cadence®

G Commands

Product Version 23.1
September 2023

Contents

1

G Commands

gate_assign	gb_to_tape	gbplot
gds verification	generaledit	genfeedformat
genrad	gerber processing	gloss
gloss area design	gloss area highlight	gloss area list
gloss area room	gloss area window	gloss execute
gloss param	gloss param fillet	graphic edit
grid toggle	group	group add
groupedit	guideport	

gate_assign

The `gate_assign` batch command assigns functions and reference designators to components. The `gate_assign.log` file discloses any errors encountered during processing.

```
gate_assign

Existing layout file name (*.brd):

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

Existing layout	The name of the drawing to which you are assigning functions.
Output layout	The optional argument for the name of the drawing that contains assigned functions. Identify the file names (the `.brd` extension is optional).

If you do not supply outdrawing information, `gate_assign` writes over the existing layout file.

Assigning Functions and Reference Designators to Components

To assign functions and reference designators to all the components in a design in a batch mode, do the following:

1. Run the `gate_assign` command in the command prompt.

2. After processing is completed, use a text editor to verify the `gate_assign.log` file for errors.

3. Correct any errors and run the `gate_assign` command again.

Repeat this process until the log file shows no errors.

gb_to_tape

The `gb_to_tape` batch command formats a file containing Gerber data so it can be read by a Gerber photoplotter, and places it on a nine-track tape.

gbplot

The `gbplot` batch command uses Gerber photoplot files created from a design to create the `.plt` and `.ctl` files that are used as input for hp_plot . Use this file to generate plots for your artwork files. The gbplot command creates a `gbplot.log` file, which lists the aperture table and photoplot parameters used, plus any errors and warnings generated during execution.

Before executing the `gbplot` command, the artwork aperture and parameter files must be accessible through the ARTPATH environment variable.

```
gbplot artwork_file_name [penplot_file_name]  [-version]
```

gbplot artwork_file_name	The name of the existing Gerber artwork file, required for processing. The `.art` extension is not required with the file name. If you do not enter an artwork file name, or the name you enter cannot be found, you are prompted for it.
penplot_file_name	Optional. Specifies the name of the output file. If you do not enter an output file name, the `gbplot` command generates an `artwork_file_name.plt` and an `artwork_file_name.ctl`.
-version	Prints the version.

Examples

- This example reads the Gerber photoplot file `layer_1.art` and creates the IPF file `layer_1.plt` and the control file `layer_1.ctl`.

  gbplot layer_1

- This example reads the Gerber photoplot file `layer_1.art` and creates the IPF file `plot1.plt` and the control file `plot1.ctl`.

  gbplot layer_1 plot1

gds verification

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

The `gds verification` command verifies the accuracy of GDSII manufacturing data based on specified parameters. The verification is based on the GDSII file, the layer conversion file, and stream out options. The stream out options saved in the design are picked by default.

The command also allows you to clean any existing geometry data.

⚠ When the command is run on a design, any previous geometry is cleaned by default.

Related Topics

- layer compare
- Verifying Accuracy of GDS Data

GDSII Verification Dialog Box

Access Using

Menu path: *Si Layout – GDS Verification*.

GDSII Data	
GDS Data	Specify the GDS file against which the design geometries should be compared.
Layer conversion file	Specify the layer conversion file used to generate the original GDS data.
Stream Settings used for GDS Output	
Minimum segments per circle	Specify the vectorization parameter to be used. By default, if existing, the value saved in the design is used. The default is 64 in the absence of any saved values.
Vectorizing type	Specify the vectorizing type. By default the value is *Default* or either of *Inside Metal* or *Outside Metal* depending on any saved value.
Rotation	Specifies the rotation.
Mirroring	Specify the mirroring option.
Options	
Check Layers	Specify the layers to be compared. The allowed values are *All* (default) or *Stack-up*.
Detail level	Specify the level of details to be stored in the design. Allowed values are *Full*, *Local*, and *Difference-only* (default). Specifying Local will not store any data but generate a report.
Run	Click to run the verification. This will close the window and open a file viewer displaying the resulting file for easy checking.
Clean	Click to remove all difference information stored in the design.

Related Topics

- layer compare

Verifying Accuracy of GDS Data

Verifying that the geometries in the design are accurate in terms of any differences with the GDS files used important. To check for any differences between the design and the GDS file, do the following:

1. Choose *Si Layout – GDS Verification*.

   Alternatively, you can also type `gds verification` in the Command window. The GDSII Verification window opens.

2. Specify the GDS data file and layer conversion file used to generate the original GDS data in the design.

3. Check if the default values need to be changed for any parameters such as minimum segments and vectorizing type.

4. Specify the layers and detail level of the report.

   Specify *Local* for detail level if you want a report but do not want to store the differences.

5. Clean previous difference information by clicking Clean.

6. Start verification by clicking Run.

   A file viewer will open displaying the differences. A report will be created if you had specified Local.

Related Topics

- layer compare

- gds verification

generaledit

The generaledit command enables the default general-edit application mode that lets you perform editing tasks, including place and route, as well as moving, copying, or mirroring, for example. An application mode provides an intuitive environment in which commands used frequently in a particular task domain, such as etch editing, are readily accessible from right-mouse-button pop-up menus, based on a selection set of design elements you have chosen.

This customized environment maximizes productivity when you use multiple commands on the same design elements or those in close proximity in the design. Application mode configures your layout editor for a specific task by populating the right-mouse-button pop-up menu only with commands that operate on the current selection set.

In conjunction with an active application mode, your layout editor defaults to a pre-selection use model, which lets you choose a design element (noun), and then a command (verb) from the right-mouse-button pop-up menu. This pre-selection use model lets you easily access commands based on the design elements you've chosen in the design canvas, which the layout editor highlights and uses as a selection set, thereby eliminating extraneous mouse clicks and allowing you to remain focused on the design canvas.

Use *Setup – Application Mode – None* (noappmode command) to exit from the current application mode and return to a menu-driven editing mode, or verb-noun use model, in which you choose a command, then the design element.

Access Using

- Menu path: *Setup – Application Mode– General Edit*

- Toolbar icon:

Related Topics

- Allegro User Guide: Getting Started with Physical Design

- noappmode

Accessing Command Help for Right Mouse Button Options within an Application Mode

To access Command Help within Application Mode, perform the following steps:

1. Choose *Setup – Application Mode– General Edit.*
   Alternatively, type `helpcmd` in the console window. The *Command Browser* dialog box appears.

2. Enable the *Help* radio button at the top of the dialog box to place the browser in Help mode.

3. Scroll the command list and select (double-click) the command you want help on.
   The command documentation displays in the Cadence Help documentation browser momentarily.

genfeedformat

The `genfeedformat` batch command generates feedback files to back annotate the changes into schematic design. The command generates netlist for differ depending on the schematic design application.

```
genfeedformat  [-s schematic_name>] [-b <.brd_name>] [-version]
```

-b	The layout board name
-o	Output directory (optional). If a project file is specified using the -proj option, places the files in the directory specified by the "view packager" global directive in the project file. If no project file is specified, the output files are placed in the current working directory. The -o argument overrides the project file.
-m <mode>	Outputs chips data (chipsview.dat). <mode> =1: Generates normal multi-pin slot chips file <mode> = 2: Generates single pin slot chips file
-c	Output Electrical data (cmdbview.dat)
-version	Prints the version.
-proj <project_file_name	The HDL project file (HDL only)

Obsolete arguments (for SCALD)

-d	Update dependency table (SCALD only)
-t	Name map file update
-x	Use Packager-XL (defaults to Package -XL)
-s	Schematics directory (SCALD only)

genrad

The `genrad` batch command provides an interface between Cadence layout editor and the GenRad Test Workstation. This command extracts a circuit description source file from your layout editor's database.

To create an output file that works with the GenRad tester, certain requirements must be observed while creating the design drawing:

- Element names restricted to nine characters or less.

- Component device labels that conform to those listed in the GenRad manuals.

- Component value labels that consist of a floating point value and one of the GenRad scale factor abbreviations.

- Component tolerance labels to be used with the component value label.

- The accepted format is "+n%" for symmetrical tolerances, or "+n%,-n%" for non symmetrical tolerances, where n is a digit from 1 to 99.

- Pin names for the component device label types: CP, CP1, CR, VZ, QN, QP, NJFET, PJFET, SCR, GD.

```
genrad [-l|-q|-v][-o outputfile][-m model_number] [-version]design_name
```

-l	Outputs a long listing to the log file.
-q	Suppresses messages generated during execution of the extract portion of the program.
-v	Displays all messages generated during execution of the extract portion of the program.
-o outputfile	Identifies the name of the output file, and appends the extension `.ckt`. If this option is not used, the command assigns the name design_name . ckt to the file.
-m model_number	Identifies the number of the GenRad tester with which the output file is used. The legal options are 2270 , 2271 , 2272 , 2275 , 2276 . The default is 2270 .
-version	Prints the version.
design_name	A required field that provides the name of the design from which the information is to be extracted.

gerber processing

The *gerber processing* command adds and deletes sequence numbers and changes the end-of-block (EOB) character used in the Gerber file. The command opens the *Gerber File Processing* dialog. Enter the values required for Gerber file processing and choose Execute to process the file. The layout editor stores the results in a temporary file. Press Done to save the results in the file you specified as the Output file.

Gerber File Processing Dialog Box

General Parameters		
	Input File	Identifies the file to be processed.
	Output File	Identifies the name of the file that is created during processing.
Sequence number Processing		
	Process sequence numbers	Indicates whether sequence numbers are processed.
	Add sequence numbers	Specify to add sequence numbers
	Delete sequence numbers	Specify to delete sequence numbers
	Sequence format	
End-of-Block char processing		
	Change EOB character	Indicates whether the end-of-block character are changed during processing.
	Current eob char	Specify the current value of EOB character.
	New eob char	Leaving this option blank, the layout editor interprets the blank as a null string and places the new end-of-block character at the end of each line.

gloss

The `gloss` batch command executes the automatic glossing program. If you are running a complete execution of line and via cleanup, batch mode is most efficient.

Before running this command, open your design and use the `gloss param` command to complete the appropriate parameter forms. You can also use that command to run glossing interactively.

Syntax `gloss <design>.brd [<new_design>] & [-version]`

<design>.brd	Name of the design you want to gloss.
<new_design>	Name of an optional output file.
&	The ampersand (&) causes glossing to run in the background.
-version	Prints the version.

Related Topics

- gloss param
- Allegro User Guide: Routing the Design

gloss area design

The `gloss area design` command lets you choose the area that the route keepin defines (Design is the default glossing area).

> ⚠ To exclude an area from glossing, enclose the area of the design with a no-gloss polygon. A no-gloss polygon is a shape on class Manufacturing, which you can place in any of the following subclasses:

- NO_GLOSS_TOP
- NO_GLOSS_BOTTOM
- NO_GLOSS_ALL
- NO_GLOSS_INTERNAL

To prevent net changes during glossing and designate nets that require special treatment, assign the following properties:

- NO_GLOSS prevents automatic glossing applications from changing a net.
- FIXED prevents all automatic routines from changing a net.

Access Using

- Menu path: *Route – Gloss – Design*

Related Topics

- gloss area room
- gloss area window
- Routing the Design
- For additional information, see the *Routing the Design* user guide in your documentation set.

gloss area highlight

The `gloss area highlight` command lets you choose individual nets or components for glossing. Select the nets or components in the design canvas and run the command.

Access Using

- Menu path: *Route – Gloss – Highlighted*

Related Topics

- gloss area list
- Allegro User Guide: Routing the Design

Highlighting Selected Nets

You can also use `gloss area highlight` command to gloss a few selected nets. Before running this command you must define *Route Keepin* areas.

1. Highlight the nets for glossing, either with the `assign color` command or `highlight` command.

2. Select *Route – Gloss – Highlight*.
   Alternatively, type `gloss area highlight` in the Command window.

3. Select area to gloss.

4. Select *Route – Gloss – List*.
   Displays the LIST AREA dialog box showing the current glossing mode and the areas selected for automatic glossing.

5. Select *Route – Gloss – Parameters*. Select *Gloss* in the *Gloss Controller* dialog box.

Related Topics

- gloss area list
- Allegro User Guide: Routing the Design

gloss area list

The `gloss area list` command displays the LIST AREA dialog box showing the current glossing mode and the areas selected for automatic glossing.

Access Using

- Menu path: *Route – Gloss –List*

Related Topics

- Allegro User Guide: Routing the Design
- gloss area highlight

gloss area room

The `gloss area room` command lets you designate a room for glossing.

Access Using

- Menu path: *Route – Gloss –Room*

Related Topics

- Allegro User Guide: Routing the Design
- gloss area highlight

Designating a Room for Glossing

Glossing a part of design can be done by running the gloss action within a pre-defined room.

1. Choose *Route – Gloss –Room.*
   Alternatively, type `gloss area room` in the Command window.
   The Room browser appears, which lists rooms defined in the design.

2. Select a room name from the list and click *OK*.

Related Topics

- gloss area room
- gloss area highlight
- add rect

gloss area window

The `gloss area window` command lets you define an area to gloss by making two diagonal selections.

Access Using

- Menu path: *Route – Gloss –Window*

Related Topics

- gloss
- gloss area highlight
- gloss area list

Defining an Area for Glossing

For glossing a part of design, you can define an rectangular area using the following steps:

1. Choose *Route – Gloss –Window.*
   Alternatively, type `gloss area window` in the Command window.

2. Click to define one corner of a rectangular window.

3. Slide the cursor to expand the window and click again to define the diagonally opposite corner.

4. Repeat the process for each window you want to include in the glossing area.

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

Related Topics

- gloss

- gloss area highlight

- gloss area list

gloss execute

The `gloss execute` command runs the glossing routines you specify in the *Glossing Controller* dialog box that you open with the `gloss param` command. This command creates a log file `gloss.log` which contains warnings and errors, if any, encountered during processing.

You can run this command interactively from the application menu options or from the console window prompt. If you are glossing a small area of the board or running one of the faster types of gloss, running it in the graphic window completes fairly quickly. For a complete execution of line and via cleanup, however, running it in batch mode is probably more efficient.

Before executing the `gloss execute` command, set all the required parameters, NO_GLOSS properties and NO_GLOSS areas, and indicate which area of the design you want to gloss, then choose Execute .

```
gloss <layout> [<
new_layout
>]
```

<layout>	Name of the layout you want to gloss.
<new_layout>	Name of an optional output file.

Related Topics

- gloss param

gloss param

The `gloss param` command displays the *Glossing Controller* dialog box that determines which glossing applications are run. You can also run glossing from this dialog box. Each application brings up its own parameter dialog box, all of which are described here.

Before glossing, set the NO_GLOSS properties and areas and specify the design area to gloss with `gloss area design`, `gloss area room`, `gloss area window`, or `gloss area highlight`. The `gloss area list` displays the currently selected glossing area.

Related Topics

- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Adding Tapered Traces (Static or Dynamic Mode)
- Deleting Tapered Traces Interactively (Static Mode)
- Creating a Templates Parameter Set for Dielectric Generation
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Glossing Controller Dialog Box

Use this dialog box to access parameters for the following glossing applications and set the relevant glossing applications you want to run. You can also display the *Glossing Controller* dialog box from the Design Parameter Editor. Choose *Setup – Design Parameters* (`prmed` command) to access the Design Parameter Editor, click the *Route* tab, select the *Gloss* folder and click *View glossing applications*.

Access Using

- Menu path: *Route – Gloss – Parameters*

Application	
Line and via cleanup	Opens the Line and Via Cleanup Dialog Box that allows you to processes one net at a time, ripping up every connect line and via and rerouting it using a high via cost. If the rerouted path is an improvement, the new path replaces the existing one.
Via eliminate	Open the Via Eliminate Dialog Box that helps reduce the number of vias used in a design. You can specify used or unused pin escapes, standalone vias, and through vias.
Line smoothing	Opens the Line Smoothing Dialog Box that allows you to evaluate the existing route and removes unnecessary line segments and arcs. This application is a good tool to help open channels during routing.
Center lines between pads	Opens the Center Lines Between Pads Dialog Box that helps reposition line segments that pass between adjacent pins to make them equidistant between pins. This application should only be run after routing has been completed to 100% because it places connect lines off-grid in order to center them. This program runs quickly.
Improve line entry into pads	Opens the Improve Line Entry Into Pads Dialog Box that eliminates acute angles between connect lines and pads. This application changes the way lines enter a pad to eliminate acute angles and executes quickly. Previous executions might open up exits for the current execution, so this application executes repeatedly while it is successfully glossing pads. The task determines if it has previously glossed a pad and connect line pair and stops when the current execution finishes.
Line fattening	Opens the Line Fattening Dialog Box that allows you to widen connect lines wherever possible to improve reliability when the design is manufactured. Uses the limits you established in the DRC rules.
Convert corner to arc	Opens the Convert Corner to Arc Dialog Box that helps convert 45- and 90-degree corners to arcs. This feature is most useful with analog and flex circuits, particularly for high-voltage and high-speed circuits. The size and radius of the arc are determined by the values defined for the maximum and minimum radius.
Fillet and Tapered Trace	Opens the Fillet and Tapered Trace Dialog Box that helps reinforce potentially stressful connections with additional etch/conductor.
Dielectric generation	Opens the Dielectric Generation Dialog Box that automatically generates the dielectric material needed between intersecting connections for hybrid design.
Gloss	Saves the settings and runs the selected glossing applications.
Close	Saves the settings and closes the dialog box.

Related Topics

- prmed

- gloss area design

- gloss area room

- gloss area window

- gloss area highlight

- gloss area list

- Via Eliminate Dialog Box

- Line Smoothing Dialog Box

- Center Lines Between Pads Dialog Box

- Improve Line Entry Into Pads Dialog Box

- Line Fattening Dialog Box

- Fillet and Tapered Trace Dialog Box

- Dielectric Generation Dialog Box

- Choosing Glossing Applications

- Editing a Line Fattening Step

- Creating a Fattening Step from the Template

- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box

- Deleting an Item in the Fattening Steps Box

- Adding Fillets (Static or Dynamic Mode)

- Deleting Fillets Interactively (Static Mode)

- Creating a New Parameter Set for Dielectric Generation

- Eliminating Unused Blind and Buried Stack Vias

Line and Via Cleanup Dialog Box

This dialog box defines how the Line and Via Cleanup glossing application determines if a more efficient route can be made.

This is the only glossing application whose parameter settings are used by the automatic router when it runs the Cleanup Router. Therefore, you should establish these parameters before you run the automatic router.

The automatic router looks at this dialog box when it is organizing to route, and any parameters it does not find here, it takes from the *Automatic Router* dialog box.

Since cleanup runs the router, the following routing parameters must be established in the *Router Setup* tab of the *Automatic Router* dialog box (`auto_route`) before cleanup is run:

- *Wire grid*
- *Via grid*
- *Routing Direction*

Jog Size Limit	Specifies the maximum allowable size of a jog created by cleanup. The default value is -1, indicating no jog size limit.
Etch Length/Via	Specifies how much more etch/conductor length can be added to a connection to eliminate vias. The larger the number, the more etch/conductor that is added to avoid adding vias. The smaller the number, the more vias are added to shorten the lines. The default value is -1, indicating this field is not used. This field is used when you have selected Lines and Vias or Lines, Vias, and Missing Connects from the Cleanup All field near the bottom of this dialog box. After the cleanup of a net is complete, the etch/conductor-to-via ratio is computed. If this ratio exceeds this parameter value, the connections are returned to their pre-cleanup condition.
Net Length Limit	Skips editing any connection or net whose total etch length exceeds the value specified to speed cleanup by avoiding extremely long connections. The default value is 10000 mils, converted to the units of the drawing. A value of -1 indicates this parameter is unused.
Maximum 45 Length	Determines the maximum orthogonal length for a 45-degree angle. A recommended value is the size of the standard grid distance so undesirably long 45-degree line segments do not result. The maximum distance may be doubled in some cases because of back-to-back diagonals. For example, a value of 50 yields a 100 length diagonal. A maximum 45-degree routing parameter also extends 45-degree segments to eliminate either horizontal or vertical segments.
Slip Slide	Indicates whether the process can shift connections when necessary during cleanup. The connection is moved the minimum allowed by DRC. Disabled by default.
Cleanup Pin Escapes	Specifies whether pin escape lines and vias connected to SMD pins are ripped up and rerouted.
Minimum Via Limit	Defines the minimum number of vias a connection must have to be considered a candidate for cleanup.
Retry	Causes the router to try to route the connection again after a path has been ripped up. The connection is retried only if the first try was unsuccessful. If Retry is checked, the router continues to successively increase and retry until either the connection is completed or until window expansion goes beyond the limits of the design.B4
Number of Executions	Specifies the number of executions to process.
Cleanup All	The Lines option is slightly faster than Lines and Vias. The slowest combination is Lines, Vias and Missing Connects.
Lines	Causes each connect line of a net to be deleted and rerouted.
Lines and Vias	Causes each path in a net to be removed and rerouted.
Lines, Vias, and Missing Connects	Causes etch/conductor associated with a net to be removed and rerouted. If the final connections are better, they are saved. Better is defined as fewer missing connections, fewer vias, shorter etch/conductor length, and fewer jogs.

Related Topics

- gloss param

- Glossing Controller Dialog Box

- Via Eliminate Dialog Box

- Line Smoothing Dialog Box

- Center Lines Between Pads Dialog Box

- Improve Line Entry Into Pads Dialog Box

- Line Fattening Dialog Box

- Fillet and Tapered Trace Dialog Box

- Dielectric Generation Dialog Box

- Choosing Glossing Applications

- Editing a Line Fattening Step

- Creating a Fattening Step from the Template

- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box

- Deleting an Item in the Fattening Steps Box

- Adding Fillets (Static or Dynamic Mode)

- Deleting Fillets Interactively (Static Mode)

- Creating a New Parameter Set for Dielectric Generation

- Eliminating Unused Blind and Buried Stack Vias

- auto_route

Via Eliminate Dialog Box

This dialog box defines how you want the Via Eliminate glossing application to reduce the number of vias used in the design. The Via Eliminate routine reduces the number of vias in a design. You control the via types to be eliminated by selecting from the following options. You should run this glossing application selectively during the routing process and when the design is completely routed.
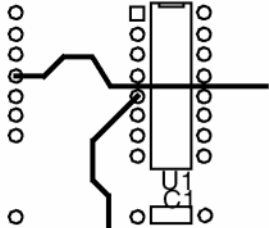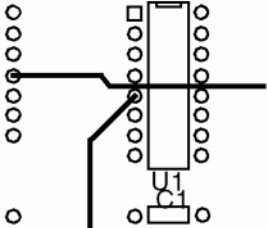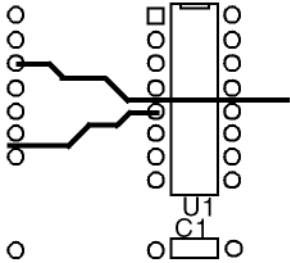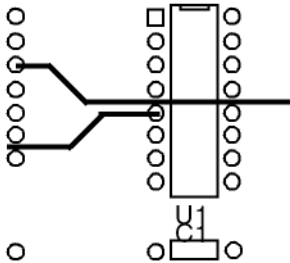
Eliminate used pin escapes	Specifies whether used pin escapes can be eliminated. A used pin escape is part of a defined net and has at least one connection to the escape v via elimination if the connect line on INTERNAL_5 can be moved to the TOP/SURFACE ETCH/CONDUCTOR subclass. When connections are r width defined on the new subclass.
Eliminate unused pin escapes	Specifies whether unused pin escapes can be eliminated. An unused pin escape has a defined net, but the only connection is to its own SMT pin.
Eliminate stand alone vias and unused pVias	Indicates whether standalone vias can be eliminated. A standalone via is not logically part of a net. Examples are a pin escape on a pin with no de
Eliminate regular through vias	Indicates whether regular through vias can be eliminated. A regular through via is a standard via in a connection attached to a pin with a defined n total number of vias in a design.
Jog Size	Specifies the maximum allowable size of a jog that can be added during via elimination. The default value of -1 indicates no jog limit.
Eliminate unused stacked vias	Removes unused blind and buried vias in an array of stacked vias, which can be identified by running the Unused Blind/Buried Via Report. An unu etch/conductor connection points, as shown below. A connection point is defined as a via connected to a cline, shape, or pin. Typically orphaned due to modifications that occur during routing or clip- at the via site. No removal occurs when: The net has the NO_GLOSS property. The net, symbol (if via is part of a symbol), or via has the FIXED pro to pins (pin escape rule).

Related Topics

- gloss param

- Glossing Controller Dialog Box

- Line Smoothing Dialog Box

- Center Lines Between Pads Dialog Box

- Improve Line Entry Into Pads Dialog Box

- Line Fattening Dialog Box

- Fillet and Tapered Trace Dialog Box

- Dielectric Generation Dialog Box

- Choosing Glossing Applications

- Editing a Line Fattening Step

- Creating a Fattening Step from the Template

- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box

- Deleting an Item in the Fattening Steps Box

- Adding Fillets (Static or Dynamic Mode)

- Deleting Fillets Interactively (Static Mode)

- Creating a New Parameter Set for Dielectric Generation

- Eliminating Unused Blind and Buried Stack Vias

Line Smoothing Dialog Box

This dialog box defines how the *Line Smoothing* glossing application removes unnecessary line segments and arcs.

Line Smoothing Eliminate	
Bubbles	Specifies whether *Line Smoothing* attempts to eliminate connect lines that have a 45-degree line segment, followed by an orthogonal segment, followed by another 45-degree segment that slopes in the opposite direction to the first 45 segment as shown in the following example. This etch/conductor configuration can result from via elimination. *Line Smoothing* is a tool that smooths bubbles configured around pads that are no longer in the design. Disabled by default.
Jogs	Specifies the elimination of repeated jogs. The jogs are removed by combining two or more into a single jog, as illustrated in the following example: Disabled by default.
Dangling lines	Specifies whether *Line Smoothing* eliminates connect lines without two owners. These lines are usually connected to a pin, via, or T junction on one end and unconnected on the other. Disabled by default.
No-net dangling lines	Specifies whether *Line Smoothing* eliminates connect lines without an associated net name.

Line Smoothing Line Segments	
Preserve odd angle lines if possible	Specifies to preserve odd angle lines (unless removing them shortens the connection). By default, these segments are removed.
Convert 90's to 45's	Indicates whether *to* convert all 90-degree angles in the design to 45-degree angles. This is a quick method for mitering a 90-degree design. Enabled by default.

Extend 45's	Examines each 45-degree segment between horizontal and vertical segments. It attempts to extend the 45-degree segment such that either the horizontal or the vertical segment can be eliminated. The result of running this option is shown in the following example: Before 45 extension After 45 extension Enabled by default.
Maximum 45 Length	Specifies the maximum orthogonal distance to which a 45 degree angle segment extends. The default value is *-1* and indicates no limit.
Length Limit	Limits the maximum length of line segments that are to be considered by Line Smoothing. Bubbles are processed if the orthogonal segment in the bubble is less than or equal to the value of this parameter. Diagonals whose orthogonal length of the diagonal is longer than this value are skipped. Jogs are only considered if the orthogonal segment in the jog is less than or equal to this limit. The default value is *-1* and indicates no length limit.
Corner Type	Specifies whether corners should be diagonal (*45*) or orthogonal (*90*). The default is *45*.
Number of Executions	Specifies the number of executions. The smoothing operation works best if it is executed three or four times. Each execution considers a line one time. Cadence recommends running multiple executions. The default value is *1*.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Via Eliminate Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Center Lines Between Pads Dialog Box

This dialog box specifies how the Center Lines Between Pads glossing application adjusts connect lines that pass between adjacent pins so they are equidistant from both pins. This option should only be run after routing has been completed to 100%, because it places connect lines off-grid in order to center them.

Minimum move size	Defines the minimum distance that glossing can move a line. When processing a group of lines that pass between two pins, if any of the lines is to be moved a distance less than this minimum, none of the lines in the group is moved. The default value is *2 mils* (expressed in the units of the drawing).
Adjacent pad tolerance	Defines the maximum center-to-center distance between two adjacent pins (measured horizontally or vertically) that affect line centering. Centering operations do not occur on pins that are greater than the maximum distance. The default is *100 mils* (expressed in the units of the drawing)
Corner type	Options specify whether corners should be diagonal (45) or orthogonal (90). The default is *45*.
Line spacing	Options define line-to-line spacing between pads as follows:
Minimum	Spaces the lines at the minimum line-to-line spacing and divides the remainder of the space evenly between the outermost lines and the pads. If this causes a DRC error, the lines are not centered (default).
Even	Spaces the lines so that they are equally distant from one another and from the pins. If this causes a DRC error, then they are reprocessed as Minimum line spacing.
Gloss layers	Opens the Glossing Subclasses Dialog Box, described below, that controls which layers are glossed with this routine.

Glossing Subclasses Dialog Box

This dialog box lets you add and delete layers for glossing with the Center Lines Between Pads glossing application.

Add	Choose an existing layer to add to the glossing routine.
Delete	Choose an existing subclass layers to delete from the glossing routine.
Maximum 45	Obsolete.

Related Topics

- gloss param

- Glossing Controller Dialog Box

- Line and Via Cleanup Dialog Box

- Via Eliminate Dialog Box

- Line Smoothing Dialog Box

- Improve Line Entry Into Pads Dialog Box

- Line Fattening Dialog Box

- Fillet and Tapered Trace Dialog Box

- Dielectric Generation Dialog Box

- Choosing Glossing Applications

- Editing a Line Fattening Step

- Creating a Fattening Step from the Template

- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box

- Deleting an Item in the Fattening Steps Box

- Adding Fillets (Static or Dynamic Mode)

- Deleting Fillets Interactively (Static Mode)

- Creating a New Parameter Set for Dielectric Generation

- Eliminating Unused Blind and Buried Stack Vias
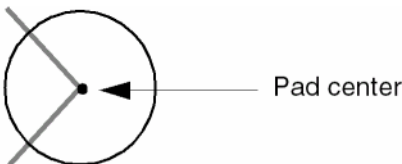
Improve Line Entry Into Pads Dialog Box
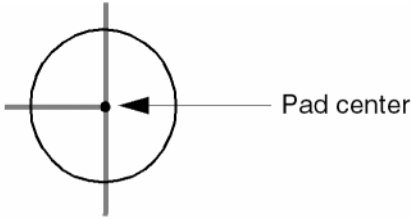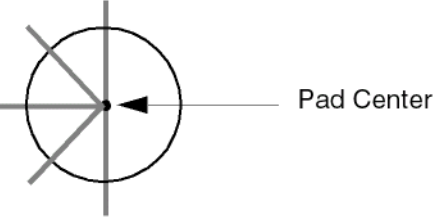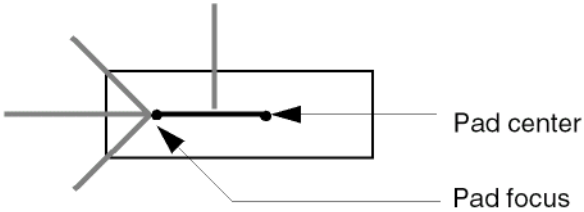
This option on the *Glossing Controller* dialog box eliminates acute angles that automatic routing creates between connections and the edge of the pad. Options change the way lines enter a pad to eliminate acute angles.
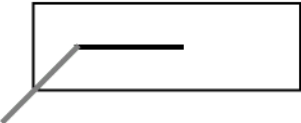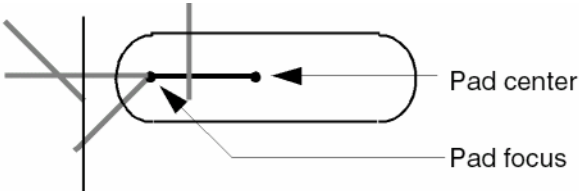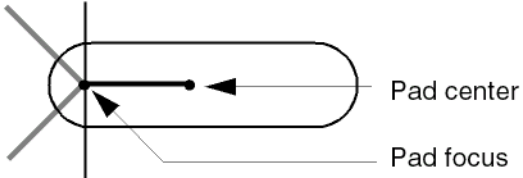
The following figure shows the results of running this option.
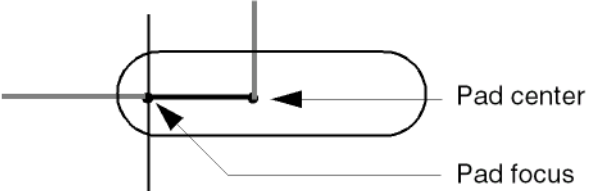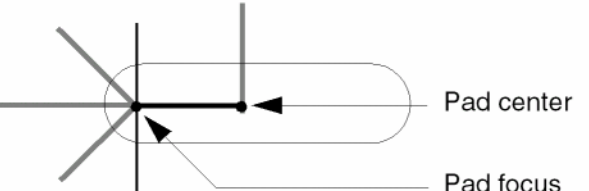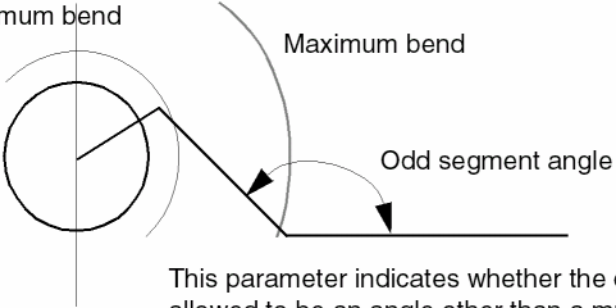
Improving Line Entry Into Pads

Before line entry improvement   After line entry improvement

Pads to Process Parameters	Choose the types of pad glossed during processing. Any combination of shapes can be selected. Enabled by default.	
Bend Distance	Defines how far from the edge of the pad a line exiting the pad must be before bending.	
	Minimum	Defines the minimum distance from the edge of a pad to the first bend outside the pad in a line connected to the pad. The default value is 12 mils (expressed in the units of the drawing).
	Maximum	Defines the maximum distance from the edge of a pad to the first bend outside the pad in a line connected to the pad. The default value is 1000 mils (expressed in the units of the drawing).
Circular Pad Entry Parameters	The processing for a circular pad focuses lines connected to a pad to enter the pad along a radius of the pad. The choices are in terms of the angle of entry: 45 degrees only, 90 only, either 45 or 90, or any angle.	
	45 only	Focuses entry into the pad on an angle that is 45 degrees out of phase with the long direction of the pad, as shown in the following example: Grey lines indicate directions of entry allowed to a circular pad with the *45 only* option selected.

	90 only	Focuses entry into a circular pad on an angle that is either in line with the lengthwise center of the pad or perpendicular to that line, as shown in the following example: Grey lines indicate directions of entry allowed to a circular pad with the *90 only* option selected.
	45 or 90	Focuses entry into the pad from either a 45- or a 90-degree angle, as shown in the following example: Grey lines indicate directions of entry allowed to a circular pad with the *45 or 90* option selected.
	Any Angle	Focuses entry into the pad from any suitable angle.
Square Pad Entry Parameters		Lines connected to a square pad are forced to enter the pad either perpendicular to a side of the square or diagonally through the corner of the square.
	Side	Indicates whether lines can enter perpendicular to a side of the square.
	Corner	Indicates whether lines can enter diagonally through the corner of the square.
Rectangular Pad Entry Parameters		On a rectangular pad, all lines are forced to enter at a right angle to a side of the rectangle or diagonally through a corner to the focus point, then travel to the pad center. A focus point is the point at which diagonal lines entering from the corners adjacent to the short side of the rectangle would intersect inside the rectangle. Grey lines indicate directions of entry allowed to a rectangular pad.

	Long	Indicates whether entry is allowed into the long side of the rectangle.
	Short	Indicates whether entry is allowed through the short side.
	Corner	Indicates whether entry through the corner is allowed.
Oblong Pad Entry Parameters		Lines that enter through the round end of an oblong pad are made to enter the pad with a segment that is some multiple of 45 in relation to the plane that travels lengthwise through the pad center and focus. If a line enters through one of the straight sides of the pad, it is forced to be at a right angle to the pad edge.
	Side	Indicates whether entry through the straight side is allowed.
	Round	Indicates whether entry through the round end is allowed.
	Angle Selection Box	Defines whether the entry angle is for an oblong pad. Be aware that some combinations of parameter choices do not result in a successful exit. For example, a side entry at 45 only is not possible. The default value is 45 or 90.
	45 only	Defines entry into the pad on an angle that is 45 degrees out of phase with the long direction of the pad.

	90 only	Defines entry into the pad on an angle that is either in line with the lengthwise center of the pad or perpendicular to where that line would be: Grey lines indicate directions of entry allowed to an oblong pad.
	45 or 90	Defines entry into the pad from either a 45- or a 90-degree angle: Grey lines indicate directions of entry allowed to an oblong pad.
	Any Angle	Indicates that entry into the pad can be from any suitable angle.
Create Odd Angle Segments		This option defines the segment entering the pad. Some of the techniques in the line-to-pad glossing process create new segments before the segment that enters the pad. This field specifies whether these segments are allowed to be any angle other than multiples of 45 degrees. The default is unselected (Off). This parameter indicates whether the odd segment is allowed to be an angle other than a multiple of 45.
	Corner Type	Options specify that the corners created by cutting and moving a line segment are to be either diagonal (45) or orthogonal (90). The default is 45. When 45 is selected, you can choose the maximum length of the 45s. The default for this field is -1 or unlimited.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Line Fattening Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Line Fattening Dialog Box

Use this dialog box to set parameters for the Line Fattening glossing application, which increases the width of connect lines to improve reliability when the design is manufactured. You can create a unique set, with a maximum of four widths, for each etch/conductor subclass.

On first invocation, the *Fattening Steps* section displays a list of current line widths presented by subclass, plus a template category for all existing line widths.

Each item listed is a candidate for a fattening step. Define a fattening step by completing the fields in the *Step Parameters* section on the right side of the dialog box. Each line fattening step can have a maximum of four new widths. A template set can be created for one line width, then applied to each subsequent layer; or an individual set can be defined for any width and layer.

Table of Contents	Lists the current step and a list of all fattening steps for which step parameters sets have been defined.		
		Delete Steps	Lets you delete an item in the *Fattening Steps* box.
Add Step for Subclass	Lets you add a parameter set for a subclass not listed in the *Table of Contents.*		
Step Parameters	Defines parameter sets or edits existing sets.		
Step for Subclass	Lets you define the subclass you are setting the line widths for.		
Existing Segment Width	Indicates the current width of the specified line segment.		
Width Step	Indicates the new line width. You can enter up to 4 new line widths.		
Copy Template	Lets you copy a defined template to another subclass with a corresponding width.		

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Convert Corner to Arc Dialog Box

This dialog box specifies how the Convert Corner to Arc glossing application changes existing corners to arcs where ease of manufacturing is enhanced.
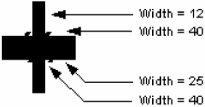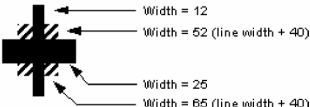
Maximum Radius	Specifies the largest radius used during execution. The default value is *25*.
Minimum Radius	Specifies the smallest radius used during execution. The default value is *2*.
Number Executions	Identifies how many times the Convert Corner to Arc application is run. The default value is *1*.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Dielectric Generation Dialog Box

Use the *Dielectric Generation* dialog box to define the size of one or two dielectric areas, depending on the number needed in the design. When the Dielectric Generation glossing application is executed, dielectric patches are placed between intersecting connections.

Table of Contents	Lists the etch/conductor layers for which dielectric parameter sets have been defined. When you initially open the dialog box, the only entry on the Table of Contents is TEMPLATES, because no parameter sets have been defined.
Delete Parameter Set	Click to delete a selected parameter set.
Pick to Add Parameter Set for Trace Layer	Select a parameter. This parameter becomes the name of the *Parameter Set* and the selection for the *Trace Layer*, *Crossover Layer*, and *First Dielectric Layer* fields.
Parameter Set for Trace Layer	Defines the original parameter sets or changes existing sets.The name in the *Trace Layer*, *Crossover Layer* and *First Dielectric Layer* reflects the Current Parameter Set.
First Dielectric	Specifies the size of the first dielectric patch. Enter an X and Y value.
Second Dielectric	Specifies the size of the second dielectric patch. Enter an X and Y value.
Merge Dielectric Shapes	Turn on to merge dielectric shapes if necessary.
Copy Template	Lets you copy a defined template to another subclass with corresponding properties.
Incremental	Determines how the program applies the X and Y size values. Turn on Incremental to customize the patch at each intersection. By default, this option is not checked. When you choose this option, the process creates a patch that is the sum of the X value plus the width of the vertical connection, and the Y value plus the width of the horizontal connection. When the patch is placed, the value in the X and Y fields is divided equally on either side of the connection. This customizes the patch at each intersection. If you do not turn on this option, the process adds a dielectric patch of the size defined by the X and Y value at each intersection. The following illustrations show a patch created with the X and Y fields set to 40. Before the Incremental option is checked. After the Incremental option is checked.

Related Topics

- gloss param

- Glossing Controller Dialog Box

- Line and Via Cleanup Dialog Box

- Via Eliminate Dialog Box

- Line Smoothing Dialog Box

- Center Lines Between Pads Dialog Box

- Improve Line Entry Into Pads Dialog Box

- Line Fattening Dialog Box

- Convert Corner to Arc Dialog Box

- Fillet and Tapered Trace Dialog Box

- Choosing Glossing Applications

- Editing a Line Fattening Step

- Creating a Fattening Step from the Template

- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box

- Deleting an Item in the Fattening Steps Box

- Adding Fillets (Static or Dynamic Mode)

- Deleting Fillets Interactively (Static Mode)

- Creating a New Parameter Set for Dielectric Generation

- Eliminating Unused Blind and Buried Stack Vias

Choosing Glossing Applications

Multiple glossing applications can be accessed through the *Glossing Controller* dialog box. To choose an appropriate application perform the following steps:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. In the *Run* column, check the boxes of the applications you want to run.

3. To set parameters to control how an application functions, click the box to the left of the application name—for example, the *Line and via cleanup* application.
   A parameter dialog box for the application appears. See the descriptions of each dialog box earlier in this command.

4. Fill out each application's dialog box as required, and exit the dialog box.

5. Click *Gloss* to save the parameters and run glossing. –or– Click *Close* to save the parameters and close the dialog box.
   If you choose several applications, glossing runs them in the order that they appear in the dialog box.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Editing a Line Fattening Step

To edit a line fattening step, perform the following steps:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Click the box next to *Line fattening* to open the *Line Fattening* dialog box.

3. Choose the step to edit in the *Fattening Steps* box.
   The subclass affected by the step appears in the *Add Step for Subclass* field of the *Step Parameters* section. The existing line width values of the specified step appear in the *Existing Segment Width* field.

4. In the *Step Parameters* section, change the step subclass to the specified layer.

This changes the flattening step name.

1. Choose the existing line width value that you want to increase.

2. Change the *Width Step* values to the specified flattened widths.

3. Click *OK*.
   The widths are entered.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Creating a Fattening Step from the Template

Perform the following steps to create a fattening step from a given template:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Click the box next to *Line fattening* to open the *Line Fattening* dialog box.

3. Create a line fattening step of the required width for the TEMPLATES listing.
   For example, for a required width of 8, choose *TEMPLATES 8* from the *Fattening Steps* section.

4. Choose the next instance of that width to change.
   For example, TOP 8. Again, for this example the layout editor would display 8 in the *Existing Segment Width* field with the *Width Step* fields blank.

5. Click *Copy Template*.
   The values from the TEMPLATE set for 8 are filled into the *Width Step* fields.

6. Repeat steps 3 through 5 for each of the Fattening Steps to which you want TEMPLATES applied.
   While the dialog box is open, you can also delete an existing width step or add a width step for a line width not currently on the list.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box

To create a custom line fattening step for line width, perform the following steps:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Click the box next to *Line fattening* to open the *Line Fattening* dialog box.

3. Choose a subclass from the *Add Step for Subclass* field.

4. Type the new line width in the *Existing Segment Width* field.

5. Proceed as described in Editing a Line Fattening Step.
   If the new line width is to be used on several layers, each subclass listing must be defined separately or by first creating a TEMPLATE set.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Deleting an Item in the Fattening Steps Box

Perform the following steps to delete an item in the *Fattening Steps* dialog box:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Click the box next to *Line fattening* to open the *Line Fattening* dialog box.

3. Highlight the item you want to delete from the *Fattening Steps* box.

4. Click *Delete Step*.
   The *Reset Steps* button updates the list appearing in the *Fattening Steps* window to reflect any width changes made to connections since the dialog box was previously opened. This includes changes that occurred as a result of running the line fattening application or any interactive editing. Using this option generates a current list of subclasses and widths in preparation for creating new step parameter sets. Note that all the old step parameter sets are lost after clicking this button.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Adding Fillets in Static or Dynamic Mode

To add fillets in *Static Mode* or *Dynamic Mode*, perform the following steps:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Disable the *Run* buttons for all other applications in the *Glossing Controller* dialog box.

3. Click the box to the left of *Fillet and Tapered Trace*.
   The Fillet and Tapered Trace Dialog Box appears.

4. Complete the parameters and choose one of the following to generate fillets in static or dynamic mode:

   ○ Dynamic mode: When you initially enable the *Dynamic Fillets* option, the entire board updates with shape-based fillets. During subsequent interactive route editing, fillets are deleted and then regenerated on modified pins or vias, based on the specified parameters.

   ○ Static mode: When you disable the *Dynamic Fillets* option, and choose *Route – Gloss – Parameters*, click *Run* next to the Pad and T Connection Fillet on the *Glossing Controller*, and click *Gloss*, shape-based fillets are added in a batch update. Whenever you modify a pin, via, or cline, the layout editor deletes the fillets and does not regenerate them on the modified pin, via, or cline.

5. Click *OK*.

6. In the *Glossing Controller* dialog box, click *Run* next to *Pad and T connection fillet*.

7. Click *Gloss*.

8. Interactively check the results in the `gloss.log` file (generated in Static mode) or in the Missing Fillets Report (generated in Dynamic mode). The Missing Fillets Report lists the parameters used to generate fillets as well as information on missing and partial fillets, including net, item, location, and subclass. This report is also available by choosing *Tools – Reports*.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias
- reports

Deleting Fillets Interactively in Static Mode

Perform the following steps to interactively delete fillets in *Static Mode*:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Click the box to the left of *Line smoothing.*

3. Choose *Dangling Lines* in the *Eliminate* section and disable all other options.

4. Click OK to close the *Line Smoothing* dialog box.

5. Run `delete fillet`.
   The *Options* panel displays the active class and subclass, and the *Find* panel displays the active design elements: *Symbols, Nets, Pins, Vias, Clines*, and *Shapes*.

6. Choose the pin, via, or fillet instance to delete. To choose multiple elements, window select or right-click to choose *Temp Group.*

   ⚠ The Dynamic Fillets option on the *Pad and T Connection Fillet* dialog box must be disabled.

7. In the *Find* panel, deselect *Nets;* otherwise fillets on nets are excluded.

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

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias

Adding Tapered Traces in Static or Dynamic Mode

To add tapered traces in *Static Mode* or *Dynamic Mode*, perform the following steps:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Disable the *Run* buttons for all other applications in the *Glossing Controller* dialog box.

3. Click the box to the left of *Fillet and Tapered Trace*.

4. Complete the parameters and choose one of the following to generate fillets in static or dynamic mode:

   ○ Dynamic mode: When you initially enable the *Dynamic* option, the entire board updates with shape-based fillets. During subsequent interactive route editing, fillets are deleted and then regenerated on modified pins or vias, based on the specified parameters.

   ○ Static mode: When you disable the *Dynamic* option, and choose *Route – Gloss – Parameters*, click *Run* next to the *Fillet and Tapered Trace* on the *Glossing Controller*, and click *Gloss*, shape-based fillets are added in a batch update. Whenever you modify a pin, via, or cline, the layout editor deletes the fillets and does not regenerate them on the modified pin, via, or cline.

5. Click *OK*.

6. In the *Glossing Controller* dialog box, click *Run* next to *Fillet and Tapered Trace.*

7. Click *Gloss*.

8. Interactively check the results in the `gloss.log` file (generated in Static mode) or in the Missing Fillets Report (generated in Dynamic mode). The Missing Fillets Report lists the parameters used to generate fillets as well as information on missing and partial fillets, including net, item, location, and subclass. This report is also available by choosing *Tools – Reports*.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias
- reports

Deleting Tapered Traces Interactively in Static Mode

Perform the following steps to interactively delete tapered traces in *Static Mode*:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Click the box to the left of *Line smoothing.*

3. Choose *Dangling Lines* in the *Eliminate* section and disable all other options.

4. Click OK to close the *Line Smoothing* dialog box.

5. Run `delete fillet`.
   The *Options* panel displays the active class and subclass, and the *Find* panel displays the active design elements: *Symbols, Nets, Pins, Vias, Clines*, and *Shapes*.

6. Choose the pin, via, or fillet instance to delete. To choose multiple elements, window select or right-click to choose *Temp Group.*

   ⚠ The Dynamic Fillets option on the *Fillet and Tapered Trace* dialog box must be disabled.

7. In the *Find* panel, deselect *Nets;* otherwise fillets on nets are excluded.

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

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Creating a New Parameter Set for Dielectric Generation
- Eliminating Unused Blind and Buried Stack Vias
- reports

Creating a Templates Parameter Set for Dielectric Generation

To create a template parameter set for dielectric generation, perform the following steps:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Click the box next to *Dielectric generation* to open the *Dielectric Generation* dialog box.

3. Choose *TEMPLATES* in the *Parameter Sets for Trace Layer* box.
   TEMPLATES displays in the *Trace Layer* field on the right side of the form.

4. In the *First Dielectric* and *Second Dielectric* sections, complete the *Size* fields and choose *Incremental*, if necessary.

5. Choose *Merge Dielectric Shapes*, if necessary.

6. Click *OK*.

To define a different parameter set than TEMPLATES

- Follow the procedure above, substituting the proper subclass name.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Adding Tapered Traces (Static or Dynamic Mode)
- Deleting Tapered Traces Interactively (Static Mode)

Creating a New Parameter Set for Dielectric Generation

Perform the following steps to create a new parameter set for dielectric geeration:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Click the box next to *Dielectric generation* to open the *Dielectric Generation* dialog box.

3. From the *Pick to Add Parameter Set for Trace Layer* field, choose the etch/conductor subclass for which you are creating a parameter set.
   The name of the specified subclass appears in the *Trace Layer* field of the dialog box.

4. Choose the appropriate etch/conductor subclass from the *Crossover Layer* field.

5. Choose the appropriate etch/conductor subclass from the *First Dielectric Layer* field.

6. Click *Copy Template*.
   The *First Dielectric* and *Second Dielectric* fields with values found in the template are completed.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Adding Tapered Traces (Static or Dynamic Mode)
- Deleting Tapered Traces Interactively (Static Mode)
- Creating a Templates Parameter Set for Dielectric Generation

Eliminating Unused Blind and Buried Stack Vias

To eliminate unused blind and buried stack vias, perform the following steps:

1. Choose *Route – Gloss – Parameters*.
   Alternatively, type `gloss param` into the Command Window. The *Glossing Controller* dialog box appears.

2. Click the box to the left of *Via Eliminate*. The *Via Eliminate* dialog box appears.

3. Enable the *Eliminate unused stacked vias* option.

4. Click *OK*.

5. Click the *Run* box to the right of *Via Eliminate*.

6. Click *Gloss* to execute the program.
   The log file reports the number of vias that were eliminated.

Related Topics

- gloss param
- Glossing Controller Dialog Box
- Line and Via Cleanup Dialog Box
- Via Eliminate Dialog Box
- Line Smoothing Dialog Box
- Center Lines Between Pads Dialog Box
- Improve Line Entry Into Pads Dialog Box
- Line Fattening Dialog Box
- Convert Corner to Arc Dialog Box
- Fillet and Tapered Trace Dialog Box
- Dielectric Generation Dialog Box
- Choosing Glossing Applications
- Editing a Line Fattening Step
- Creating a Fattening Step from the Template
- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box
- Deleting an Item in the Fattening Steps Box
- Adding Fillets (Static or Dynamic Mode)
- Deleting Fillets Interactively (Static Mode)
- Adding Tapered Traces (Static or Dynamic Mode)
- Deleting Tapered Traces Interactively (Static Mode)
- Creating a Templates Parameter Set for Dielectric Generation

gloss param fillet

The `gloss param fillet` command defines the parameters for the fillet and tapered traces glossing application to place fillets of etch/conductor at junctions to reinforce connections. You can run this command on analog and high-speed circuits, or areas of a design where shock and vibration to the design might disrupt connections.

Fillets are also added at the point of cline width transition to reduce stress. The layout editor tapers the clines by adding fillets to prevent abrupt changes in line width which is very common in RF and Rigid Flex applications.The fillet capability is an automatic glossing function that helps establish and maintain strong connections by adding extra copper. Fillets are checked by automatic DRC.

⚠ Pad fillets are suppressed when the pad to be filleted is covered by another pad or by a static/dynamic shape.

Related Topics

- Generating Fillets and Tapered Traces

- gloss param

Fillet and Tapered Trace Dialog Box

Access Using

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

Global Options

Dynamic	Updates the entire board with shape-based fillets. During subsequent interactive route editing, fillets are deleted and then regenerated on modified pins or vias, based on the specified parameters (unless an element has the NO_FILLET property assigned). If disabled, shape-based fillets are added in a batch update. Whenever you modify a pin, via, or cline, the layout editor deletes the fillets and does not regenerate them.
Curved	Creates shape-based fillets or tapered traces using an arc instead of a line as part of the shape outline from the cline to the pad intersection.
Allow DRC	Creates fillets and taper traces even if DRCs result.
Unused Nets	Allows tapering and filleting on unused nets.

Objects Parameters

Objects	Choose options for the pad shapes: circular pads, square pads, rectangular pads, oblong pads, octagon pads, Complex pads, pads as shapes, pins, vias, bond fingers, pads without drills, and t connections. For circular, square, rectangular, octagon and oblong pads, you can indicate the maximum size for the fillet. The default size is *100 mils* (in drawing units).

Fillet Options

Fillet Objects	Specifies the object for fillet. Valid objects are Pins, Vias and Ts.
Desired angle	Specifies the angle created by the generated fillet shapes. The default value is *90 degrees*. A larger *Desired Angle* and a smaller *Max Offset* create a short fillet. A smaller *Desired Angle* and larger *Max Offset* create a long fillet. This option is not applicable for creating arc fillets.
Max angle	Specifies the maximum angle for the fillet. The default value is *90 degrees*. The maximum possible value is 99 degrees. This value must always be equal to or greater than the *Desired Angle*. A larger *Desired Angle* and a smaller *Max Offset* create a short fillet. A smaller *Desired Angle* and larger *Max Offset* create a long fillet. This option is not applicable for creating arc fillets.
Max offset	Specifies the maximum distance between the intersection of the pad edge and the connecting line, forming the fillet length. The default value is *25 mil* (in drawing units). This option is not applicable for creating arc fillets.
Max arc offset	Specifies the maximum distance between the pad edge and the point along the curved trace, forming the fillet length. The default value is *5 mil* (in drawing units).
Min arc offset	Specifies the minimum distance between the pad edge and the point along the curved trace, forming the fillet length. The default value is *1 mil* (in drawing units). Must always be lesser than the *Max arc offset*. You can fillet arc segments for only pins and vias. The fillets generated for arc segments always have curved lines.
Min line width	Specifies the minimum line width of the cline entering the pad. If the line width of the cline is less than the value specified here, the fillet is not created. The default value is *3 mil* (in drawing units).
Max line width	Specifies the maximum line width of the cline entering the pad. If the line width of the cline is greater than the value specified here, the fillet is not created. The default value is *25 mil* (in drawing units).

Tapered Traces Options

Tapered traces	Set to enable tapering of traces. Not on by default.
Min segment angle	Specifies the minimum segment angle. The default is 135 degrees. The smaller included angle should be greater than or equal to 90 degrees.
Desired Angle	Specifies the angle of taper created by the fillet. A larger *Desired Angle* and a smaller *Max Offset* create a short fillet. A smaller *Desired Angle* and larger *Max Offset* create a long fillet. The default value is *60 degrees*.
Max Offset	Specifies the maximum distance the fillet may extend from vertex to the point of cline width transition. The default value is *635 mil* (in drawing units).

Related Topics

- gloss param

Fillet and Tapered Trace Dialog Box1

Run this option on analog and high-speed circuits, or areas of a design where shock and vibration to the design might disrupt connections. This dialog box defines the parameters for the Fillet and tapered traces glossing application to place fillets of etch/conductor at junctions to reinforce connections.

As design density increases, pad sizes and line widths decrease, creating potential breakout when through-holes are drilled. A fillet is a triangular area of etch/conductor placed at junctions to reinforce connections. Adding traces at these junctions helps prevent signal failure. Additionally, sharp corners can be eliminated on high-voltage designs by adding fillets to T intersections.

A T is an intersection of three or more lines. A fillet can be formed between any two of the lines in a T that intersect at an angle of 90 degrees and less and where the lines are drawn at an angle divisible by 45. In the most common case, there are two intersections of 90 degrees for which fillets are formed.

Fillets are also added at the point of cline width transition to reduce stress. The layout editor tapers the clines by adding fillets to prevent abrupt changes in line width which is very common in RF and Rigid Flex applications.The fillet capability is an automatic glossing function that helps establish and maintain strong connections by adding extra copper. Fillets are checked by automatic DRC.

Global Options

Allow DRC	Creates fillets and taper traces even if DRCs result.
Dynamic	Updates the entire board with shape-based fillets. During subsequent interactive route editing, fillets are deleted and then regenerated on modified pins or vias, based on the specified parameters (unless an element has the NO_FILLET property assigned). **Note:** The *Dynamic* option is disabled in PCB L Performance. Opening a design in Orcad PCB L or PCB L Performance disables *Dynamic* if it had been enabled. If disabled, when you choose *Route – Gloss – Parameters*, click *Run* next to *Fillet and tapered traces* on the *Glossing Controlle*r, and click *Gloss*, shape-based fillets are added in a batch update. Whenever you modify a pin, via, or cline, the layout editor deletes the fillets and does not regenerate them.
Curved	Creates shape-based fillets or tapered traces using an arc instead of a line as part of the shape outline from the cline to the pad intersection.
Unused Nets	Allows tapering and filleting on unused nets.

Objects Parameters

Choose from the following options for the pad shapes: circular pads, square pads, rectangular pads, oblong pads, octagon pads, complex pads, pads as shapes, pins, vias, bond fingers, pads without drills, and t connections. For circular, square, rectangular, octagon and oblong pads, you can indicate the maximum size for the fillet. The default is *100 mils* (expressed in drawing units).

Fillet Options

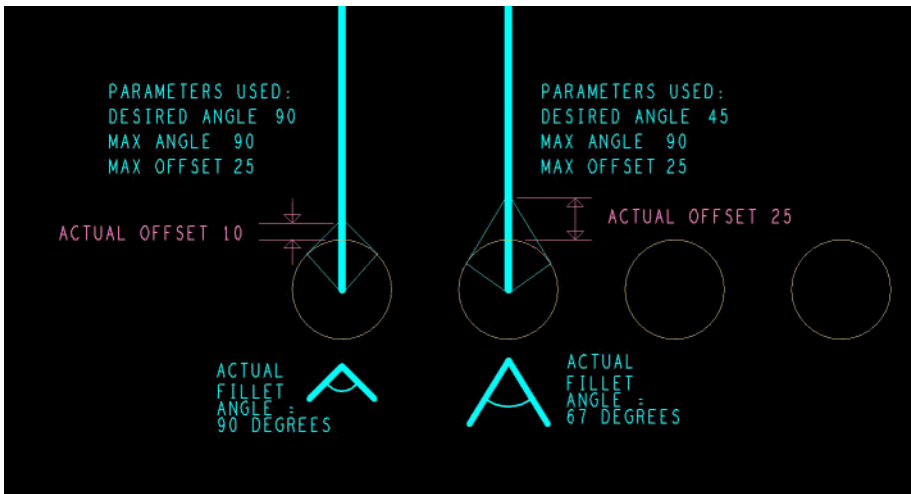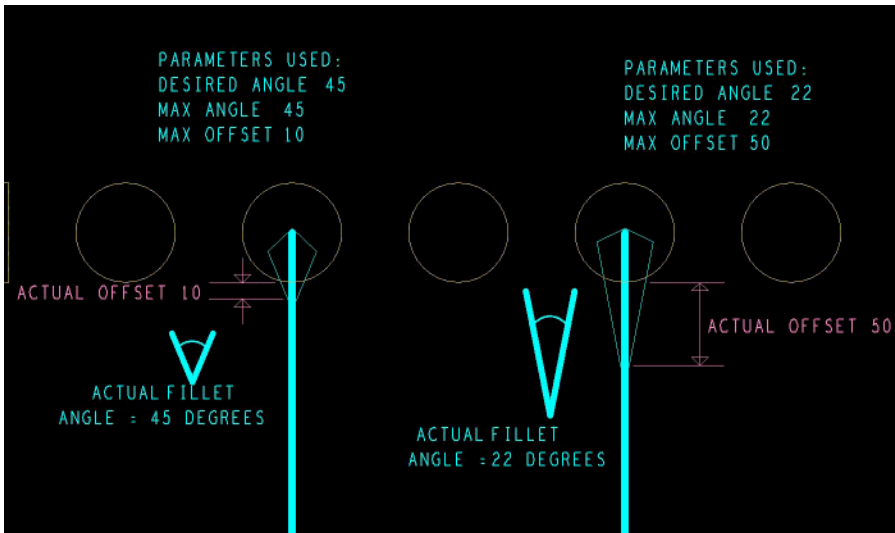|	
Fillet Objects	Specifies the object for fillet.
Desired angle	Specifies the angle created by the generated fillet shapes. A larger *Desired Angle* and a smaller *Max Offset* create a short fillet. A smaller *Desired Angle* and larger *Max Offset* create a long fillet. Not applied for creating arc fillets.
Max angle	Specifies the maximum angle for the fillet.The maximum possible value is *99*. This value must always be equal to or greater than the *Desired Angle*. A larger *Desired Angle* and a smaller *Max Offset* create a short fillet. A smaller *Desired Angle* and larger *Max Offset* create a long fillet. Not applied for creating arc fillets.
Max offset	Specifies the maximum distance between the intersection of the pad edge and the connecting line, forming the fillet length. Not applied for creating arc fillets.
Max arc offset	Specifies the maximum distance between the pad edge and the point along the curved trace, forming the fillet length. The default is *5 mil* (expressed in drawing units).
Min arc offset	Specifies the minimum distance between the pad edge and the point along the curved trace, forming the fillet length.The default is *1 mil* (expressed in drawing units). Must always be lesser than the *Max arc offset*.

⚠ You can fillet arc segments for only pins and vias. The fillets generated for arc segments always have curved lines.

Min line width	Specifies the minimum line width of the cline entering the pad.If the line width of the cline is less than the value specified here, the fillet is not created.
Max line width	Specifies the maximum line width of the cline entering the pad. If the line width of the cline is greater than the value specified here, the fillet is not created.

Examples

Tapered Trace Options

Tapered traces	Set to enable tapering of traces. Not on by default.
Min segment angle	Specifies the minimum segment angle. The default is 135 degrees. The smaller included angle should be greater than or equal to 90 degrees.
Desired Angle	Specifies the angle of taper created by the fillet. A larger Desired Angle and a smaller Max Offset create a short fillet. A smaller Desired Angle and larger Max Offset create a long fillet.
Max Offset	Specifies the maximum distance the fillet may extend from vertex to the point of cline width transition.

Related Topics

- gloss param

- Glossing Controller Dialog Box

- Line and Via Cleanup Dialog Box

- Via Eliminate Dialog Box

- Line Smoothing Dialog Box

- Center Lines Between Pads Dialog Box

- Improve Line Entry Into Pads Dialog Box

- Line Fattening Dialog Box

- Choosing Glossing Applications

- Editing a Line Fattening Step

- Creating a Fattening Step from the Template

- Creating a Line Fattening Step for a Line Width not Listed in the Fattening Steps Box

- Deleting an Item in the Fattening Steps Box

- Adding Fillets (Static or Dynamic Mode)

- Deleting Fillets Interactively (Static Mode)

- Adding Tapered Traces (Static or Dynamic Mode)

- Deleting Tapered Traces Interactively (Static Mode)

- Creating a Templates Parameter Set for Dielectric Generation

- Creating a New Parameter Set for Dielectric Generation

- Eliminating Unused Blind and Buried Stack Vias

Generating Fillets and Tapered Traces

For creating fillets and tapered traces with additional etch/conductor use the following steps:

1. Choose *Route – Teardrop/Tapered Trace – Parameters*.
   Alternatively, type `gloss param fillet` in the Command window.
   The *Fillet and Tapered Trace* dialog box appears.

2. Set the parameters and choose to generate fillets and tapers in static or dynamic mode.

3. Click *OK*.

Examples
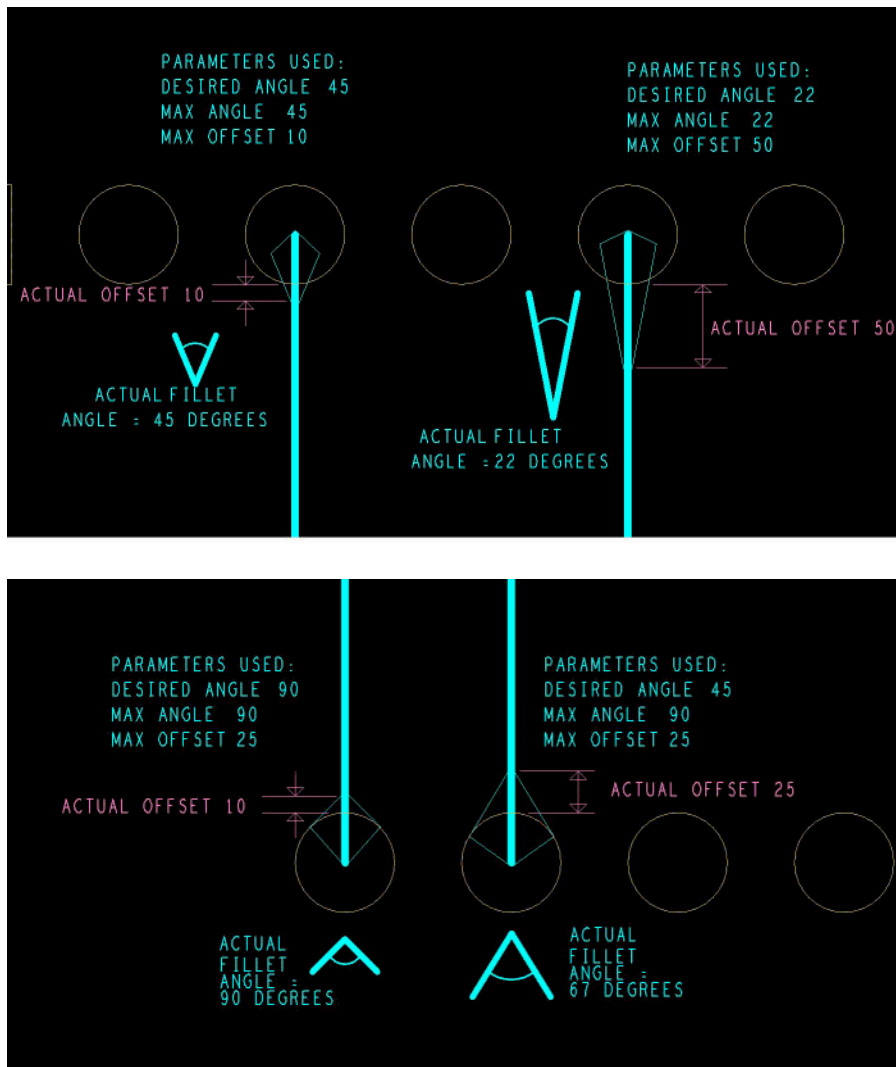
Related Topics

- gloss param fillet

- gloss param

graphic edit

The `graphic edit` command allows you to manually resize RF shapes.

Access Using

- Menu path: *RF Module – Graphic Edit*

Related Topics

- groupedit

Editing RF Shapes Manually

To modify RF shapes do the following:

1. Choose RF Module – Graphic Edit.
   Alternatively, type `graphic edit` in the Command window.

2. Select the RF component whose shape you want to modify.

3. Drag one of the handles and resize as required.

4. Double-click to complete resizing.

5. Right-click and select *Done.*

Related Topics

- groupedit

grid toggle

The `grid toggle` command turns On or Off the grid display in your user interface.

Access Using

- Toolbar icon:

group

The `group` command enables any edit command to operate on multiple design elements that you specify by selecting the elements. The behavior of this command is identical to selecting *Group* from the right button pop-up.

Related Topics

- groupedit
- group add

Editing Multiple Design Elements

To create a group of design objects in edit modem, perform the following steps:

1. When the `edit` command is active, type `group` at the console window prompt.

2. Specify the group by sequentially choosing each element to be included in the group.

3. When you have completed the group, enter `complete` at the console window prompt (or click right and choose *Complete* from the pop-up menu).
   Each element highlights as you choose it and is operated on by the active edit command.

Related Topics

- group

- groupedit

- group add

group add

The `group add` command adds physical elements to an existing group or creates a new group in the pre-selection use model, in which you choose an element first, then right-click and execute the command.

Available only in the placement and general edit application modes, the *Add to Group* command appears on the right-mouse-button pop-up menu when you pre-select the following group-supported physical elements:

- Groups
- Components
- Symbols
- Pins
- Vias
- Clines
- Lines
- Shapes
- Text

⚠ Because nets are logical rather than physical elements, the *Add to Group* command does not appear on the right-mouse-button pop-up menu when you pre-select nets.

Not all preselected elements become part of a group. Elements promote to their top-level database element in the hierarchy, and based on the following rules, become group members:

- A selected group is incorporated into a new group
- Parent elements of a selected child: For example, selecting a group of symbols also selects their pins and other elements, but only the selected symbol becomes a group member.
- Symbols of selected components; components are excluded as they are not physical elements
- Vias, Clines, Lines, Shapes, and Text not belonging to a symbol

Related Topics

- Adding Objects to a Group
- group
- groupedit

Add to Group Dialog Box

Enter new group name	Specify the name of the group to create.
Or pick group to add elements to	Choose to combine the selected elements with those already in an existing group.
Overwrite existing group	Choose to replace elements in the existing group with the selected elements.
OK	Creates the group and closes the dialog box.
Cancel	Closes the dialog box without creating a group.

Related Topics

- group add

- group

- groupedit

Adding Objects to a Group

You can add more objects to an existing group using the following steps:

1. Choose elements to include in the group.

2. Hover your cursor over one of the elements.

3. Right-click and choose *Add to group* from the pop-up menu that appears.
   The *Add to Group* dialog box appears.

4. Enter the name of the new group or choose an existing group to which to add the elements. To replace elements in the existing group with the selected elements, click *Overwrite existing group*.

5. Click *OK* to create the new group.

To disband a group in the pre-selection use model, you can pre-select a group, right-click and choose *Disband Group*.

Related Topics

- group add

- group

- groupedit

groupedit

The `groupedit` command lets you choose arbitrary database objects and collect them in a named database. The named database is referred to as a permanent group because it is saved with the database and can be referenced as a single object. This command functions in the menu-driven editing mode, in which you choose a command (verb), then the design element (noun).

Permanent groups let you reuse portions of your designs by extracting them to new or existing designs—in effect, using them as building blocks upon which more complex objects can be constructed.

Related Topics

- Allegro User Guide: Placing the Elements
- Add to Group
- Creating a Group For Design Reuse
- Editing a Group
- Disbanding a Group

Groupedit Command: Options Panel

Access Using

- Menu path: *Edit – Groups*

Group Type	Specify the type of the group. • Generic Group • RKO Group
Group Name	Specify the name of the group to create.
Disband	Dissolves the selected group.

Related Topics

- Placing the Elements
- Add to Group
- Editing a Group
- Disbanding a Group

Creating a Group For Design Reuse

Groups are building blocks upon which complex objects can be constructed. To create groups that can be reused in a design do the following:

1. Choose *Edit – Groups.*
   Alternatively, type `groupedit` in the Command window.

2. Select the group type.

3. In the *Options* panel, enter the name you want to give the group and press Enter or Return.

4. Click *Yes* when a confirmation message appears.
   The group name is added to the existing groups listed in the *Options* panel.

5. Choose the database elements you want to add to the group.
   All database elements added to the group are highlighted.

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

Related Topics

- Allegro User Guide: Placing the Elements

- groupedit

- Disbanding a Group

Editing a Group

A group can be modified by replacing its objects with other elements. To modify a group members do the following:

1. Choose *Edit – Groups*.
   Alternatively, type `groupedit` in the Command window.

2. Select the group type.

3. Choose the group you want to edit by choosing it in the *Options* panel group list or by clicking on a group member in your drawing.
   All objects in the group become highlighted.

4. Using the Find filter to define the objects you are going to choose, modify the group by choosing objects to be new members or deselecting objects to remove by using the `Cntrl` key.

5. To edit other groups, choose *Next* from the pop-up menu.

6. When editing is completed, choose *Done* from the pop-up menu.

Related Topics

- Allegro User Guide: Placing the Elements

- groupedit

- Groupedit Command: Options Panel

- Creating a Group For Design Reuse

Disbanding a Group

A group of design objects can be split into individual design objects using the disband command.

> ⚠ Properties you attach to objects at the group level remain attached to the individual objects after you disband the group. Delete properties at the group-level before disbanding.

1. Choose *Edit – Groups*.
   Alternatively, type `groupedit` in the Command window.

2. In the *Options* panel, enter the name of the group you want to disband. –or– Click on the name in the group list box.

3. Click *Disband*.

4. Click *Yes* when a confirmation message appears.

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

Related Topics

- Allegro User Guide: Placing the Elements

- Add to Group

- groupedit

- Groupedit Command: Options Panel

- Creating a Group For Design Reuse

guideport

The `guideport` command creates visual checkpoints that suggest potential connections for unrouted nets that cross partition boundaries. The lead designer uses the command after creating design `partitions`, but prior to exporting them. (Guideports are unavailable during partition creation.) You can fine-tune, move, and reconfigure guideport locations suggested by the design tool based on the *Spacing Criteria* parameters in the *Options* panel.

Only connections with one pin inside the partition and a target connection outside the partition receive a guideport, excluding pass-through connections; consequently, a guideport functions much like a Rat T in that it visually breaks a ratsnest line where it crosses the partition boundary, assisting the partition designer to run the trace.

Spacing and line width constraints locate guideports around the partition boundary. Guideports appear for every from-to based on the default grid in the same color as ratsnest lines. Multiple guideports may exist on a single ratsnest line if it passes through a partition or enters multiple partitions where the edges are not coincident. Guideports contain no layer information.

When guideports exist on a net or on one associated with the chosen object, and you choose *Display – Element* (`show element` command) in a design partition file, previously unscheduled nets appear in the text display dialog box as guideport-scheduled nets, as shown below. Nets you wholly or partly scheduled before creating guideports appear as user-scheduled, guideport scheduled nets, and net schedule appears as locked.

> ⓘ This command is available only with Allegro X PCB Editor and PCB Team Design product option.

```
LISTING: 1 element(s)


<NET>


Net Name:      VCLKA
* user scheduled net*
```

```
* guideport scheduled net * schedule is locked*
U5: 34 U18.11 U8.11 T.1 T.2 U21.11


Via Count:        2
Total Etch Length:         3180.5 MIL
Total Path Length:         3781.5 MIL
Total Manhattan Length:      3793 MIL
Percent Manhattan:        99.70%


Pin      Type  SigNoise Model
---      ----  --------------
U5.34     OUT  CDSDefaultOutput
U18.11     IN  CDSDefaultOutput
U8.11      IN  CDSDefaultOutput
VCLKA.T.1
VCLKA.T.2
U21.11     IN  CDSDefaultOutput


3 unrouted connection(s) remaining
VCLKA.T.2 to U21.11
VCLKA.T.1 to VCLKA.T.2
U8.11 to VCLKA.T.1


Properties attached to net
NET_PHYSICAL_TYPE = SYNC
NET_SPACING_TYPE = SYNC


Electrical Constraints assigned to net
pin order type: all rats are user defined
Object is read only
```

Related Topics

- Allegro User Guide: Placing the Elements

- Using Guideports in a Partition Design

- partition

- show element

Guideport Command: Options Panel

Access Using

- Menu path: *Place – Design Partition – Guideports*

Guideport Commands	
Create	Accepts the guideports suggested by PCB Editor. The *Total Guideports* field changes to reflect the number of suggested guideports you accepted, and the guideports are instantiated in the design.
Replace All	Deletes any guideports that you modified and reverts to their original, system-generated placement.
Delete All	Removes guideports.
Spacing Criteria	These parameters function with the *Move* and *Collapse/Spread* fields in the *Select Action* section.

Default Grid	Click to use the X and Y grid values defined for the TOP etch/conductor layer, which default from the *Define Grids* dialog box, available by choosing *Setup – Design Parameters* (`prmed` command), clicking the *Display* tab from the *Design Parameter Editor*, and then clicking *Setup Grids*. You can also access the *Define Grids* dialog box by running the `define grids` command.
Min Line/Line by Net	Click to use net constraints (a combined value of spacing and width) for minimum line-to-line spacing, which may differ by net if you chose by window or by temp group.
User Defined	Specifies spacing criteria in the unit of measure that defaults from the *Design* tab of the Design Parameter Editor, available by choosing *Setup – Design Parameters* (`prmed` command).
Ignore Spacing Rules	Overrides specified design constraints; doing so allows guideports to stack one upon another.
Select Action	
Move	Repositions chosen guideports based on the specified *Spacing Criteria*.
Delete	Removes one guideport or those you chose by window or by temp group.
Collapse/Spread	Narrows or expands the spacing between guideports based on the specified *Spacing Criteria*, particularly useful for multiple guideports chosen by window or by temp group.
Auto-Select Diff Pair Mate	Automatically chooses the mate of a differential pair.
Suggested Guideports	Displays the number of guideports that the layout editor recommends based on the number of crossing connections in the design.
Total Guideports	Displays the actual number of guideports currently in the design.

Related Topics

- define grid

- prmed

Using Guideports in a Partition Design

Guideports suggest potential connections for unrouted nets that cross partition boundaries. You can create guideports after partitioning a design using the following steps:

1. Create design partitions using *Place – Design Partition – Create Partitions*.

2. Prior to exporting partitions, ensure that the ratsnest that will cross partition boundaries are displayed.

3. Choose *Place – Design Partition – Guideports*.
   Alternatively, type `guideport` in the Command window.

4. Review the number of automatically generated guideports that appear in the *Suggested Guideports* field on the *Options* panel.

5. Click *Create* to accept the suggested guideports. The *Total Guideports* field changes to reflect the number of suggested guideports you accepted, and the guideports are instantiated in the design.

6. Modify the *Spacing Criteria* in the *Options* panel as necessary.

> ⚠ These parameters function with the *Move* and *Collapse/Spread* fields in the *Select Action* section.

Related Topics

- guideport
- partition