cadence®

Allegro® X Layout Editor Design Parameters Reference

Product Version 23.1
September 2023

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Contents

2

Introduction

This document lists PCB Editor drawing parameters that controls the display of the design. This user guide describes display controls for the following layout editors:

- Allegro PCB Designer
- Allegro Venture PCB Designer
- Allegro Enterprise PCB Designer (Symphony Team Design Option included in bundle)
- Allegro Physical Viewer Plus
- Allegro Package Designer+
- SiP Layout XL

Note: The information provided in this document is based on Cadence® Allegro® release 17.4-2019.

3

Display Parameters

Display

Connect point size	Specifies the size of a connect point in user units. The default is 10. Connect point sizes should be set hight enough to be easily seen in a drawing. The appropriate setting (for visibility) depends on the line width in the design.
DRC marker size	Determines the size, in user-defined units, of the DRC markers that appear in a design. The default is 25.
Rat T (Virtual pin) size	Allows the specification of the graphical size of Rat T.
Max rband count	Specifies an upper limit on the number of lines drawn during editing sessions. This value should not have to be changed with modern graphic cards. The default is 500.
Ratsnest geometry	Displays a pop-up menu that allows the specification of shape of the ratsnest lines. The default is Jogged.
Ratsnest points	Displays a pop-up menu that allows the specification of the closest distance between existing etch or pads (Closest endpoint) or between two pins (Pin to pin). The default is Closest endpoint. **Note:** This option does not change the scheduling of the net. It only changes the way the endpoints of ratsnest are shown in the display for ratsnest that is defined by net scheduling.

Display net names (openGL only)

Clines	Displays net names on cline segments.
Shapes	Displays net names on dynamic shapes.
Pins	Displays net names on pins.

Enhanced display modes

Plated holes	Displays plated drill holes in the design. Unchecked by default, because plating hole visibility often is not required due to the volume of entities and supporting geometries such as etch layer pads.
Backdrill holes	Displays backdrill holes in the design.This option should be used only after the backdrill process is done.
Non-plated holes	Displays non-plated drill holes in the design. Non-plated hole visibility may be preferred during placement and routing because they have no pads associated with them on etch layers and are normally invisible.As a result, they may lack proper keepout areas for guidance. By default, this option is not selected.
Padless holes	Displays padless holes in a design due to visible pads all being NULL or suppressed as unused. When enabled padless holes will be displayed even if plated or non-plated hole display is not enabled. This setting can also be controlled from *Setup – Cross-section – Unused Pads Suppression* user interface.
Connect points	Enables visibility of connect points.
Filled pads	Controls onscreen design display and fills pads. Unchecked by default. Leaving pads unfilled can improve display performance.
Connect line endcaps	Controls onscreen design display and rounds line vertices to more closely approximate artwork.
Thermal pads	Displays thermal and anti pads rather than regular pads when there is a negative plane. Unchecked by default.
Bus rats	Displays the middle part of the ratsnest lines with the same BUS_NAME property so that they appear to be merged into a thick line. Unchecked by default.
Waived DRCs	Displays waived DRCs. Disabled by default.

Drill labels	Displays via label on top of each blind/buried via, indicating the from-to subclasses spanned by the via. Subclass numbers are displayed, which are shorter than subclass name. Labels fit within the visible pad shape. A colon character indicates a single via, for example, 3:4. A dash character indicates a stack of connected vias spanning the entire subclass range, for example, 2-6. Labels are not displayed for thru-hole and single-layer pad-only vias. Displays backdrill labels for the backdrill holes with format <Start-Layer>-<Must-Cut-Layer>-<Must-Not-Cut-Layer>, for example, B1-3-4, B27-16-15, and B2-2-3.
Design origin	Displays a crosshair at the drawing origin.
Diffpair driver pins	Displays a figure on top of each visible differential pair driver pin. the pin use code of OUT is required for the figure to be displayed. Figures will not be displayed if the product does not support differential pairs.
Use secondary STEP models in 3D Canvas	Displays secondary STEP models in the 3D Canvas.

Grids

Grids on	Displays the grid. Checked by default.
Setup grids	Opens Define grid dialog box.

4

Design Parameters

Size

User units	Identifies the units of measure for the active design. Options are Mils, Inch, Microns, Millimeter, or Centimeter. Note that changing user units during the design process can introduce errors into the database. The editor automatically sets the database accuracy based on the unit of measurement selected. Database accuracy can be reset for additional decimal places, but not less than the default. Warning: Gerber (Artwork) export is not supported with microns and accuracy of 3 decimals or higher.
Size	Specifies the size of an active design through a pop-up menu. Options are A, B, C, D, and Other. The available drawing sizes if the User units are Mils or Inches are: A, B, C, D, or Other. The available drawing sizes if the User units are Millimeter or Centimeter are: A1, A2, A3, A4 or Other. If one of the available drawing sizes is chosen, the editor automatically displays the extents for the size chosen in the drawing extent section.
Accuracy	Determines the number of decimal places for calculations in the active design. The choice of the number of decimal places is determined by the User Units fields. Note that changing the user units during the design process can introduce errors into the database. Warning: Gerber (Artwork) export is not supported with microns and accuracy 0f 3 decimals or higher.
Long name size	Defines the maximum number of characters for names for the following strings: net name, padstack name, slot name, and function pin name. This value can only be increased. The minimum value is 32 and the maximum is 255. The default value is 255.

Pad flash mode	Shapes
Extents ■ Left X ■ Lower Y ■ Width ■ Height	Determined the location and the size of your drawing. If you specified `Other` in the size field, specify the X and Y coordinate extents of the drawing size in the Drawing Extents section. The Left X and Lower Y Fields represent the X,Y coordinates of the lower left corner of the drawing. The Width and Height fields represent the available drawing work space.
Shrink Extents to Design Contents	Reduces design extent limited to a size so that all geometries of the design fit.
Move Origin ■ X ■ Y	Changes the location of the drawing according to the values you enter in the X and Y boxes. The field is then reset to 0 after the origin is moved.
Symbol options	This field is enabled only in Symbol Editor.
Type	Package, Mechanical, Format, Shape, Flash
Line lock	
Lock direction	Displays a pop-up menu that specifies the direction of lines added to a drawing. The options are Off, 45 and 90. The default is Off.
Lock mode	Specifies the type of segments to use when adding lines or connect lines. The options are Line and Arc. The default is Line.
Minimum radius	Determines the minimum radius allowed for an arc. The default is 0.
Fixed 45 Length	Specifies the length, in user units, of 45-degree segments. By default, this is unchecked. If checked, the default is 25.
Fixed radius	Specifies the radius, in user units, of arcs. By default, this is unchecked. If checked, the default is 25.
Tangent	Specifies whether tangent lines are locked. If checked, arcs construct tangent to lines. By default, this is checked.
Symbol	
Mirror	Mirrors symbols added to a drawing. By default, this is not checked.

Angle	Specifies the default angle that is used to place symbols and text. An angle with up to three decimal places can be entered, or an available angle can be chosen from the pop-up menu. The default is 0.
Default symbol height	Determines the height for symbols that do not have associated PACKAGE_HEIGHT_MIN/<MAX property values.

5

Text Parameters

Size

Justification	Indicates the anchor point within text added to the layout. The anchor location determines how the text appears in the text block. The default is Left.
Parameter block	Defines the size and spacing of the text added to the design. Using the `define text` command, up to 64 text blocks can be defined. The default is 1.
Parameter name	
Text marker size	Determines the size, in user units, of the displayed text markers. These markers indicate locations for text that will be entered later. The default is 50.
Setup text sizes	Opens Text Setup dialog box.

6

Shapes Parameters

Dynamic Shape	Dynamic shapes allow real-time shape updating during interactive etch editing. Shape parameters are structured into a global, shape instance, and object level hierarchy
Edit global dynamic shape parameter	Opens Global Dynamic Shape Parameters dialog box
Edit layer dynamic shape parameter	Opens Dynamic Shape Layers Parameter dialog box
Static Shapes	Static shapes can be used for critical handcrafted etch that should not be modified automatically
Edit static shape parameter	Opens Static Shape Parameters dialog box
Split plane	A split plane is an embedded plane with two or more copper areas associated with different nets. Split plane parameters control the fill style of shapes on the plane
Edit split plane parameters	Opens Split Plane Params dialog box

7

Flow Planning Parameters

General

Flow line relative width — Specifies a display scale for flow line width. The value must be in the range of 1 to 300 inclusive and represents a percentage of the width of the physical bundle as determined by the number of rat members including the line width and spacing requirements of their nets. The display scale applied to all bundles in the design and does not affect the display of random logic.

Flow via relative size — Specifies a display scale for flow via size (diameter). The value must be in the range of 1 to 300 inclusive and represents a percentage of the size determined by twice the widest unscaled flow line width. The display scale applies to all bundles in the design and does not affect the display of random logic.

Router plan

- *Auto plan visibility*
- *Random logic plan visibility*

Specifies the route planner's ability to make visibility changes to bundles and random logic during a planning phase. Off indicates that you wish to control router plan visibility manually. On indicates that route plan visibility is automatically turned on when the plan is modified by the route engine.

Copy flow when split bundle — Controls whether the "Split Bundle" command will automatically invoke copy flow. Otherwise, you must manually choose "copy Flow" from the pop-up menu if you want to copy the flow from the source bundle.

Default Bundle Properties

Bundle naming

Bundle name prefix	Specifies a prefix for naming bundles. The string entered is type checked using the same character set allowed by Allegro for net names.
Prompt for name	Specifies whether manual bundle creation commands (bundle create, bundle split, etc) prompt you for a bundle name. When enabled (checked) these commands prompt for a bundle name. This control does not apply to automatic bundling.
Display controls	
Bundle	Sets the initial visibility of a new bundle at the time of its creation. Existing bundles in the design are not affected. This setting also affects the visibility of flow line, flow vias, and rake lines associated with a new bundle.
Expanded rakes	Controls whether new bundles are created with expanded rakes enabled. Existing bundles in the design are not affected.
Plan	Sets the initial visibility of the route plan lines associated with a bundle at the time of its creation. Existing route plan lines are not affected by this control except for those associated with rats added to a new bundle. The visibility of those plan lines change to match this setting.
Ratsnest	Sets the initial visibility of the ratsnest lines associated with a bundle at the time of its creation. Existing ratsnest lines are not affected except for rats added to a new bundle. Their visibility changes to match this setting.

Auto Bundle

Minimum rat count	Specifies the minimum number of rats required before a bundle can be created.
Pin proximity	Specifies the maximum center-to-center distance between pins before a rat can become a member of a bundle.
Members must go to the same component at	■ One end: Specifies that rats must have at least one end on the same component before they can join a bundle. ■ Both ends: Specifies that rats must have both ends share a component before they can join a bundle.
Members must be in the same Bus	Specifies that rats must be on nets that have the same Bus property on them before they can join a bundle.

Plan Invocation

Termination control

	Physical and Spacing errors	*Unroutes*	Specifies that the planner stops at the end of the current pass and terminate the route run when a rat fails to complete a connection.

Routing Controls

Etch shapes	Controls whether GRE pays attention to etch shapes when routing. When set to “Router thru”, GRE will route over etch shapes as if they are not there. When set to “Route around”, GRE will route around etch shapes.
Elongation control	
Allow in constraint areas	Enables (Yes), or disables (No) the ability of the planner to locate tuning patterns within constraint areas.

Tuning pattern

Choose to select Accordion or Trombone tuning patterns.

Accordion

Gap	Specifies the gap between adjacent wraps of regular elongation patterns. Previously used values that are entered are retained in the pull-down menu.
Min amplitude	Specifies the minimum amplitude value allowed for any wrap of a regular elongation pattern created by the planner. Any wrap with an amplitude below this value is eliminated from the pattern.
Max amplitude	Specifies the maximum amplitude value allowed for any wrap of a regular elongation pattern created by the planner. Any wrap with an amplitude above this value is eliminated from the pattern.
Corner type	Specifies a corner type for bends in the elongation pattern created by the planner.
Miter size	Specifies a minimum corner length base in the elongation pattern created by the planner.

Trombone

Gap	Specifies the gap between adjacent wraps of regular elongation patterns. Previously used values that are entered are retained in the pull-down menu.
Min amplitude	Specifies the minimum amplitude value allowed for any wrap of a regular elongation pattern created by the planner. Any wrap with an amplitude below this value is eliminated from the pattern.
Max levels	Specifies the maximum amplitude value allowed for any wrap of a regular elongation pattern created by the planner. Any wrap with an amplitude above this value is eliminated from the pattern.
Corner type	Specifies a corner type for bends in the elongation pattern created by the planner.

	Miter size	Specifies a minimum corner length base in the elongation pattern created by the planner.
Bundle controls		
Space within bundle	Default	Specifies that the route engine is not constrained with regard to packing or unpacking.
	Min DRC	Specifies that the route engine be given an additional spacing goal to meet. A Min value must be entered. Min is the minimum amount of additional space the router attempts to meet if it can. It must be less than the Max value (when Max is specified). The value can be any real number including 0. Max is the maximum amount of additional space the router attempts to meet if it can. If it meets this value, it stops trying to spread out the affected bundles. The value can be any real number including 0 and it must be equal to or greater than the Min value.
Max transitions		
	Unlimited	Specifies that the number of via transitions allowed per bundle member is unlimited. The transition count does not include pin escapes. The value does not apply to fanout vias.
	Limit to	Specifies a maximum number of via transitions that are allowed per bundle member. The value does not apply to fanout vias.

Layer matching

	Off	Specifies that bundle members be allowed to route on different layers
	Route on the same layer	Specifies that bundle members must route together on the same layer. If a layer transition does occur, then all members must transition together. This choice is disabled if there is no single layer on which all bundle members can be routed.
Layer Controls	Choose the direction of random logic on layers. Options are: Horizontal, Vertical, and No preference	

8

Route Parameters

Add Connect

Layer mode — Specifies the use-model for adding vias with the Add Via (or double-click) button. Set to Alternate Subclass for the original-style use-model, where the user selects an alternate subclass and via padstacks in the Options. Set to Working Layers, which removes the alternate subclass and via padstack Options, but through the use of an add-via popup-gui, the user can choose a padstack or choose the via-stop layer, where the Via List determines (by priority ordering) which via or via-stack is added, with stacking/staggering rules determined by the constraints.

Line lock — Line lock allows default settings for routing as straight line segments, or arc segments, with either 45, 90, or 0 degrees for angle control. Arc linelock disables bubble mode, and an angle of off implies that any-angle routing is allowed.

Route offset — Route Offset provides an enable button and angle value for routing at angles that are offset from multiples of 45 degrees (0, 45, 90, 135...). When enabled, routes are snapped to an angle that is +- the offset angle from the 45 degree incremental angle. This feature is only applicable when Line Lock is Line/45.

Miter — This field sets the corner size/radius. It accepts both integers and values relative to the current line width in the format <n>x.

Line width — Defines the width of the line in user units.

Style

Bubble	Off: The wire path entered is unchanged. All clearance violations are flagged with error markers. Hug only: New clines hug all objects spaced at minimum design rule. Hug Preferred: New cline moves itself around etch objects where possible to avoid spacing DRCs; if not possible, then shoving of etch objects is attempted. Shove Preferred: New clines moves other etch objects where possible to avoid spacing DRCs; if not possible, then hugging of other etch objects is attempted.
Shove vias	Off: Vias are not shoved. Minimal: Vias are shoved in hug-preferred manner. Vias are not moved unless there is no way to draw a cline around them. Full: Vias are shoved in a shove-preferred manner. Any new or edited etch always shoves vias out of the way.
Gridless	Specifies that the etch/conductor can go off the routing grid. This allows connections to be added at maximum density while accommodating varying design rules and line widths. The DRC minimum space separate objects. When Bubble is disabled, this field controls the removal of a small segment at the end of the new route.
Clip dangling lines	Application for shove-preferred mode. When enabled, new etch routes through dangling clines, removing the unconnected section.
Smooth	Off disables smoothing. Minimal executes dynamic smoothing to minimize unnecessary segments. Full executes more extensive smoothing to remove any unnecessary jogs. Super extensively removes any unnecessary vertices from all shoved clines. Smooth option is applies when Bubble is enabled.

Snap to connect point — *Specifies whether the connection snaps to the connect point if it is close to a target element.*

Replace etch — Changes the path of an existing trace without extra delete and add steps. When a new loop is added into an existing trace, forces the older portion of the loop to be automatically deleted.

Auto-blank other rats — Enables automatic blanking of rats that are on nets not being routed. Those rats are blanked when you start routing a trace and are restored when you are done routing the trace.

Optimize in channel — Optimize by centering segments in a defined channel. Opens Channel options dialog box.

Clearance view — Enables polygons generations over obstacles on the active subclass. The obstacle size is determined by the spacing constraint between the trace and the obstacle for “Spacing” mode and the same spacing constraint plus half the width of the trace being routed for “Channel” mode. For “Channel” mode when adjacent polygons do not touch, routing in the channel is possible without creating a DRC.

Delay Tune

Style — *Geometry used to add delay to clines. Only one of Accordion, Trombone, and Sawtooth can be selected.*

- *Centered* — Selecting centered option will center elongation around the chosen cline segment. The mouse position defines one half of the amplitude. If elongation is not centered, all elongation is on one side of the specified cline segment.
- *Gap* — Values can be entered as integers to represent the gap in user units or <n>x space or <n>x width, where n is an integer that represents line width or line-to-line spacing respectively.
- *Corners* — Values allowed are 90, 45 and FullArc. FullArc creates a single semi-circle connecting the ends of the parallel cline segments of the elongation etch.
- *Miter size* — Values can be entered as integers to represent corner segment length.

Allow DRCs — Selecting this field will ignored design rules yet still report DRC errors when creating the elongation rectangle. In this mode, elongation is created regardless of any objects that cause a DRC. If this field is unselected, no physical DRCs are created and elongation rectangles which would result in a DRC are not allowed.

Edit Vertex

Style

Bubble	Enabling a hug or shove-preferred mode disallows arc linelock mode selection since bubble mode does not support arcs. Off flags all clearance violations with error markers. Hug Only will, where possible, force the routed cline to contour other etch objects to avoid spacing DRC errors and does not change other etch. Hug Preferred behaves as Hug Only does except will attempt to shove etch objects other than the routed cline to open routing paths if they cannot be successfully contoured. Shove Preferred will force the routed cline to push and shove other etch objects to avoid spacing DRC errors. If not possible, it attempts to hug other etch objects. This is true for vias if Shove via capability is enabled.
Shove vias	Off: Vias are not shoved. Minimal: Vias are shoved in a hug preferred manner. Any new or edited etch always shoves vias out of the way. Full: Vias are shoved in a Shove Preferred manner. Vias are not moved unless there is no way to draw a connect line around them.
Clip dangling connect lines	Application for shove-preferred mode. When enabled, new etch routes through dangling clines, removing the unconnected section.
Smooth	Off: No smoothing errors. Minimal: Executes dynamic smoothing to minimize unnecessary segments. Full: Executes more extensive smoothing to remove any unnecessary jogs. Full could, in some cases, hamper the ability to successfully edit a vertex.
Allow DRCs	Specifies that design rules can be violated in order to make a connection. If Bubble is disabled, the vertex is set at a point (between the last good point and the current point) that does not cause a DRC.

Allow gridless	Specifies whether the connect line or via has to adhere to the routing grid. When this option is enabled, the tool can slide connections at maximum density while accommodating varying design rules and line widths. In addition, bubbled vertices are snapped to grid. This affects connections only if Allow DRCs is disabled.
Snap to 45	If enabled, the vertex location is snapped to create 45 degree routing when the cursor is near a point which would provide such routing. Snapping also will happen to line up segments which are almost colinear.

Slide

Min corner size	Min Corner Size determines the size that Slide will use when sliding segments into corners. Applies to existing mitered segments only, and not to segments that meet at approximate 90-degree angles. Does not apply to jogs or end segments. Note that the Min Corner Size is used as a length in both X and Y directions; so the length of the mitered segment is actually square-root-of-2 times the Min Corner Size.
Min arc radius	Min Arc Radius determines the smallest allowable radius when sliding arcs or sliding segments that are connected to arcs.
Vertex action	Vertex Action has 3 choices (Line Corner, Arc Corner, Move), which determines the action that will be taken when the user picks at a single vertex. Line Corner: the vertex gets converted to a straight mitered segment (with its size honoring the Min Corner Size option). Arc Corner: the vertex gets converted to an arc (honoring Min Arc Radius option). Move: No conversion, but just slides the vertex dynamically. Note that some vertices may not qualify for Line Corner or Arc Corner conversions, if Min Corner sizes cannot be honored, or if picking at an arc.
New Seg Angle	Specifies the angle to be used when a line changes direction or moves around an obstacle.

Style

Bubble	Enabling a hug or shove-preferred mode disallows arc linelock mode selection since bubble mode does not support arcs. Off flags all clearance violations with error markers. Hug Only will, where possible, force the routed cline to contour other etch objects to avoid spacing DRC errors and does not change other etch. Hug Preferred behaves as Hug Only does except will attempt to shove etch objects other than the routed cline to open routing paths if they cannot be successfully contoured. Shove Preferred will force the routed cline to push and shove other etch objects to avoid spacing DRC errors. If not possible, it attempts to hug other etch objects. This is true for vias if Shove via capability is enabled.
Shove vias	Off: Vias are not shoved. Minimal: Vias are shoved in a hug preferred manner. Any new or edited etch always shoves vias out of the way. Full: Vias are shoved in a Shove Preferred manner. Vias are not moved unless there is no way to draw a connect line around them.
Clip dangling connect lines	Application for shove-preferred mode. When enabled, new etch routes through dangling clines, removing the unconnected section.
Smooth	Off: Choose to disable smoothing. Minimal: Executes dynamic smoothing to minimize unnecessary segments. Full: Executes more extensive smoothing to remove any unnecessary jogs. An additional segment on each end of the changed segments can be included. Super (Apd/Cdnsip products only): Extensively remove any unnecessary vertices from all shoved clines.

Enhanced Arc Support	Enhanced Arc Support will enable bubble to generate arcs when encountering arc or circular objects. Any etch changed as a result of the slide operation will generate arc corners.

Allow DRCs	Specifies that design rules may be violated to make the etch edit. The violations are flagged with DRC markers which must be resolved for a successful design. If allow DRCs is disabled and DRCs already exist on the trace or in a group of traces chosen for routing, or if the tool determines that DRCs are introduced to the design, the tool does not slide the connection.
Gridless	Specifies whether the connect line or via to slide has to adhere to the routing grid. When gridless routing is enabled, the tool can slide connections at maximum density while accommodating varying design rules and line widths. This affects connections only if bubble is active or if Allow DRCs is disabled.
Auto join	Auto Join determines the behavior when sliding past a parallel segment. With Auto Join Off, construction is done to connect the parallel segments. With Auto Join On, the slide segment algorithm will look past the parallel segment to intersect with segments farther down the cline. Minimum corner controls apply wherever Auto Join intersects down the cline. Holding down the control-key will temporarily cause Slide to use the inverse value of this option.
Extend selection	Extend Selection option allows the selection to include the segments and/or vias adjacent to the selection while maintaining the direction of movement that you would get with the original selection. Reduces use of window-pick and improves control by limiting direction of movement for single segment selection. Can be useful, for example, when sliding tuning bump corners, to slide 3 segments together with one pick. Holding down the shift-key will temporarily cause Slide to use the inverse value of this option. Has an associated drop-down menu field for setting the mode for extending the selection.
Auto-I. Phase Tune	Displays Auto-I Phase Tune Parameters
Auto-I. Delay Tune	Displays Auto-I Delay Tune Parameters
Timing Vision	

Display control

	Style	Solid or Striped (candycane). Controls how clines of the Nets selected for the Timing Vision are displayed. This will override any current Custom Color until the Nets are removed from the Timing Vision.
	Colors	Select the color to use for clines that are Shorter, Longer or Within (Satisfying) Timing Constraints. Additional choices can be made for those that are a small amount shorter or longer
	Pattern for critical signals	Controls the pattern to display pinpairs that are the target of a relative propagation delay group. This has no effect if the Striped style is in use; or if Smart Calculation is selected
Timing Mode	Controls type of data that should be used by Timing Vision to determine proper color coding	
Minimum for Smart Goal calc (% of total length)	Influences Smart Goals for pin-pin portions of a longer Pin Pair. If the direct pin-pin length as a percentage of the overall Pin Pair length is less than the specified amount, then the calculated goal for that pin-pin portion will not be longer than its current length. Use zero (0) to disable this logic	
Match Group Selection Mode	Allows selection of all member nets of a match group to the Timing group	
XNet Selection Mode	Allows selection of XNets to the Timing group	

Auto-I. Convert Corner

Convert type	Accepts `Arc, 45` an `90`. If `Arc` is chosen, the *Preferred Radius Size* and *Min Radius Size* fields are enabled. If the `45` is chosen, the *Preferred Corner Size* and *Min Corner Size* fields are enable. If the `90` is chosen, all those fields are enable.
Allow in cns areas	There are two choices, `YES` or *NO*, for Allow in cns areas field. The default value is `YES` which means the corner conversion is allowed in cns areas. The value of `NO` indicates the corner conversion is disallowed in the cns areas.
Preferred radius size	When Corner type Arc is selected, Preferred Radius Size field is applicable, which specifies the Preferred or Maximum radius size.
Min radius size	When Corner type Arc is selected, Min Radius Size field is applicable, which specifies the Minimum radius size.

Preferred corner size	When Corner type 45 is selected, *Preferred corner size* field is applicable, which specifies the Preferred or Maximum corner size.
Min corner size	When Corner type 45 is selected, *Min corner size* field is applicable, which specifies the Minimum corner size.
Allow DRCs	If Allow DRCs field is checked which indicates the design rule may be violated when we do the corner conversion.
Auto-I. Breakout	Displays Auto-I Breakout Parameters
Auto-I Trunk Route	Displays Auto-I Trunk Route Parameters
Auto-I. Auto Connect	Displays Auto-I Auto Connect Parameters
Gloss	
Set Gloss parameters	Opens Glossing controller dialog box
Create Fanout	
Create Fanout Parameters	Opens Create Fanout Parameters form

9

Manufacturing Applications Parameters

Testprep

Edit testprep parameters	Opens Testprep Parameters dialog box

Thieving

Edit thieving parameters	Opens Thieving Parameters dialog box

Silkscreen

Edit silkscreen parameters	Opens Auto Silkscreen dialog box

Drafting

Edit drafting parameters	Opens Dimensioning Parameters dialog box