.
3. In the *Values* field, specify the value.
4. Optionally, you can choose *values* in the drop-down list of the *Operator* field.
The *Selection* table becomes enabled.
5. Filter the value from the list of *Available Values* list.
6. Use arrow buttons to add/remove values to the *Selected Values* list.
7. Click *OK*.
The value for the object attribute is set and is displayed in the *Filters*.

Related Topics

- [find_by_query](#)
- [Find by Query Dialog Box](#)

Procedure for Running a Query

1. In the command window, type `find_by_name` or click *Find by Query* button from the *Find* filter or in an active command, run `find_by_name`.
The *Find by Query* dialog box appears.
2. Double-click an object in the *Objects* tab.
This results in the object and its attributes being moved to the *Fields* section.
3. Optionally, enable the logical operator *OR* for adding objects under *OR* node in the *Filters*.
4. Choose attributes for an object and double-click to move them into *Filters* . You can also use arrow button to add object attributes.
5. Click an object attribute for specifying the filter setting.
The *Filter Setting* dialog box is displayed.
6. Specify the value(s) and click *OK*.
The *Matching Objects* table displays the results.
7. Click *Save Query* to save the current query in a `.qfnd` file.
The saved query can be run any time by loading it into the UI using *Load Query* button.
8. Click *Rerun Query* to update the results.
9. Click *Close* to complete the command and close the dialog box.

Related Topics

- [find_by_query](#)
- [Find by Query Dialog Box](#)
- [Find Setting Dialog Box](#)

find_control

An internal Cadence engineering command.

findfilter

Displays the Find Filter tab that lets you specify which elements on a design can be selected. The Find Filter command is an option on the pop-up menu in the design window.

findprop

The `findprop` command is used in conjunction with `property edit` to locate an object by property, and with `show property` to display information on the named object.

Related Topics

- [Displaying Information](#)

findprop Dialog Boxes

Depending on which commands you run `findprop` with, the following dialog boxes are displayed:

- Show Element
- Edit Property
- Show Properties

Procedures

Displaying Information

1. Run the `show element` command.
2. Choose the appropriate object types in the Find filter.
3. Type `findprop <property name>` at the console window prompt.
The Show Element display window for the specified property appears.

Selecting an Object for Editing

1. Run the `property edit` command.
2. Choose the appropriate object types in the Find filter.
3. Type `findprop <function designator name>` at the console window prompt.
The Edit Property and Show Properties dialog boxes are displayed.
4. Edit the property. For additional information, see `property edit` in the Allegro PCB and Package Physical Layout Command Reference.

fix

The fix command assigns the FIXED property to elements without requiring the use of the *Edit Property* dialog box. A database element that is "fixed" is restricted from additional modification. For example, mechanically placed components or critical high-speed nets often are fixed to prevent accidental movement or deletion. Fixed nets are not ripped up during auto-routing, nor updated during glossing.

You can free elements for editing by removing the FIXED property with:

- the `fix` command with the `no` argument
- the `unfix` command

Toolbar Icon



You can also quickly edit the FIXED property on elements using the `unfix` command icon.



To unfix all elements in the design, use the Unfix icon, then right click and choose Unfix All from the popup menu.

For more information on the `FIXED` property, see the *Allegro Platform Properties Reference*.

This command functions in a pre-selection use model, in which you choose an element first, then right click and execute the command, if the chosen element contains the FIXED property. Elements ineligible for use with the command generate a warning and are ignored. Valid elements are:

- Symbols

- Nets
- Pins
- Vias
- Clines
- Lines
- Shapes

Related Topics

- [Restricting Elements to Prevent Modification](#)

fix
[no]

no	Removes the FIXED property from specified elements and allows them to be modified.
-----------	--

Restricting Elements to Prevent Modification

1. Hover your cursor over an element or draw a window around the elements you do not want modified. The tool highlights the element and a datatip identifies its name.
2. Right click and choose *Fix* from the popup menu to automatically launch the command. The following message appears in the console window for each chosen element to which the tool added the FIXED property to prevent modification:

Property FIXED added to element <variable>: <variable>.

Related Topics


- [fix](#)

flash_convert

Migrates pre-14.0 .bsm flash symbol files to .fsm files and converts pre-14.0 databases to the flash methodology inaugurated in version 14.0. You can choose to define flash symbols interactively for the database you are currently working in or for one or more designs in a project hierarchy.

When you run flash_convert, the program:

1. Targets all .bsm flash symbols within the referenced design.
2. Converts them.
3. Verifies integrity.
4. Saves .bsm files as .fsm files.

 When creating flash files, an error is generated if anything but flash geometry is encountered in the target .bsm file.

If any errors or warnings occur, they are recorded in the flash_convert.log located in the current working directory.

For more details, see Creating Flash Symbols in the *Allegro X Advanced Package Designer User Guide: Defining and Developing Libraries*.

```
flash_convert [-t] [-b] <filename.brd> <filename.dra> <filename.mcm>
```

-t	Indicates a test run. The database is not converted and converted files are not saved.
-b	Updates only the board name listed on the command line. It does not convert any dependencies.


flipdesign

Use this command to flip the design along the Y-axis on the drawing canvas. It sets the active layer to bottom etch when enabled and to top etch when disabled. Grids do not display when this command is active. The active *Flipboard mode* is indicated in the in the status bar at the bottom of the Allegro X PCB Editor window; and in the title bar, with the design file name suffixed with the flip mode.

To return to normal view, run the flip design command again.

Access Using

- Menu Path: *View – Flipdesign*

- Toolbar Icon: 

flow copy

The `flow copy` command lets you copy the flow and properties of a of a pre-selected source bundle to a selected target bundle in the design.

⚠ The flow width, layers, and properties copied from the source bundle are adjusted to be compatible with the constraints of the target bundle. For example, layers that are not allowed by layer set constraints of target bundle members are removed from the flow layer usage of the target bundle (possibly leaving the bundle with default layer usage).

Access Using

Menu Path: *FlowPlan – Copy Flow*

Right Mouse Button Option

Copy Flow

Copying the Flow of a Bundle

1. In IFP application mode, hover your cursor over a source bundle whose flow you wish to copy. The 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.

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

2. Right-click and choose *Copy Flow* from the menu.
The following message appears in the Command console.


```
"Pick to select a bundle to receive the flow"
```
3. Click on a target bundle in the design whose flow you wish to modify with the copied flow. The flow is copied and the target bundle becomes attached to your cursor. The following message appears in the Console window.

```
"Pick to place the flow"
```
4. Move your mouse to re-locate the target bundle in the design, then click to anchor it.
5. Repeat steps 1 through 4 to modify the flows of other bundles in the design as needed.

flow create

The `flow create` command lets you interactively create a persistent bundle and the flow path on the specified layers. This command also provides options to automatically route the connection using Auto-I, Breakout or Auto Connect commands.

You can use `flow create` command on rats for dynamic flow planning and creation of bundle directly in canvas. It can also be used on existing bundles or fully-routed clines for re-routing using bundle based auto-interactive routing commands available in context menu.

 The auto-interactive routing operations are available if Design Planning product option is enabled.

Right Mouse Button Option

Create Flow

Related Topics

- [Running the Flow Create Command](#)

Flow Create Options Dialog Box

<i>Bundle name</i>	Specify the name of the bundle created or edit the name of an existing bundle. By default, an auto-generated name is displayed.	
<i>Layer selection</i>	Specify the layers for creating a bundle. These layers are also used by routing operation. By default, all the layers are selected.	
<i>One Layer Only</i>	Enable to allow only a single layer for creating bundle. By default, this option is disabled.	
<i>Ripup Existing Etch</i>	Enable to remove existing clines on the selected rats, bundle, or clines. By default, this option is disabled.	
<i>Auto Blank other rats</i>	Enable to hide unselected rats while creating a new flow. By default, this option is disabled. This option must be set before the first pick of the flow path.	
<i>Routing Operation</i>	Provides options to automatically route the connections after flow path is created. Routing options are available only when <i>Design Planning</i> product option is enabled.	
	<i>Auto Connect (Prototype)</i>	Runs <i>Auto Connect</i> command after flow path is drawn using the selected layers of the flow. <i>Compress</i> Enable this option to compress the trunk routing of the flow to minimum DRC spacing. This option is available only when <i>Auto Connect</i> is enabled. By default, this option is disabled.
	<i>BreakOut Both Ends</i>	Runs Auto-I. BreakOut Both Ends command after flow path is drawn using the selected layers of the flow. By default, this option is disabled.
	<i>BreakOut Closest End</i>	Runs Auto-I. BreakOut Closest End command after flow path is drawn using the selected layers of the flow. This option uses the starting point as the closest end. By default, this option is disabled.
	<i>None</i>	No automatic routing occurs after flow path is drawn. By default, this option is set.

Running the Flow Create Command

1. In *Flow Planning* application mode, hover your cursor over a group of objects (rats, bundles, or fully-routed clines).
The selected objects highlight.
2. Right-click and choose *Create Flow* from the pop-up menu.
The dynamic flow path of the bundle is displayed.
3. Specify a name of the bundle in the *Options* tab or accept the default identifier displayed in the *Bundle name* field.
4. Enable the checkbox *One Layer Only* and specify the desired layer in the *Options* tab.
5. Enable *Ripup Existing Etch* if routed clines are part of the bundle in the *Options* tab.
6. Choose any of the *Routing Operations* in the *Options* tab.
7. Click close to starting end of the bundle to define the breakout bar. This line is the "location" or how far "out" the route from the component will be when the AiBT command is ran.
8. Continue building flow path with subsequent clicks in canvas.
By default, the flow segment snapping occurs in the orthogonal direction.
9. Alternatively, to create off-angle snapping hold Ctrl key while building flow path.
10. The last click defines the location of the breakout bar on opposite end of bundle.
11. Right-click and choose *Done*.
A flow/bundle is created that can be routed using auto-interactive commands. If *Auto-I. BreakOut* or *Auto Connect* options was enabled the etch is automatically generated.
12. Repeat above steps to modify the flows of other bundles in the design.


Related Topics

- [flow create](#)

flow default

The `flow default` command removes all flow segments and flow vias from selected bundles and restores the default flow path that was produced when the bundles were created.

Access Using

- Menu Path: *FlowPlan – Restore Default Flow*
- Right Mouse Button Option: *Restore Default Flow*
- Toolbar Icon: 

Restoring the default flow to selected bundles

1. In IFP application mode, select one or more bundles whose default flow you wish to restore.

✓ 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 bundles highlight and also appear in the *WorldView* window.

2. With your cursor on a selected bundle, right-click and choose *Restore Bundle Flow*.
All flow segments and flow vias are removed from the selected bundles, their default flow configuration is restored, and their *Flow's x/y Guidance* property is set to off (no router guidance).

flow move

The `flow move` command lets you move the entire flow of a pre-selected bundle (including its gather points) to a new location in the design.

Access Using

- Menu Path: *FlowPlan – Move Flow*
- Right Mouse Button Option: *Move Flow*

Moving a flow

1. In IFP application mode, hover your cursor over the bundle whose flow you want to move. The 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 *Move Flow* from the menu.
The bundle flow attaches to your cursor.
3. Move your cursor to relocate the bundle flow, then click to anchor it back in the design.
4. Repeat steps 1 through 3 to move other bundle flows in the design as needed.

✔ You can shortcut this procedure by dragging the bundle flow with your mouse. This is especially convenient when you need to move several bundle flows in the design. Note that "Ratbundle" must be selected for the mouse drag to function. Hover your cursor over a bundle and use the Tab key to pre-select the ratbundle (note the data tip) for the drag operation.

flow rat layer control

The `flow rat layer control` command lets you assign routing layer for individual rats on bundles, without splitting the bundle. The command has two options: edit and restore default.

In edit mode, rakes are expanded. You can pick individual rats and change the layers.

Right Mouse Button Option

Flow Edit – Rat Layer Control

Related Topics

- [Running the Rat Layer Control Command](#)

Editing Bundle Flow

1. In IFP application mode, hover your cursor over the flow line segment you want to slide.

✓ Design density may make flow segment selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

The segment highlights.

2. Right-click and choose *Slide Flow* from the menu.
3. Move your cursor to slide the segment in a negative or positive direction and reposition it in the canvas, then click to lock its location.
The segment is repositioned and the lengths of the adjacent segments are adjusted accordingly.
4. Repeat steps 1, 2, and 3 to slide other flow line segments as needed.

Running the Rat Layer Control Command


1. Hover your cursor over a bundle end. The tool highlights the segment and a datatip identifies its name.
2. Right-click and choose *Flow Edit - Rat Layer Control - Edit*.
3. Click to choose a rake or group of rakes.
4. Click to choose the second rat for swapping.
The selected rats are swapped.
5. Alternatively, you can choose a rat to slide. Use LMB to select and move the rat at the desired location in the sequence.
6. Right-click and choose *Done* to complete the command.

Related Topics

- [flow rat layer control](#)

flow sequence

The flow sequence command swaps the position of the selected rats in a sequence to define the desired pattern when exiting a component's pin/via fields.

 In PCB Editor, this command is available with the Design Planning option only.

Running the Flow Sequence Command


1. Hover your cursor over a bundle end. The tool highlights the segment and a datatip identifies its name.
2. Right-click and choose *Flow Edit - Sequence - Edit*.
3. Click to choose the first rat in the sequence.
4. Click to choose the second rat for swapping.
The selected rats are swapped.
5. Alternatively, you can choose a rat to slide. Use LMB to select and move the rat at the desired location in the sequence.
6. Right-click and choose *Done* to complete the command.

flow slide

The `flow slide` command lets you slide a flow element. You can slide a flow segment in a direction perpendicular to its length with the orientation of the segment remaining fixed. You can slide a flow vertex in any direction while maintaining the orientation of adjacent segments. It also lets you slide a flow via in a similar fashion. Where you position your cursor on the flow line determines how the command responds. This command does not slide multiple flow elements simultaneously.

Access Using

- Menu Path: *FlowPlan – Flow Slide*
- Right Mouse Button Option: *Slide Flow*

 For a list of flow editing shortcuts, see [Flow Line Editing Shortcuts](#)

Related Topics

- [Sliding a Flow Line Vertex](#)
- [Sliding a Flow Line Via](#)
- [Flow Line Editing Shortcuts](#)

Sliding a Flow Line Segment

1. In IFP application mode, hover your cursor over the flow line segment you want to slide.

✓ Design density may make flow segment selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

The segment highlights.

2. Right-click and choose *Slide Flow* from the menu.
3. Move your cursor to slide the segment in a negative or positive direction and reposition it in the canvas, then click to lock its location.
The segment is repositioned and the lengths of the adjacent segments are adjusted accordingly.
4. Repeat steps 1, 2, and 3 to slide other flow line segments as needed.

Related Topics

- [Sliding a Flow Line Via](#)
- [Flow Line Editing Shortcuts](#)

Sliding a Flow Line Vertex

1. In IFP application mode, hover your cursor over the flow line vertex you want to slide.

✓ Design density may make flow vertex selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

2. Right-click and choose *Slide Flow* from the menu.
3. Move your cursor to slide the flow vertex and reposition it in the canvas, then click to lock its location.

The vertex is repositioned with the orientation of the adjacent segments maintained.

⚠ This operation may add additional flow line segments as needed to maintain connectivity.

4. Repeat steps 1, 2 and 3 to slide other flow line vertices as needed.

Related Topics

- [flow slide](#)
- [Flow Line Editing Shortcuts](#)

Sliding a Flow Line Via

1. In IFP application mode, hover your cursor over the flow line via you want to slide.

✓ Design density may make flow via selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

The flow line via highlights.

2. Right-click and choose *Slide Flow* from the menu.
3. Move your cursor to slide the flow via and reposition it in the canvas, then click to lock its location.

The via is repositioned with the orientation of the adjacent segments maintained.

4. Repeat steps 1, 2 and 3 to slide other flow line vias as needed.

Related Topics

- [flow slide](#)
- [Sliding a Flow Line Segment](#)

Flow Line Editing Shortcuts

To ...	Position your cursor here ...	Press and hold this key ...	and use this mouse action ...
Insert and position a new flow vertex.	Over a flow line segment	n/a	Depress the left button and drag
Slide an existing flow line segment.	" "	Shift	" "
Insert a new flow via.	" "	n/a	Double-click
Move an existing flow line vertex.	Over a flow line vertex	n/a	Depress the left button and drag
Slide an existing flow line vertex.	" "	Shift	" "
Move an existing flow via.	Over a flow via	n/a	" "
Slide an existing flow via.	" "	Shift	" "
Remove a flow via.	" "	n/a	Double-click

Related Topics

- [flow slide](#)
- [Sliding a Flow Line Segment](#)
- [Sliding a Flow Line Vertex](#)

flow vertex

The `flow vertex` command lets you insert a new vertex or move an existing vertex in a flow line, changing its path configuration. Where you click on the flow line determines how the command responds. If you click on top of an existing vertex, the command lets you move it. Otherwise, a new vertex is inserted in the flow line at the cursor location and the command lets you locate it using the mouse.

Access Using


Menu Path: *FlowPlan – Edit Flow Vertex*

Related Topics

- [Moving an Existing Flow Line Vertex](#)
- [Flow Line Editing Shortcuts](#)
- [flow vertex insert](#)
- [flow vertex move](#)
- [flow vertex delete](#)


Inserting a vertex into a flow line segment

1. In IFP application mode, click on a flow line segment where you want to insert a new vertex.

 Design density may make flow segment selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

The flow line segment highlights and also appears in the *WorldView* window.


2. Choose *FlowPlan – Edit Flow Vertex* from the menu bar.
3. Move your cursor to locate the new vertex in the canvas.
The adjacent flow line segments snap to 45 and 90 degree positions as you move your cursor across the canvas.

 You can depress the Ctrl key to disable the angle snapping.


4. When the lengths and angles of adjacent flow line segments are acceptable, click to lock the location of the vertex.
5. Repeat steps 1, 2, 3, and 4 to insert vertices into other flow line segments as needed.

Moving an Existing Flow Line Vertex

1. In IFP application mode, click on the flow line vertex that you want to move.

 Design density may make flow segment selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

2. Choose *FlowPlan – Edit Flow Vertex* from the menu bar.
3. Move your cursor to relocate the vertex in the canvas.
The adjacent flow line segments snap to 45 and 90 degree positions as you move your cursor across the canvas.

 You can depress the Ctrl key to disable the angle snapping.

4. When the lengths and angles of the adjacent flow line segments are acceptable, click to lock the location of the vertex.
5. Repeat steps 1 through 4 to move other flow line vertices as needed.

Related Topics

- [flow vertex](#)

flow vertex delete

The `flow vertex delete` command lets you remove a vertex in a flow line changing its path configuration.

Access Using

- Menu Path: *FlowPlan – Delete Flow Vertex*
- Right Mouse Button Option: *Delete Flow Vertex*

Related Topics

- [Flow Line Editing Shortcuts](#)
- [flow vertex insert](#)
- [flow vertex move](#)

Deleting a Flow Line Vertex

1. In IFP application mode, hover your cursor over the flow line vertex that you want to remove.

✓ Design density may make flow vertex selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.


2. Right-click and choose *Delete Flow Vertex* from the menu.
The vertex is deleted and the path of the flow line updates.
3. Repeat steps 1 and 2 to delete other flow line vertices.

flow vertex insert

The `flow vertex insert` command lets you insert a new vertex in a flow line changing its path configuration.

Access using:

- Right Mouse Button Option: *Insert Flow Vertex*

 For a list of flow editing shortcuts, see [Flow Line Editing Shortcuts](#)

Related Topics

- [Flow Line Editing Shortcuts](#)
- [flow vertex move](#)
- [flow vertex delete](#)

Inserting a Vertex into a Flow Line

1. In IFP application mode, hover your cursor over a flow line segment where you want to insert a new vertex.

✔ Design density may make flow segment selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

The flow line segment highlights.

2. Right-click and choose *Insert Flow Vertex* from the menu.
3. Move your cursor to locate the new vertex in the canvas.
The adjacent flow segments snap to 45 and 90 degree positions as you move your cursor across the plan to locate the vertex.
4. When the lengths and angles of the adjacent flow segments are acceptable, click to lock the location of the vertex.
5. Repeat steps 1 through 4 to insert vertices into other flow lines as needed.

flow vertex move

The `flow vertex move` command lets you move an existing vertex in a flow line changing its path configuration.

Access using:

- Right-click Command: *Move Flow Vertex*

Related Topics

- [Flow Line Editing Shortcuts](#)
- [flow vertex insert](#)
- [flow vertex delete](#)

Moving a flow line vertex

1. In IFP application mode, hover your cursor over the flow vertex that you want to move.

✓ Design density may make flow vertex selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

2. Right-click and choose *Move Flow Vertex* from the menu.
3. Move your cursor to relocate the vertex in the canvas.
The adjacent flow line segments snap to 45 and 90 degree positions as you move your cursor across the plan.
4. When the lengths and angles of the adjacent flow segments are acceptable, click to lock the location of the vertex.
5. Repeat steps 1 through 4 to move other flow line vertices as needed.

flow via delete

The `flow via delete` command lets you remove a flow via from a flow line.

Access Using

- Menu Path: *FlowPlan – Delete Flow Via*
- Right-click Command: *Delete Flow Via*

Related Topics

- [Flow Line Editing Shortcuts](#)
- [flow via insert](#)
- [flow via move](#)

Deleting a Flow Line Via

1. In IFP application mode, hover your cursor over the flow via that you want to remove.

✓ Design density may make flow via selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

The flow via highlights.

2. Right-click and choose *Delete Flow Via* from the menu.
The via is removed from the flow line.
3. Repeat steps 1 and 2 to delete other flow line vias as needed.

flow via insert

The `flow via insert` command lets you insert a flow via into a flow line.

Access Using

- Menu Path: *FlowPlan – Insert Flow Via*
- Right-click Command: *Insert Flow Via*

Related Topics

- [Flow Line Editing Shortcuts](#)
- [flow via delete](#)
- [flow via move](#)

Inserting a flow via into a flow line segment

1. In IFP application mode, hover your cursor over a flow segment or flow vertex where you want to insert a new flow via.

✔ Design density may make flow line selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

The flow line segment highlights.

2. Right-click and choose *Insert Flow Via* from the menu.
A flow via is inserted into the flow segment at the cursor location and the segment is divided into two segments allowing you to set layer properties differentially.
3. Repeat steps 1 and 2 to insert flow vias in other flow segments as needed.

flow via move

The `flow via move` command lets you move an existing flow via in a flow line.

Access Using

- Menu Path: *FlowPlan – Move Flow Via*
- Right-click Command: *Move Flow Via*

Related Topics

- [Flow Line Editing Shortcuts](#)
- [flow via delete](#)
- [flow via delete](#)

Moving an Existing Via in a Flow Line

1. In IFP application mode, hover your cursor over a flow via that you want to move.

✓ Design density may make flow via selection difficult. You can limit the find criteria to just flow objects by right-clicking in the Design window, then choosing *Super filter – Flow Edit* from the menu.

The flow via highlights.

2. Right-click and choose *Move Flow Via* from the menu
3. Move your cursor to relocate the flow via in the canvas.
The adjacent flow segments snap to 45 and 90 degree positions as you move your cursor across the plan.
4. When the via location is appropriate, click to lock it.
5. Repeat steps 1 through 4 to move other flow vias.

form

Used in conjunction with the [funckey](#) command to navigate the padstack list when you execute *Route – Connect* ([add connect](#) command).

- [Creating a Function Alias](#)
- [form_prev](#)
- [form_next](#)

form_next

Used in conjunction with the [funckey](#) command to navigate the padstack list when you execute *Route – Connect* ([add connect](#) command).

To cycle through your via list by clicking the `v` key, and eliminate mouse travel to the Options window pane to select an alternative via, add the following to your `env` file or enter on the command prompt:

```
funckey v form_next mini padstack_list
```

- [Creating a Function Alias](#)

form_prev

Used in conjunction with the [funckey](#) command to navigate the padstack list when you execute *Route – Connect* ([add connect](#) command).

- [Creating a Function Alias](#)
- [form](#)
- [form_next](#)


front

Brings a window that has been partially hidden by another window to the front of the desktop.

`front`

fse_arc_tangent

The `fse_arc_tangent` command lets you draw a tangent arc between two points from two different line or arc segments. Once the points are selected, you can reposition your cursor to choose a different result.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the Allegro X *User Guide: Working with RF PCB*.

Related Topics

- [Drawing an arc tangent between two points from two different segments](#)


Arc Tangent Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Arc Tangent*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Pick tangent point by mouse</i>	Specifies that the start point of the arc tangent is designated using a mouse pick on the first segment selected.
<i>Use end point as tangent point</i>	Specifies that the start point of the arc tangent is the closest endpoint of the first segment selected.
<i>Clockwise</i>	When enabled (checked), specifies that the arc tangent be drawn in a clockwise direction from the designated start point.
<i>Inner tangency</i>	When enabled (checked), specifies that the arc tangent be drawn so that it encloses the source or destination arcs. Note: This option is available only when the source and destination segments have been chosen and at least one of those segments is an arc.

Drawing an arc tangent between two points from two different segments

1. Choose *RF-PCB – Flexible Shape Editor – Arc Tangent*.
The options appear in the Options pane.
 2. Select the Active Class and Subclass to choose the etch layer.
 3. Choose *Pick tangent point by mouse* to choose the point that the tangent is drawn from.
- or -
Choose *Use end point as tangent point* to have the tangent drawn from the closest end point of the first selected segment.
 4. Choose the direction for the tangent arc by enabling (checked) or disabling the *Clockwise* option. With the option disabled, the tangent arc adopts a counter-clockwise direction.
 5. Click on a source (line or arc) segment in the design, then click on a destination (line or arc) segment.
An arc tangent result appears.
 6. If the result is satisfactory, click again to draw the arc tangent as shown, then proceed to the last step. Otherwise, continue with the next step.
 7. Try one or more of the following sub-steps to change the drawing of the arc tangent.
 - a. Reposition your cursor until the arc tangent display shows it drawn from the opposite end of the source segment.
 - b. Choose whether the result should be an inner or an outer arc tangent by enabling (checked) or disabling (unchecked) the *Inner tangency* option.
-  An inner arc tangent encloses the source and destination arcs. This option is only available when at least one of the segments you select is an arc and only after selecting the source and destination segments.
8. Once your desired result is displayed, click again to draw the arc tangent.
 9. Repeat steps 2 through 6 to draw other arc tangents.
- or -
Right-click and choose *Done*.

Related Topics


- [fse_arc_tangent](#)

fse_break_delete

The `fse_break_delete` command lets you delete extra lines and segments that result from creating a shape outline to convert into a filled shape. You must have a closed outline in order to compose a shape.

Allegro X PCB Editor will delete any lines that have:

- at least one intersection point on either end
- no intersection point on one side
- no intersection point with any other lines
- a bounding box surrounding them

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro X User Guide: Working with RF PCB*.

Related Topics

- [Removing extra lines and segments from a shape boundary](#)

Break and Delete Command: Options Panel


Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Break and Delete*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
------------------------------	--

Removing extra lines and segments from a shape boundary

1. Choose *RF-PCB – Flexible Shape Editor – Break and Delete*.
2. Select the Active Class and Subclass to choose the etch layer.
3. Click on the line or segment that you want to remove.
The line you chose highlights.
4. Click to delete the line.

 Or if you choose a command from the right-click pop-up menu, you can undo, complete, or cancel the command at any time.


5. Repeat steps 1 through 3 until you are satisfied with the results.
6. Right-click and choose *Done* from the pop-up menu.
The shape boundary is now ready to convert to a filled shape using *Shape – Compose Shape* in Allegro.

Related Topics

- [fse_break_delete](#)

fse_edge_move

The `fse_edge_move` command lets you move a shape edge to a different location while maintaining its angle and length. Dual x and y offset values control the offset angle.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro X User Guide: Working with RF PCB*.


Related Topics

- [Moving a shape edge](#)

Edge Move Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Edge Move*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Move by mouse</i>	Specifies that the new location of the selected shape edge is designated using a mouse pick.
<i>Move by accurate offsets</i>	Specifies that the new location of the selected shape edge is designated using the offset values specified in the <i>Horizontal offset</i> and <i>Vertical offset</i> fields.
<i>Horizontal offset</i>	The horizontal offset for the shape edge from its original position.
<i>Vertical offset</i>	The vertical offset for the shape edge from its original position.
<i>Use reference edge</i>	<p>Enable (check) this option to move the selected edge relative to a reference edge. If you're moving by mouse, the cursor snaps to the nearest reference edge. If you are moving by accurate offsets the cursor snaps to a distance from the reference edge, specified by the horizontal and vertical offsets. Change the active class and subclass if the reference edge is not on current active layer.</p> <div> The reference edge must be parallel to the selected edge. This field does not display if you select an arc.</div>

Moving a shape edge


1. Choose *RF-PCB – Flexible Shape Editor – Edge Move*.
The Edge Move options appear in the Options pane.
2. Select the Active Class and Subclass to choose the etch layer on which the edge resides.
3. Click on the edge to move, to select it.
4. Choose *Move by mouse* to specify the new location for the selected edge using a mouse pick.
- or -
Choose *Move by accurate offsets* to specify the new location for the selected edge, relative to the source picking point, using the values in the *Horizontal offset* and *Vertical offset* fields.
5. To snap the mouse cursor to edges that are parallel to the selected edge enable *Use reference edge*.
 - a. Select the Active Class and Subclass to choose the etch layer on which the reference edge resides.
 - b. Move the mouse and the cursor snaps to the reference edge.
- or -
Right-click and choose *Snap pick to* and select a reference edge.
6. If you chose *Move by mouse*, click on the shape edge to select it, drag it to its new position, then click again to anchor it.
- or -
If you chose *Move by accurate offsets*, edit the values in the *Horizontal offset* and *Vertical offset* fields, then select the edge to move.
The edge moves and the shape re-fills itself.
7. Repeat steps 2 and 3 to move other shape edges.
- or -
Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_edge_move](#)

fse_edge_spread

The `fse_edge_spread` command lets you move a shape edge to a new position while maintaining its angle. The length of the edge is constrained within the angle of the two adjacent segments.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.


For further details, see the RF Shape Editing chapter in the Allegro X User Guide: Working with RF PCB. Related Topics

- [Spreading a shape edge](#)

Edge Spread Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Edge Spread*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
Spread by mouse	Choose this option to spread the edge using the mouse pointer.
Spread by accurate offset	Choose this option to spread the edge using the value provided in the Offset field.
Offset	Specifies the edge offset from its original position. Use either a negative or positive offset value to control the offset direction. Note: If the shape boundary was constructed in a counter-clockwise direction, a positive offset spreads the edge to the outside of the shape. If the shape boundary was constructed in a clockwise direction, a positive offset spreads the edge to the inside of the shape.
Use reference edge	<p>Enable (check) this option to spread the selected edge relative to a reference edge. If you're spreading by mouse, the cursor snaps to the nearest reference edge. If you are spreading by accurate offsets the cursor snaps to a distance from the reference edge, specified by the horizontal and vertical offsets. Change the active class and subclass if the reference edge is not on current active layer.</p> <div> The reference edge must be parallel to the selected edge. This field does not display if you select an arc.</div>

Spreading a shape edge


1. Choose *RF-PCB – Flexible Shape Editor – Edge Spread*.
The Edge Spread options appear in the Options pane.
2. Select the Active Class and Subclass to choose the etch layer on which the edge resides.
3. Click on the edge to move, to select it.
4. Choose *Spread by mouse* to specify the new location for the selected edge using a mouse pick.
- or -
Choose *Spread by accurate offset* to specify the new location for the selected edge, relative to the source picking point, using the value in the *Offset* field.
5. To snap the mouse cursor to edges that are parallel to the selected edge enable *Use reference edge*.
 - a. Select the Active Class and Subclass to choose the etch layer on which the reference edge resides.
 - b. Move the mouse and the cursor snaps to the reference edge.
- or -
Right-click and choose *Snap pick to* and select a reference edge.
6. If you chose *Spread by mouse*, click on the shape edge to select it, drag it to its new position, then click again to anchor it.
- or -
If you chose *Spread by accurate offset*, edit the value in the *Offset* field, then select the edge to spread.
The edge moves and the shape re-fills itself.
7. Repeat steps 2 and 3 to spread other shape edges.
- or -
Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_edge_spread](#)

fse_edge_stretch

The `fse_edge_stretch` command lets you move a shape edge to a different location while maintaining its angle, length, and perpendicularity.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro User Guide: Working with RF PCB*.


Related Topics

- [Stretching a shape edge](#)

Edge Stretch Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Edge Stretch*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Stretch by mouse</i>	Specifies that the new location of the selected shape edge is designated using a mouse pick.
<i>Stretch by accurate offset</i>	Specifies that the new location of the selected shape edge is designated using the value in the <i>Offset</i> field.
<i>Offset</i>	The offset for the shape edge from its original position. Use a negative or positive offset value to control the offset direction. Note: If the shape boundary was constructed in a counter-clockwise direction, a positive offset stretches the edge to the outside of the shape. If the shape boundary was constructed in a clockwise direction, a positive offset stretches the edge to the inside of the shape.
<i>Use reference edge</i>	<p>Enable (check) this option to stretch the selected edge relative to a reference edge. If you're spreading by mouse, the cursor snaps to the nearest reference edge. If you are spreading by accurate offsets the cursor snaps to a distance from the reference edge, specified by the horizontal and vertical offsets. Change the active class and subclass if the reference edge is not on current active layer.</p> <div style="border: 1px solid #fde725; padding: 10px; margin-top: 10px;"> <p> The reference edge must be parallel to the selected edge. This field does not display if you select an arc.</p> </div>

Stretching a shape edge

1. Choose *RF-PCB – Flexible Shape Editor – Edge Stretch*.
The Edge Stretch options appear in the Options pane.
2. To snap the mouse cursor to edges that are parallel to the selected edge enable *Use reference edge*.
 - a. Select the Active Class and Subclass to choose the etch layer on which the reference edge resides.
 - b. Move the mouse and the cursor snaps to the reference edge.
- or -
Right-click and choose *Snap pick to* and select a reference edge.
3. Choose *Stretch by mouse* to specify the new location for the selected edge using a mouse pick.
- or -
Choose *Stretch by accurate offset* to specify the new location for the selected edge using the value in the *Offset* field.
4. To snap the mouse cursor to edges that are parallel to the selected edge enable *Use reference edge*.
 - a. Select the Active Class and Subclass to choose the etch layer on which the reference edge resides.
 - b. Move the mouse and the cursor snaps to the reference edges.
5. If you chose *Stretch by mouse*, click on the shape edge to select it, drag it to its new position, then click again to anchor it.
- or -
If you chose *Stretch by accurate offset*, edit the value in the *Offset* field, then select the edge to move.
The edge stretches and the shape re-fills itself.
6. Repeat steps 2 and 3 to stretch other shape edges.
- or -
Right-click and choose *Done* to complete the operation.

Related Topics


- [fse_edge_stretch](#)

F Commands

F Commands--fse_edge_stretch

fse_end_connect

The `fse_end_connect` command lets you connect the ends of two lines with a line.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro X User Guide: Working with RF PCB*.

Related Topics

- [End connecting the ends of two lines](#)

Line End Connect Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Line End Connect*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
------------------------------	--

End connecting the ends of two lines


1. Choose *RF-PCB – Flexible Shape Editor – Line End Connect*.
2. Select the Active Class and Subclass to choose the etch layer.
3. Click near the end of the first line to connect.
4. Click near the end of the second line to connect.
A line is created that connects the two lines from the ends closest to where you clicked.
5. Repeat steps 2 and 3 to connect other lines.
- or -
Click the right mouse button and choose *Done* to end the command.

Related Topics

- [fse_end_connect](#)

fse_seg_tangent

The `fse_seg_tangent` command lets you draw a tangent line or arc from a start point on a line or arc segment.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro X User Guide: Working with RF PCB*.

Related Topics

- [Drawing a tangent line or arc from a designated start point](#)

Tangent Segment Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Tangent Segment*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Pick tangent point by mouse</i>	Specifies that the tangent line start point is designated using a mouse pick on the selected line or arc segment.
<i>Use end point as tangent point</i>	Specifies that the tangent line start point is the endpoint of the selected line or arc segment closest to the mouse pick.
Tangent arc radius	Specifies the radius to use for a tangent arc. Note: The value must be positive. If the value is zero, a tangent line is created.
Tangent line/arc length	Specifies a fixed length for the tangent line or arc. Note: Only positive non-zero values are allowed.
Use specified length	Enables (checked) or disables (unchecked) the use of the fixed length specified in the Tangent Line/Arc Length field.
Clockwise	Specifies that a tangent arc is drawn in a clockwise direction. Note: This field is visible when the Tangent arc radius is greater than zero.
Reverse direction	Specifies that a tangent line is drawn in the opposite direction.

Drawing a tangent line or arc from a designated start point


1. Choose *RF-PCB – Flexible Shape Editor – Tangent Segment*.
The Tangent Line/Arc options appear in the Options pane.
2. Select the Active Class and Subclass to choose the etch layer.
3. Choose *Pick tangent point by mouse* to specify that the start point for the tangent on the selected line or arc segment is designated using a mouse pick.
- or -
Choose *Use end point as tangent point* to specify that the start point for the tangent is the endpoint closest to the mouse pick on the selected line or arc segment.
4. If you want to draw a tangent arc, enter a positive non-zero value in the Tangent arc radius field. Otherwise, leave the value at zero (indicating a tangent line).
5. If you want to draw the tangent at a fixed length, enter a value in the *Tangent line/arc length* field and enable (check) the *Use specified length* field. Otherwise, proceed to the next step.
6. If you chose *Pick tangent point by mouse*, click on the line or arc segment where you want the tangent to start.
- or -
If you chose *Use end point as tangent point*, click on the line or arc segment near the endpoint where you want the tangent to start.
7. If you enabled (checked) *Use specified length*, the tangent is already drawn from the start point using the specified length. Otherwise, move your mouse to adjust the tangent length, then click to fix the endpoint.
8. If you have drawn a tangent arc and want to redraw it in a clockwise direction, enable (check) *Clockwise*, then repeat step 6. Otherwise, proceed to the next step.
9. If you have drawn a tangent line and want to redraw it in the opposite direction, enable (check) *Reverse direction*, then repeat step 6. Otherwise proceed to the next step.
10. Repeat steps 2 through 8 to draw other tangent line or arc segments.
- or -
Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_seg_tangent](#)

fse_shape_chamfer

The `fse_shape_chamfer` command lets you convert all corners of a shape to arcs or miters. The length of each miter leg can be specified separately.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro User Guide: Working with RF PCB*.

Related Topics

- [Converting all corners of a shape to arcs or miters](#)

Shape Chamfer Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Shape Corner Chamfer*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Chamfer to arc</i>	Choose to convert all chamfers to arcs.
<i>Arc radius</i>	Specifies the radius to use for arc corners. Note: This option is enabled when <i>Chamfer to arc</i> is selected.
<i>Chamfer to miter</i>	Choose to convert all chamfers to miters.
<i>Left miter length</i>	Specifies the length for the left leg of the miter. Note: This option is enabled when <i>Chamfer to miter</i> is selected.
<i>Right miter length</i>	Specifies the length for the right leg of the miter. Note: This option is enabled when <i>Chamfer to miter</i> is selected.

Converting all corners of a shape to arcs or miters


1. Choose *RF-PCB – Flexible Shape Editor – Shape Corner Chamfer*.
The Corner Chamfer options appear in the Options pane.
2. Select the Active Class and Subclass to choose the etch layer.
3. Choose *Chamfer to arc* to specify that all shape corners be converted to arcs, then enter a value in the *Arc radius* field.
- or -
Choose *Chamfer to miter* to specify the conversion of all shape corners to miters, then enter values in the *Left miter length* and *Right miter length* fields to specify lengths for each miter leg.
4. Click on the shape to chamfer.
All corners of the shape are converted.
5. Repeat steps 2 and 3 to convert the corners of other shapes.
- or -
Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_shape_chamfer](#)

fse_shape_logicop

The `fse_shape_logicop` command lets you perform logical operations with two groups of overlapping shapes to create a new shape. Each group may contain one or more shapes. You can create the shape groups on-the-fly using the right mouse button. Choose *Temp Group*, click on each group member, then choose *Complete*.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the Allegro X *User Guide: Working with RF PCB*.

Related Topics

- [Creating a boolean shape from two groups of overlapping shapes](#)

Shape Logicop Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Shape Operations*

Operation Type

<i>Union</i>	Returns the union of the two shape groups.
<i>Intersection</i>	Returns the intersection of the two shape groups.
<i>Difference</i>	<i>Returns the difference of the two shape groups. (group 1 - group 2)</i>
<i>Symmetric difference</i>	Returns the union of two opposing difference results. ((group1 - group 2) + (group 2 - group 1))

Creating a boolean shape from two groups of overlapping shapes

1. Choose *RF-PCB – Flexible Shape Editor – Shape Operations*.
The Logic Operations options appear in the Options pane.
2. Choose an operation type to specify the boolean operation you want to perform.
3. Choose the first shape group by doing one of the following:
Click on a single shape.
Press the left mouse button and drag-select two or more shapes using a bounding box.
 - a. Click the right mouse button and choose *Temp Group*.
 - b. Click on two or more shapes.
 - c. Click the right mouse button and choose *Complete*.

Each of the selected shapes highlight and you are prompted to select the second shape group.

4. Repeat the previous step to select the second shape group.
The operation applies and returns a shape result.
5. Click anywhere in the Design window to continue, and then repeat steps 2, 3, and 4 to create other logical shapes.
- or -
Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_shape_logicop](#)

fse_shape_scale

The `fse_shape_scale` command lets you create a scaled copy of a shape. You can create the copy on the same layer using the same net as the source shape, or you can send it to a different etch layer and specify a different net name.

Note:

- Although the shape copy must reside on an etch layer, the source shape may reside on any layer.
- This command only supports the etch class. To scale copy a shape to layers of different classes, see the [fse_shape_zcopy](#) command.
- Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro User Guide: Working with RF PCB*.

Related Topics

- [Creating a scaled copy of a shape](#)

Shape Scale Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Shape Scale*


<i>Scale Factor</i>	Specifies the scale factor used to create the shape copy. Note: The factor must be a positive, non-zero value.
<i>Destination layer</i>	<i>Specifies the etch layer for the shape copy.</i>
<i>Shape Net</i>	Specifies and displays a net name for the shape copy. Note: This option does not appear when <i>Destination Layer</i> is set to the current layer.

Creating a scaled copy of a shape

1. Choose *RF-PCB – Flexible Shape Editor – Shape Scale*.
The Scaled Shape options appear in the Options pane.
2. Select the Active Class and Subclass to choose the etch layer.
3. In the *Scale Factor* entry box, enter a positive, non-zero value to specify the desired scale for the shape copy.

 Using a value less than 1.0 will reduce the size of the shape copy.

4. Click the down-arrow in the *Destination layer* drop-down field and select an etch layer for the shape copy.
5. In the Design window, click on the shape that you want to copy.
The shape is copied, scaled, and placed on the selected destination layer.

 To see the copy, the destination layer must be visible.


6. Repeat steps 2, 3, and 4 to scale copy other shapes.
- or -
Right-click and choose *Done* to complete the operation.


Related Topics

- [fse_shape_scale](#)

fse_shape_zcopy

The `fse_shape_zcopy` command lets you create a scaled copy of a shape on multiple classes and subclasses. This command differs from the `fse_shape_scale` command in that it supports all classes and copies to multiple layers simultaneously.

 Some classes only support unfilled shapes. Dynamic shapes are only supported by the etch class. When you use this command to copy a shape to different classes, it automatically decides whether to fill shapes or whether to create dynamic shapes.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing Yes in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing No, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the Allegro X *User Guide: Working with RF PCB*.

Related Topics

- [Creating scaled copies of a shape on ETCH subclasses](#)
- [Creating scaled copies of a shape on non-ETCH subclasses](#)

Shape Z-Copy Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Multi-Layer Shape ZCopy*

<i>Class/Subclass</i>	Specifies the class and subclasses to copy the shape to.
<i>Select All Subclasses</i>	<i>Selects all subclasses of the selected class to copy the shape to.</i>
<i>Deselect All Subclasses</i>	<i>Removes checks from all subclass check boxes for the currently selected class.</i>
<i>Clear All Class Selections</i>	Removes checks from subclass check boxes in all classes.
<i>Create Dynamic Shape</i>	When enabled (checked), specifies that dynamic shapes are used on just the currently selected (checked) etch subclass. Note: This option is available only when the currently selected class is ETCH.
<i>Dynamic For All Subclasses</i>	Once clicked, specifies that dynamic shapes are used on all etch subclasses. Note: This option is available only when the selected class is ETCH.
<i>Shape Net</i>	Enables you to select a net name for the shape copy on the currently selected (checked) subclass. Note: This option is available only when the currently selected class is ETCH.
<i>Same Net For All Subclasses</i>	Once clicked, specifies that the same net name (the one displayed) is used on all etch subclasses.
<i>Expand(+)/Contract(-)</i>	Specifies the scale of the shape copy on the selected (checked) subclass. Note: Use a negative or positive value to control expansion or contraction of the shape copy. A value of zero specifies that it is copied at original scale.
<i>Same Value For All Subclasses</i>	Once clicked, specifies that the same shape scale is used on all etch layers.
<i>Offset X</i>	Specifies a horizontal offset for the shape copy on the selected (checked) subclass.

<i>Offset Y</i>	Specifies a vertical offset for the shape copy on the selected (checked) subclass.
<i>Same Value For All Subclasses</i>	Once clicked, specifies that the same X and Y offset values are used for shape copies on all etch subclasses.

Related Topics

- [Creating scaled copies of a shape on non-ETCH subclasses](#)

Creating scaled copies of a shape on ETCH subclasses

1. Choose *RF-PCB – Flexible Shape Editor – Multi-Layer Shape ZCopy*
The Multi-Layer ZCopy options appear in the Options pane.
2. Click the down-arrow in the *Class/Subclass* entry box, and select the ETCH class.
The window displays the ETCH subclasses.
3. Select (check) the box next to the subclass that you want the shape copy sent to.

✓ Although you can select several etch subclasses at the same time, if you expect your shape copy parameters (dynamic, net, scale, offset) to vary from layer to layer, you should select subclasses one at a time. This enables you to assign individual shape copy parameters as described in the following steps.

4. If you want to create a dynamic shape copy, enable (check) the *Create Dynamic Shape* option. Otherwise, a static shape is created.
5. If you want to assign a different net name to the shape copy, click the Net Browser button and select a net from the list. Otherwise, DUMMY NET is used.
6. If you want to scale the shape copy, enter a negative or positive value in the *Expand(+)/Contract(-)* entry box. Otherwise, no scale applies.

⚠ A negative value reduces and a positive value increases the size of the shape copy.

7. If you want to offset the location of the shape copy on its destination layer from the location of the source shape, enter negative or positive values in the *Offset X* and *Offset Y* entry boxes. Otherwise, the shape uses the same location coordinates as the source shape.
8. Repeat steps 3 through 7 to specify shape copy parameters for other etch subclasses.
9. In the Design window, click on the shape that you want to copy.
The shape is copied, scaled, and placed on the selected destination layers.

⚠ To see the copies, the destination layers must be visible.

10. Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_shape_zcopy](#)

Creating scaled copies of a shape on non-ETCH subclasses

1. Choose *RF-PCB – Flexible Shape Editor – Multi-Layer Shape ZCopy*
The Multi-Layer Scale ZCopy options appear in the Options pane.
2. Click the down-arrow in the *Class/Subclass* entry box, and select a non-ETCH class containing the subclasses where you want to copy the shape to.
The window displays the subclasses of the selected class.
3. Select (check) the boxes next to the subclasses that you want to copy the shape to.
4. If you want to assign a different net name to the shape copies, click the Net Browser button and select a net from the list. Otherwise, DUMMY NET is used.
5. If you want to scale the shape copies, enter a negative or positive value in the *Expand(+)/Contract(-)* entry box. Otherwise, no scale applies.

 A negative value reduces and a positive value increases the size of the shape copies.

6. If you want to offset the location of the shape copies on the destination layers from the location of the source shape, enter negative or positive values in the *Offset X* and *Offset Y* entry boxes. Otherwise, the shape copies use the same location coordinates as the source shape.
7. In the Design window, click on the shape that you want to copy.
The shape is copied, scaled, and placed on the selected destination layers.

 To see the copies, the destination layers must be visible.


8. Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_shape_zcopy](#)
- [Shape Z-Copy Command: Options Panel](#)

fse_shape_zdelete

The `fse_shape_zdelete` command lets you delete z-copied shapes from specified classes and subclasses that were created using the `fse_shape_zcopy` command.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro X User Guide: Working with RF PCB*.

Related Topics

- [Deleting Z-copied shapes from selected classes and subclasses](#)

Shape Z-Delete Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Multi-Layer Shape ZDelete*

<i>Class/Subclass</i>	Specifies the class and subclasses to remove z-copied shapes from. Note: Subclasses containing z-copied shapes have check boxes beside them in the window. Checking a box indicates deletion when <i>Delete Selected ZCopied Shapes</i> is clicked.
<i>Clear All Subclasses</i>	<i>Removes checks from all subclass check boxes for the currently selected class.</i>
<i>Clear All Class Selections</i>	Removes checks from subclass check boxes in all classes.
<i>Delete Selected ZCopied Shapes</i>	Deletes z-copied shapes from the currently selected class and subclasses. Note: This action cannot be undone within the command.

Deleting Z-copied shapes from selected classes and subclasses

1. Choose *RF-PCB – Flexible Shape Editor – Multi-Layer Shape ZDelete*
The Multi-Layer Shape ZDelete options appear in the Options Pane.
2. Click the down-arrow in the *Class/Subclass* entry box, and select a class containing subclasses where z-copied shapes reside.
The subclasses of the selected class are displayed.

 The subclasses containing z-copied shapes have check boxes beside them.


3. Check the boxes for those subclasses that you want to delete z-copied shapes from.
4. Repeat steps 2 and 3 to mark other classes and subclasses for z-copied shape deletion.
- or -
Click *Delete Selected Z-Copied Shapes* to invoke the deletion from all currently selected subclasses.
5. Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_shape_zdelete](#)

fse_vertex_convert

The `fse_vertex_convert` command lets you convert individual corners of a shape to an arc or a miter. The length of each miter leg can be specified separately.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro X User Guide: Working with RF PCB*.

Related Topics

- [Converting a corner of a shape to an arc or a miter](#)

Vertex Convert Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Vertex Convert*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Convert to arc</i>	Specifies that the shape vertex converts to an arc.
<i>Arc radius</i>	Specifies the radius to use for an arc corner. Note: This option is only enabled when <i>Convert to arc</i> is selected.
<i>Convert to miter</i>	Specifies that the shape vertex converts to a miter.
<i>Left miter length</i>	Specifies the length for the left leg of the miter. Note: This option is only enabled when <i>Convert to miter</i> is selected.
<i>Right miter length</i>	Specifies the length for the right leg of the miter. Note: This option is only enabled when <i>Convert to miter</i> is selected.

Converting a corner of a shape to an arc or a miter


1. Choose *RF-PCB – Flexible Shape Editor – Vertex Convert*.
The Vertex Convert options appear in the Options Pane.
2. Select the Active Class and Subclass to choose the etch layer.
3. Choose *Convert to Miter* to specify that the selected shape vertex converts to a miter, then enter a values in the *Left miter length* and *Right miter length* fields.
- or -
Choose *Convert to arc* to specify that the selected shape vertex be converts to an arc, then enter a value in the *Arc radius* field.
4. Click on the shape vertex to convert.
The shape vertex converts to the selected type.
5. Repeat steps 2 and 3 to convert other shape vertices.
- or -
Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_vertex_convert](#)

fse_vertex_insert

The `fse_vertex_insert` command lets you insert a vertex into the boundary edge of a shape. Once the vertex is in place, the shape boundary reconfigure itself.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro X User Guide: Working with RF PCB*.

Related Topics

- [Inserting a vertex in the boundary edge of a shape](#)

Vertex Insert Command: Options Panel

Access Using


Menu Path: *RF-PCB – Flexible Shape Editor – Vertex Insert*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Initial insertion offset</i>	Specifies an offset distance from the start point or end point of the selected shape edge to locate an initial insertion datum point for the vertex. Note: <i>A value of zero implies the mid-point of the selected edge.</i>
<i>From edge start point</i>	Specifies that the <i>Initial insertion offset</i> is measured from the start point of the selected shape edge. Note: The outer boundary edges of a shape run in a counter-clockwise direction.
<i>From edge end point</i>	Specifies that the <i>Initial insertion offset</i> is measured from the end point of the selected shape edge. Note: The outer boundary edges of a shape run in a counter-clockwise direction.
Destination insertion parameters	Specifies the offsets from the initial insertion point.
<i>Horizontal offset</i>	Specifies the horizontal offset distance from the initial insertion point to place the vertex. Note: Use either a negative or positive value.
<i>Vertical offset</i>	Specifies the vertical offset distance from the initial insertion point to place the vertex. Note: Use either a negative or positive value.
<i>Start remain</i>	Specifies the length of an edge segment (measured from the edge start point) that will remain intact after inserting the vertex.
<i>End remain</i>	Specifies the length of an edge segment (measured from the edge end point) that will remain intact after inserting the vertex.

Inserting a vertex in the boundary edge of a shape

1. Choose *RF-PCB – Flexible Shape Editor – Vertex Insert*.
The Vertex Insert options appear in the Options Pane.
2. Select the Active Class and Subclass to choose the etch layer.
3. In the *Initial insertion offset* area, specify the location of an insertion datum point on the shape boundary to reference the vertex placement from.


- a. Specify which endpoint of the shape edge (to be selected) to reference the datum point location from. Choose either *From edge start point* or *From edge end point*.

 Shape outer boundary edges run in a counter-clockwise direction.

- b. Enter a value in the *Initial insertion offset* entry box to specify an offset distance from the chosen edge start point or end point used to locate the insertion datum point on the shape boundary.

 Entering a value of zero implies the mid-point of the selected edge.

4. In the *Destination insertion parameters* area, specify the location of the vertex with reference to the insertion datum point.
 - a. In the *Horizontal offset* entry box, enter a negative or positive value to specify the horizontal offset distance from the datum point.
 - b. In the *Vertical offset* entry box, enter a negative or positive value to specify the vertical offset distance from the datum point.
5. In the *Destination insertion parameters* area, specify a length for an edge segment at each end of the shape edge (to be selected) that are to remain intact after inserting the vertex.

 Using length values of zero implies that no edge segments are to remain intact.

- a. In the *Start remain* entry box, enter an edge segment length value (measured from the start point of the selected edge).
 - b. In the *End remain* entry box, enter an edge segment length value (measured from the end point of the selected edge).
6. Click on the shape edge to insert the vertex.

The vertex is inserted using the specified location parameters and the shape boundary is reconfigured.


7. Repeat steps 2 through 5 to insert other shape vertices.
 - or -
 - Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_vertex_insert](#)

fse_vertex_move

The `fse_vertex_move` command lets you move individual corners of a shape.

 Usage of Flexible Shape Editor (FSE) commands in Allegro X PCB Editor are currently restricted to objects (shapes and lines) that have the RFPCB_OBJECT property. If you invoke an FSE command and then select an object that does not have this property, a Confirmation dialog box is presented. Choosing *Yes* in the dialog box attaches an RFPCB_OBJECT property to the object and allows the operation to continue. Choosing *No*, leaves the object as-is and aborts the command.

For further details, see the RF Shape Editing chapter in the *Allegro X User Guide: Working with RF PCB*.

Related Topics

- [Moving a vertex](#)

Vertex Move Command: Options Panel

Access Using

Menu Path: *RF-PCB – Flexible Shape Editor – Vertex Move*

Active Class and Subclass	Choose the proper etch layer. Color boxes in the subclass section align with the etch color on that particular subclass layer.
<i>Move by mouse</i>	Specifies that the new location of the selected shape vertex is designated using a mouse pick.
<i>Move by accurate offsets</i>	Specifies that the new location of the selected shape vertex is designated using the offset values specified in the <i>Horizontal offset</i> and <i>Vertical offset</i> fields.
<i>Horizontal offset</i>	The horizontal offset for the shape vertex from its original position.
<i>Vertical offset</i>	The vertical offset for the shape vertex from its original position.
<i>Use reference vertex</i>	Check (enable) to move the vertex to a selected vertex. When you use this option, the mouse cursor snaps to the nearest vertex. Change the active class and subclass if the reference vertex is not on current active layer.

Moving a vertex

1. Choose *RF-PCB – Flexible Shape Editor – Vertex Move*.
The Vertex Move options appear in the Options pane.
2. Select the Active Class and Subclass to choose the etch layer on which the vertex resides.
3. Click to select the vertex to move.
4. Choose *Move by mouse* to specify the new location for the selected vertex using a mouse pick.
- or -
Choose *Move by accurate offset* to specify the new location for the selected edge using the value in the *Offset* fields.
5. To snap the mouse cursor to a reference vertex enable *Use reference edge*.
 - a. Select the Active Class and Subclass to choose the etch layer on which the reference vertex resides.
 - b. Move the mouse and the cursor snaps to the reference vertex.
- or -
Right-click and choose *Snap pick to* and select a reference vertex.
6. If you chose *Move by mouse*, click on the vertex to select it, drag it to its new position, then click again to anchor it.
- or -
If you chose *Move by accurate offset*, edit the value in the horizontal and vertical Offset fields, then select the vertex to move.
The vertex moves and the shape re-fills itself.
7. Repeat steps 2 and 3 to move other vertices.
- or -
8. Right-click and choose *Done* to complete the operation.

Related Topics

- [fse_vertex_move](#)

fsp auto pinswap

The `fsp auto pinswap` command performs pin assignment that minimizes the length of the rats and number of crossovers on the layout. The optimizations are based on layout specific parameters of a bundle such as gather point, rake order, breakout or fanout locations and so on.

This command lets you optimize in three ways:

- **Rake Order:** The Rake Order optimization cleans up the rats from the routing end-points to the rakes.
- **Breakout Order:** The Breakout Order optimization allows the radial order of the routing the end-points of the connected pins on both sides of a bundle to optimize the crossovers.
- **Reassign Bundle Pins:** The Reassign Bundle Pins command lets you reassign the bundle pins to a new set of pins that are proximal to the bundle gather point.

Access Using

Menu Path: *Place – FPGA System Planner – Auto Pinswap*

Requirements for Auto Pinswap

This command is available with 4FPGA System Planner and ASIC Prototype W/FPGA's options.

Running the Auto Pinswap Command

1. Choose the bundle for optimization.
2. Choose from the menu *Place – FPGA System Planner – Auto Pinswap* or from the pop-up menu.
The *FSP Auto Pinswap Option* dialog box is displayed.
3. Select *Rake Order* or *Breakout Order* or *Reassign Bundle Pins* option.
4. Click *OK*.

Or

1. Choose multiple bundles for optimization.
2. Choose from the menu *Place – FPGA System Planner – Auto Pinswap*.
3. Select bundle to optimize.
The *FSP Auto Pinswap Option* dialog box is displayed for the selected bundle.
4. Select *Rake Order* or *Breakout Order* or *Reassign Bundle Pins* option.
5. Select next bundle to optimize.
6. Choose *Done* from the pop-up menu when all the bundles are optimized.

fsp load database

The `fsp load database` command imports the FSP database for synchronization with layout database in PCB Editor.

When started, this command prompts for the FSP database location and invokes FSP in the background. For more information on FPGA-based system design with the FSP tool see the [Allegro Design Entry HDL-FPGA System Planner Flow Guide](#).

Access Using

Menu Path: *Place – FPGA System Planner – Load Database*

fsp manual pinswap

The `fsp manual pinswap` command provides an interactive environment for swapping the pins on FPGA devices, in PCB Editor. This command automatically recommends pins on the FPGA component to swap for reducing crossovers.

Requirements

To use the `fsp manual pinswap` command first time, you need the following:

- FSP license
- Synchronized FSP, Schematic and PCB Editor databases

Access Using


- Menu Path: *Place – FPGA System Planner – Manual Pinswap*

Running the Manual Pinswap

1. Select a pin on the FPGA component for swapping.
2. To invoke command
 - Type `fsp manual pinswap` in the command prompt.

The PCB Editor highlights the pins of the FPGA component that are available for swapping with the selected pin.

3. Choose a pin from the highlighted pins for swapping with the selected pin.

 For differential pairs, quad signal, and other pins that belong to the signal group are also highlighted and all the signals are moved when a destination is selected.

1. Right-click and choose *Done* to terminate the command.

fsp synchronize

The `fsp synchronize` command provides an interactive environment for synchronizing FSP and layout databases. There are four types of changes which can be synchronized when an FSP database is transfer into layout:

- Pin Swaps
- Schedules
- Placement
- Reference Designator

The `fsp synchronize` command ignores any other changes made in the FSP or schematic databases and generates a report of differences. For example, addition of new components, addition or deletion of nets and terminations. These changes can be added into layout database by regenerating the schematics and updating the layout.

This command lets you choose the category in which you want to synchronize the FSP and layout databases.

Access Using

Menu Path: *Place – FPGA System Planner – Synchronize*

func

The `func` command is used in conjunction with `show element` to display information on a named object of type Function, and with `property edit` to locate the named object.

Related Topics

- [Displaying Information](#)

func Command Dialog Boxes

Depending on which commands you run `func` with, the following dialog boxes are displayed:

- Show Element
- Edit Property
- Show Properties

Related Topics

Displaying Information

1. Run the `show element` command.
2. Choose the appropriate object types in the Find filter.
3. Type `findprop <property name>` at the console window prompt.
The Show Element display window for the specified property appears.

Related Topics

- [findprop](#)

Editing Object Properties

1. Run the `property edit` command.
2. Choose the appropriate object types in the Find filter.
3. Type `findprop <function designator name>` at the console window prompt.
The Edit Property and Show Properties dialog boxes are displayed.
4. Edit the property. For additional information, see `property edit` in the Allegro PCB and Package Physical Layout Command Reference.

Related Topics

- [findprop](#)

funckey

The `funckey` command allows you to create a function alias using alpha-numeric keys. The tools support groupings of up to four alpha-numeric character keys for operation as a function alias. When keys operate as a function alias, you press the keys but you do not have to press `Enter` to execute the command(s). Be sure that your cursor is not active in the command window when executing the function alias.

As an example of a function alias, you can associate the alphanumeric characters `addl` with the `add line` command. When you type `addl`, the `add line` command becomes active.

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

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

To obtain the Defined Aliases/Funckey list of the aliases and function keys defined in your environment file, type `alias` or `funckey` at the command window prompt. You can also choose *Tools – Utilities – Aliases/Function Keys* from the menu bar.

The `unalias` command deletes aliases and function aliases.

`funckey <user-defined name> <command(s) to execute>`

user-defined name	Specifies your abbreviation, up to four alphanumeric characters, that executes a command. Be careful that you do not create a function alias with the same root as another, or you can never access the function alias with the longer name. For example, if you create two function aliases named <code>al</code> and <code>all</code> , you cannot access <code>all</code> .
command(s) to execute	Specifies the command(s) to be executed when you press the function key. When entering multiple commands, enclose them with quotation marks (" ") and separate them with semicolons (;).

Access Using

Menu Path: *Tools – Utilities – Aliases/Function Keys*

Related Topics

- [Creating a Function Alias](#)

Creating a Function Alias

To create a function alias for the current work session:

1. At the command window prompt, type `funckey`, a *user-defined name* up to four alphanumeric characters, and the command string to which you are applying the `funckey` command.

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

1. At the command window prompt, type the *user-defined name* to execute the specified command.

Examples

The following examples use the `funckey` command. After you define the function alias at the command window prompt, type only the *user-defined name* to execute the command(s).

- To create this function alias, run the `add connect` command. In the Options tab of the Control Panel, set the *Line Lock* direction to *Off*. Then type the following example at the command window prompt. To execute this command, type 0.

```
funckey 0 options lock_direction Off
```

- To create this function alias, run the `add connect` command. In the Options tab of the Control Panel, set *Line width* to 25. Then type the following example at the command window prompt. To execute this command, type 2.

```
funckey 2 options line_width 25
```

- To create this function alias, run the `add connect` command. In the Options tab of the Control Panel, set the *Line Lock* direction to 45. Then type the following example at the command window prompt. To execute this command, type 4.

```
funckey 4 options lock_direction 45
```

- To create this function alias, run the `add connect` command. In the Options tab of the Control Panel, set the *Line Lock* direction to 90. Then type the following example at the command window prompt. To execute this command, type 9.

```
funckey 9 options lock_direction 90
```

- To create this function alias, run an editing command such as `add connect` or `slide`. In the Options tab of the Control Panel, set the *Bubble* field to *Hug preferred*. Then type the following example at the command window prompt. To execute this command, type h.

```
funckey h options bubble_space Hug preferred
```

- To create this function alias, run an editing command such as `add connect` or `slide`. In the Options tab of the Control Panel, set the *Bubble* field to *Shove preferred*. Then type the following example at the command window prompt. To execute this command, type `s`.

```
funckey s options bubble_space Shove preferred
```

- To create this function alias, type the following example at the command window prompt. To execute this command, run the `place manual` command, choose an object to place, and type `m`.

```
funckey m pop mirror
```

- To create this function alias, type the following example at the command window prompt. To execute this command, type `p`. The editor activates the `place manual` command and also hides the Placement dialog box. You can now move components.

```
funckey p "place manual; setwindow form.plc_manual; FORM plc_manual hide"
```

- To create this function alias, type the following example at the command window prompt. To execute this command, run the `place manual` command, choose an object for placement, and type `r` to rotate the specified object by 90 degrees.

```
funckey r iangle 90
```

- To create this function alias, type the following example at the command window prompt. To execute this function alias, press `+` to increment the active subclass to the next subclass.

```
funckey + subclass ++
```

- To create this function alias, type the following example at the command window prompt. To execute this function alias, press `-` to decrement the active subclass to the previous subclass.

```
funckey - subclass --
```

- To change *Persistent Snap* values during command execution, create the following function alias. These function keys can be used during command execution to set the persistent snap value *Off*, *Pin*, or *Via*.

```
funckey o "pop dyn_option_select 'Snap pick to@:@Persistent snap (Pin)@:@Off';pop  
dyn_option_select 'Snap pick to@:@Persistent snap (Via)@:@Off'"
```

```
funckey p "pop dyn_option_select 'Snap pick to@:@Persistent snap@:@Pin'; pop  
dyn_option_select 'Snap pick to@:@Persistent snap (Via)@:@Off' ; prepopup 0 0; pop  
dyn_option_select 'Snap pick to@:@Persistent snap@:@Pin'"
```

```
funckey v "pop dyn_option_select 'Snap pick to@:@Persistent snap@:@Via'; pop
```

```
dyn_option_select 'Snap pick to:@Persistent snap (Pin)@:@Via';pop  
dyn_option_select 'Snap pick to:@Persistent snap@:@Via''
```

Related Topics

- [funckey](#)
- [subclass](#)

