

Best Practices: Working with Shapes

**Product Version 23.1
October 2023**

© 2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

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

Allegro PCB Editor contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission. All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR52.227-7013 et seq. or its successor.

Contents

1

<u>Dynamic Positive Shapes</u>	5
<u>Why Dynamic Positive Shapes?</u>	5
<u>Product Availability</u>	6
<u>Additional Information</u>	6
<u>Review of the Shapes Menu and Parameter Dialog Box</u>	7
<u>Polygon</u>	7
<u>Rectangular</u>	7
<u>Circular</u>	8
<u>Options tab</u>	9
<u>Step 1 - Choosing a Class/Subclass</u>	9
<u>Step 2 – Choosing Shape Fill</u>	10
<u>Step 3 – Assigning a Net Name</u>	11
<u>Step 4 – Entering a Shape Grid (Optional)</u>	12
<u>Step 5 – Choosing a Segment Type</u>	13
<u>Defer Performing Dynamic Fill</u>	13
<u>Select Command</u>	14
<u>Manual Void</u>	15
<u>Edit Boundary</u>	16
<u>Raise Priority</u>	16
<u>Delete Islands</u>	16
<u>Change Shape Type</u>	17
<u>Merge Shapes</u>	17
<u>Shape Check</u>	18
<u>Compose shapes</u>	18
<u>Decompose Shape</u>	19
<u>Shape Connectivity Report</u>	19
<u>Status Tab</u>	20
<u>Global Dynamic Params ...</u>	21
<u>Best Practices and Performance Guidelines</u>	23
<u>Getting Started with Shapes</u>	23

Working with Shapes

<u>Negative Planes – Dynamic or Static Fill?</u>	25
<u>Global Parameter Considerations for Dynamic Shapes</u>	26
<u>Shape Instance Parameters</u>	32
<u>Object Level Properties</u>	33
<u>Interactive Etch Editing</u>	34
<u>Interface to Allegro PCB Router</u>	37
<u>Outputting Manufacturing Data</u>	37
<u>User Preference Variables</u>	38
<u>Questions and Answers</u>	40

Dynamic Positive Shapes

Shape parameters are structured into a global, shape instance, and object level hierarchy. Shape fill can be dynamic, allowing users to easily modify circuitry without regenerating, or static, which follows the traditional model for shapes. During interactive editing, you can modify component placement or add connect lines and vias while the shape updates in real time, which is referred to as “dynamic” throughout this document. Improved graphics that involve a transparent stencil pattern allow objects from multiple layers to be visible through the shape. Allegro PCB Router alignment includes the passing back of vias through dynamic shapes but not clines. Provisions for producing quality artwork are also included in the parameter settings.

Why Dynamic Positive Shapes?

If designing a printed circuit board (PCB) did not involve changes, we could retain static implementations of creating copper areas and add them near the end of the design process. This of course is not the case in today’s concurrent engineering driven world, where change is more of the normal course of events throughout the entire design process. Currently in Allegro PCB Editor, changes to areas occupied by copper shapes require the user to either remove and to re-add the shape or add changes directly within the shape area while creating a vast amount of DRC conditions. On a multilayer PCB, a simple via hole added though the board could impact several shapes, each requiring some level of intervention. The migration to dynamic-based shapes offers a real-time editing environment where changes made within the shape trigger an immediate shape updating routine otherwise known as auto-voiding. Changes that happen dynamically include:

- Adding/Deleting/Moving a component
- Adding/Slide a cline
- Adding/Sliding a via
- Updating shape parameters, that is, additional clearance
- Modifying the shape’s boundary
- Adding a manual voided area

Working with Shapes

Dynamic Positive Shapes

In terms of performance, for example, how long would it take to add then make edits within a ground shape on an outer layer of a dense 15x15 inch pcb? The answer is somewhat proportional to the performance of autovoicing. This document expands upon the options to consider when using dynamic shapes on large complex PCBs. Controls to disable shape fill in favor of a batch process or disabling the internal smoothing operation are options to maintain productivity.

Product Availability

All Tiers of Allegro PCB Editor, APD, Allegro PCB SI.

Additional Information

For more information regarding positive shapes, refer to the technical documentation provided on Cadence Online Support.

<http://www.support.cadence.com>

Review of the Shapes Menu and Parameter Dialog Box

The shape menu offers an intuitive use model that allows users to easily create and edit shapes in the same session. The former method requiring users to enter a secondary menu application has been eliminated. From the main toolbar, choose *Shape* to bring up the menu of commands. A shape toolbar appears on the main toolbar. Parameter settings can be entered in the *Global Dynamic Shape Parameters* dialog box and overridden at the Shape instance or Object level. Bitmap images inside the dialog box help users better understand the implied function.



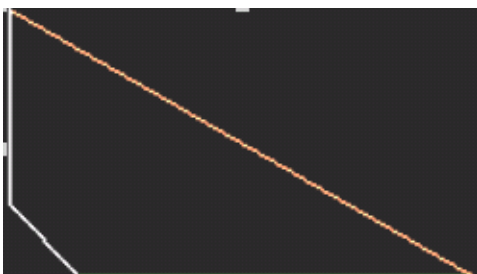
Tip

If the Shape Toolbar does not appear as part of the main toolbar display, in Allegro PCB Editor, choose *View – Customize Toolbar*, then enable the *Shape* option in the dialog box, followed by OK.

Polygon



Use *Shape – Polygon* to draw multi-side shapes that may be used for a placebound, route keepout, or a board outline. Like many other Allegro PCB Editor commands, the *Options* tab is used in conjunction with the main command. When drawing a polygon, choices for segment type and angle are available. When entering a polygon, an extra dynamic line is added from the last end point to the starting point of the polygon, maintaining a closed polygon image at all times. Left mouse double click or clicking *Done* from the popup completes the shape boundary. *Oops* backs up to the last pick. The dynamic line used for maintaining a closed polygon adheres to the current *Segment Type* set in the *Options* tab and appears in orange.



Rectangular



Shape – Rectangular adds a rectangle to the subclass layer chosen in the Options tab.

Working with Shapes

Dynamic Positive Shapes

Circular



Shape – Circular adds a circle to the subclass layer chosen in the Options tab.



Tip

Dynamic filled shapes are only allowed on ETCH subclasses in Allegro PCB Editor.

Options tab

When adding a dynamic shape, physical options pertinent to the shape are controlled in the Options tab, including choice of a subclass layer, net name, type of shape fill, shape grid, and line style. Voiding-related options are controlled in the *Global Dynamic Shape Parameters* dialog box.

Dynamic shapes can only be added to ETCH layers. When you add a dynamic etch shape that crosses the route keepin, or modify the route keepin boundary, the dynamic shape boundary is clipped to the route keepin by default. When you edit or move a dynamic shape that crosses a keepin, the tool does not clip the shape at the keepin by default.

To preserve any user-defined dynamic shape boundary and re-clip it to the route keepin during a dynamic shape update, enable the `shape_rki_autoclip` board-level environment variable in the *User Preferences* dialog box, available by choosing *Setup – User Preferences* (enved command).

For example, mechanical engineering changes a board outline and route keepin, thereby generating a new Intermediate Data Format (IDF) file. Upon reading the IDF file into Allegro X PCB Editor, any dynamic shapes currently clipped by the route keepin automatically update to the new route keepin location specified by the IDF file.

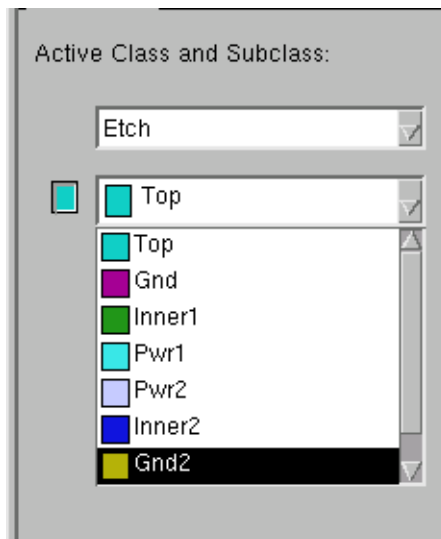
If you add a dynamic shape that is completely outside the route keepin, the tool ignores the route keepin when voiding.

After choosing a command to add a shape, follow these steps to add a shape on an electrical subclass.

Step 1 - Choosing a Class/Subclass

Color swatches appear in the subclass section in the *Options* tab that align with the etch color on that particular subclass layer. The subclass layer can be chosen prior to the first instantiated pick or at any time during shape creation.

Figure 1-1 Color Swatches



Step 2 – Choosing Shape Fill

Five shape fill types are available: cavity, dynamic copper, static solid, static crosshatch, and unfilled.

- **Cavity:** A cavity is the space around the embedded component in the dielectric between two etch layers. If the component is placed between two internal layers, the cavity associated with the component is a closed cavity. However, if the component is placed on an internal layer but protrude out from the external layers, the associated cavity can be an open cavity. By default, the cavity shape is derived from the Placebound shape. However, if required, cavity shape can be added to the symbol definition.
- **Dynamic Copper:** The preferred option for most conductive shapes, dynamic copper triggers an auto-voiding process upon each edit to the boundary or elements within. The editing of elements can include moving a component, adding or sliding clines and vias, or changing parameters. Global parameter file controls type of dynamic fill (*Smooth*, *Rough*, or *Disabled* mode). Dynamic fill applies only to etch subclass layers (electrical layers), including both positive and negative planes.
- **Static Solid:** The traditional method of adding copper shapes in Allegro PCB Editor. Shape remains a constant until manually voided by element. Static solid shapes are typically used for copper paths carrying high current, copper heat sinking, or with keepout areas.
- **Static Crosshatch:** Typically used in advanced packaging or RF applications for power distribution.

Working with Shapes

Dynamic Positive Shapes

- Unfilled: Use for board outlines, package geometry, rooms. Cannot be used on etch subclass layers.

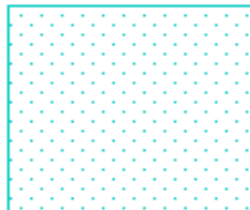


Tip

Static and dynamic shapes have unique graphic patterns. Although drawn in the same color, the stencil pattern associated with dynamic shapes is drawn more densely than that for static.



Dynamic Pattern



Static Pattern

Step 3 – Assigning a Net Name

Nets can be assigned to shapes by choosing the desired net from the net browser that lists all nets in the design or from the pulldown list of nets with the voltage property assigned to them, or by directly picking a pin or cline after right mouse clicking and choosing *Assign Net*.

Since most nets assigned to shapes are power/ground in nature, a dropdown list provides a shorter list that is easier to navigate. To change a shape's net name, choose the shape, change the net name in the *Options* tab, or right mouse click and choose *Assign Net*.

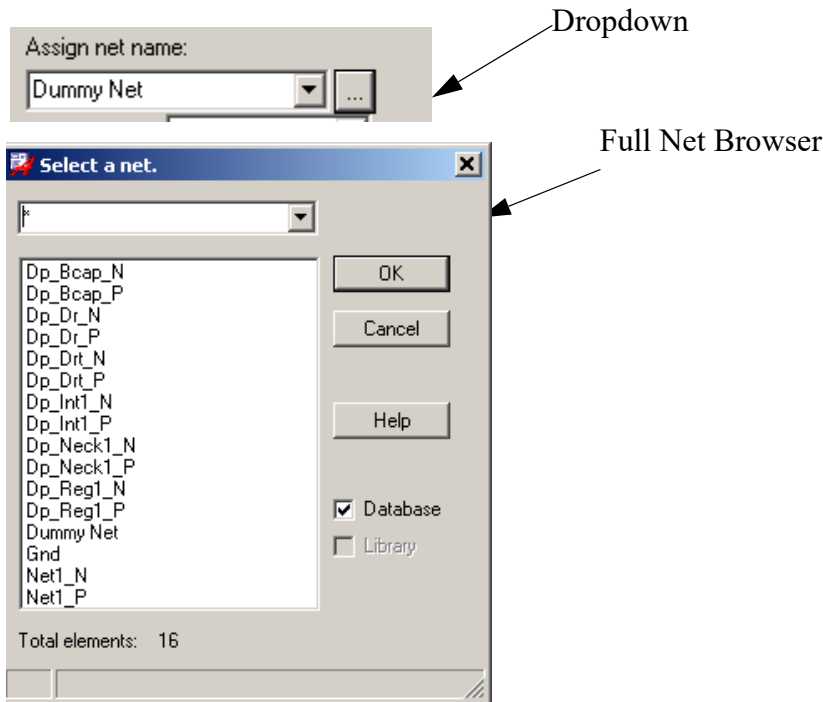
Working with Shapes

Dynamic Positive Shapes



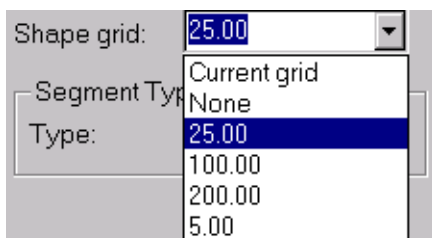
Tip

Voltage properties can be assigned to nets in all tiers of Allegro PCB Editor. The voltage property is also required to enable the plane rat option introduced in Version 14.2. Assign voltage properties by choosing *Logic – Assign DC Nets*, choose a net, and enter a voltage level.



Step 4 – Entering a Shape Grid (Optional)

Since shape grids tend to be more coarse than routing grids, a separate shape grid on the *Options* tab saves time toggling between the *Setup – Grids* menu. If a shape grid is not entered, it uses the current subclass grid. Up to five grid entries can be entered during any Allegro PCB Editor session. Exiting Allegro PCB Editor clears the grid settings from memory. Once the shape editing session is completed, the working grid reverts back to the original database settings.



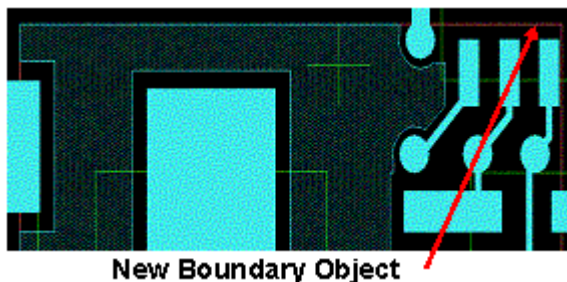
Step 5 – Choosing a Segment Type

When using *Shape – Polygon*, choose among four options for line segments.

- Line: freehand or any angle line draws
- Line 45: corners are drawn at 45 degrees
- Line Orthogonal: corners are drawn at 90 degrees
- Arc in conjunction with radius value: standard 3 pt arc, radius displays in the status window as arc stretches. Also available from a right mouse click.

When drawing the shape, an enclosed polygon is continually maintained. A dynamic line connecting the last end point to the starting point is added. Double clicking or right mouse clicking and choosing *Done* completes the boundary and fills the shape to its respective parameter settings.

A Boundary color class is available by choosing *Display – Color/Visibility* and *Stack- Up*. The boundary appears in this color when adding shapes, but the shape fill color overrides it when the shape is completely drawn. For example, a shape with a fill color of blue and boundary of red appears as solid blue if the fill overlays the boundary. If a voided area overlays part of the boundary, the latter appears as the boundary subclass color (red), as in the graphic below.



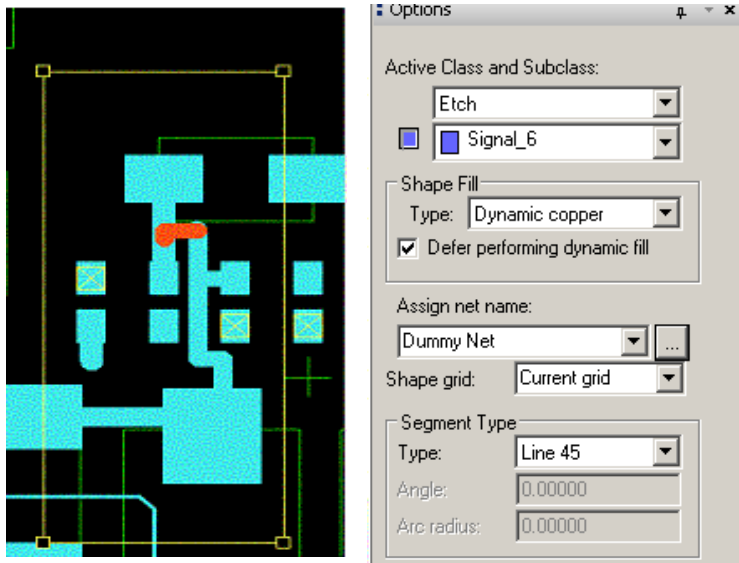
Defer Performing Dynamic Fill

Enabling *Defer Performing Dynamic Fill* retains shape boundaries; however, the dynamic fill process is deferred. After the initial shape boundary is created, often the shape boundary is refined to meet the final design intent. It may be advantageous to enable this switch while using *Shape – Select Shape or Void/Cavity*, whether you are dragging the shape boundary or adding new vertex points. The recommended use model for this feature applies to a single active shape and not for a global application. The chosen shape becomes temporarily unfilled until the *Defer Performing Dynamic Fill* option is disabled or you right

Working with Shapes

Dynamic Positive Shapes

mouse click and choose *Done* from the popup menu. Cadence recommends using this option when wholesale editing a shape's boundary outline.



Defer Performing Dynamic Fill/Unfilled Shape

Select Command



Use the *Shape – Select Shape or Void/Cavity* to choose a shape or manual void for boundary editing or in conjunction with right mouse click options to customize shape-instance parameters, copy to other layers, or to produce an instance-level report. After choosing any part of the shape or boundary edge, the shape turns the setting for temporary highlight color. Handles appear at all vertex locations. New vertex points can be added by choosing any part of the boundary and dragging it to its final destination. Boundary edges can be moved in whole by passing the cursor over it until the cursor changes its form then dragging accordingly with the left mouse button pressed. The entire shape can also be moved by dragging the cursor with the left mouse button pressed.

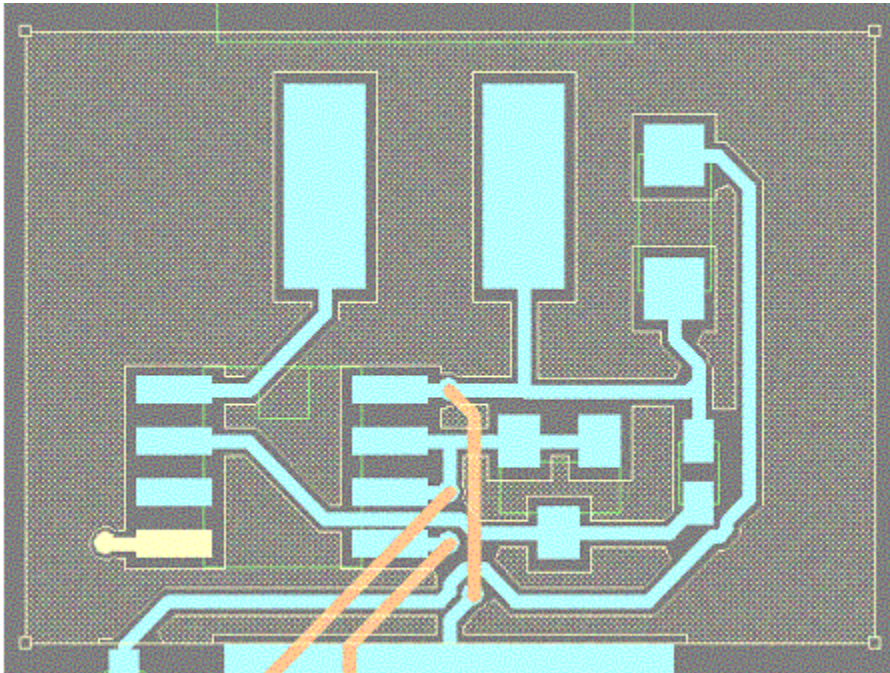
Working with Shapes

Dynamic Positive Shapes



Tip

If you are unsuccessful in dragging the entire shape with the left mouse button, enable the `shape_drag_move` variable available by choosing *Setup – User Preferences* and the *Shape* category. This variable is disabled by default.



Handles appear on 'active' shape vertex locations

Manual Void



Use the geometrical based commands *Shape – Manual Void/Cavity – Polygon*, *Shape – Manual Void/Cavity – Rectangular*, and *Shape – Manual Void/Cavity – Circular* to add custom voids within a dynamic shape. Modify void boundaries as you do shape boundaries. Choosing just inside the void boundary places the object in a chooseable state with handles appearing at all vertex locations. Overlapping voids are not permitted.



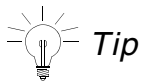
Tip

Only manual voids can be modified or deleted. Auto generated voids are not database objects and cannot be edited.

Edit Boundary



Improvements in the edit boundary function now allow the design window to provide a better gravitational mechanism to snap to the boundary object. Use *Shape – Edit boundary* to modify the shape's boundary edge. After choosing the command, choose any part of the shape, pick the origin of where the boundary is to be edited. Continue to add to the boundary and end the command by choosing the final destination, any location on the existing boundary. Overlapping during the editing is not allowed. The shape redraws and updates its voiding upon completing of the command.



Tip

When using *Shape – Edit boundary* on complex shapes, Cadence recommends working in *Disabled* dynamic copper fill mode as *Defer Performing Dynamic Fill* in the *Options* tab is unavailable when in this command.

Raise Priority

When dynamic shape boundaries overlap, use *Shape – Select Shape or Void/Cavity*, then right mouse click and choose *Raise Priority* from the pop-up menu that appears to control which of the two shapes controls the voiding path around its respective boundary. When you choose *Raise Priority*, Allegro PCB Editor prompts you to choose a shape, which then essentially becomes the master and is brought to the front.

Delete Islands



Auto-voiding a shape often creates fragments or unconnected areas, called islands in Allegro PCB Editor. *Shape – Delete Islands* highlights and deletes chosen or all islands in any one shape or subclass layer. After choosing *Shape – Delete Islands*, use the *Options* tab to delete or navigate to each island. Alternatively with the left mouse, choose an island or choose by window an area to delete a group of islands. For enhanced viewing, choose *Display – Color/Visibility, Display*, and enable *Shadow Mode* and the *Dim Active Layer* field to accentuate the highlighted islands. When islands are deleted, a manual void remains to prevent this area from re-filling with copper.



Tip

The *Shape – Delete Islands* command can only be used when *Global Dynamic Fill* is set to *Smooth*. If used on *Rough* filled shapes, this message appears in the pop-up window.

```
Dynamic Shapes present are not Smooth. Should I update the shapes to Smooth  
(YES) or exit command (NO) ?
```

Delete islands at the end of the design cycle since permanent manual voids are added to the shapes. Use *Shape – Manual Void/Cavity – Delete* to remove them if more design changes occur.

Change Shape Type

Shape – Change Shape Type can be used to convert a dynamic shape into a static shape if the shape is critical, and if it should not be automatically modified or voided. Converting from static to dynamic allows older database to migrate into a dynamic state without re-adding each existing shape. Cadence recommends editing the edges so that the changed shapes contain the original outline which was drawn before any voiding occurred. Although a conversion to dynamic eliminates the existing voids, it can not clean up edges that have clearances surrounding other etch elements. For best results, edit the boundary to remove auto-voided clearance around etch elements that overlapped the original border before converting the shapes. When converting a dynamic shape to static, Allegro PCB Editor advise that loss of boundary, parameters and manual voids occur as a result of the change to static. For this reason, setting desired dynamic voiding parameters should be done prior to the conversions. See *Shape – Global Dynamic Params...* in Allegro PCB Editor.



Tip

If a dynamic shape needs to be locked, that is, prevent future dynamics such as adding vias through it, changing it to static is recommended. Fixing a shape only fixes its boundary outline, preventing it from being moved as an entity.

Merge Shapes

To merge or combine two or more overlapping shapes, use the *Shape – Merge Shapes* command. When using this command, consider it in terms of primary-secondary where the primary shape is chosen first, followed by the secondary shape(s), which inherit the parameters of the primary shape when completing the command.



Tip

Shapes must have a common net name to merge.

Shape Check

Shape – Check executes legacy code that works on a single shape at a time. It's most useful on a shape that fails to run through Gerber 4x or 6x, but not as relevant for raster artwork. In theory, a vector-based dynamic shape should process without problems, but a mix of user-added and dynamic-generated voids might cause issues, along with user-defined void patterns that are too close to each other or have edges that the *Minimum Aperture* field (defined in the *Global Dynamic Shape Parameters* dialog box) could not outline.

Compose shapes

When you choose *Shape – Compose Shape* (compose_shape command), you can build complex shapes using arcs and lines. You compose shapes on a DRAWING FORMAT. Cadence recommends that you compose such shapes on a user-defined subclass; for example, a CONSTRUCTION subclass, using the following procedure:

1. Choose *Setup – Subclasses* (define_subclass command) to display the Define Subclass dialog box.
2. Click on DRAWING FORMAT to display the Define Non-Conductor (or Define Non-Etch) Subclass dialog box.
3. Enter "USER-DEFINED" in the New Subclass field and press Enter.
4. Click *OK* in the Define Subclass dialog box.

Note: While it is recommended that you create shapes on a layer that you create and define, the program creates a shape when data from any CLASS/SUBCLASS is chosen for compose shape.

To set up shape parameters, choose *Setup – Design Parameters* (prmed command) to access the Design Parameter Editor or right mouse button click whenever you are working in an application mode, then click the *Shapes* tab to edit global dynamic shape parameters, static shape parameters and split plane parameters.

Using a drawing tool on the subclass layer, you create the basic shape that you want. Then you pick the lines and arc that constitute the shape. Choose *Shape – Compose Shape* (compose_shape command) to connect the ends of the lines (or trims crossing lines) to

create a single solid shape. You can also create voids in the solid shape where they are needed using the same procedure you use to create the solid shape.

Decompose Shape

To decompose a previously composed shape, choose the shape and choose *Shape – Decompose Shape* decompose_shape command. Line and arc segments remain trimmed, chamfered, or rounded, but each segment is detached from each other. Then you can modify the shape and reconnect the segments with the *Shape – Compose Shape* (compose_shape command).

Shape Connectivity Report

A summary of analysis results and shape-based parameters for all shapes is available by choosing *Tools – Reports and Dynamic Shapes*. For shape- instance-based information, choose a shape, right mouse click, and choose *Report* from the popup menu. The following example displays the generation results of one shape.

Generation Results

Number of Etch Shapes	2	
Number of Islands(non-conducting)	0	
Number of etch shapes in conducting Area(1)	1	
Number of etch Shapes in conducting Area(2)	1	
Number of Manual Voids	0	
Number of pins/vias not connected	1	
Number of pins/vias partially connected	1	
Total Area of Etch Shape(s)	3300961.8668	square MILS
Total Area of Boundary Shape	4431985.0000	square MILS
Etch area/Boundary area(%)	74.48	

Pins/vias Not Connected

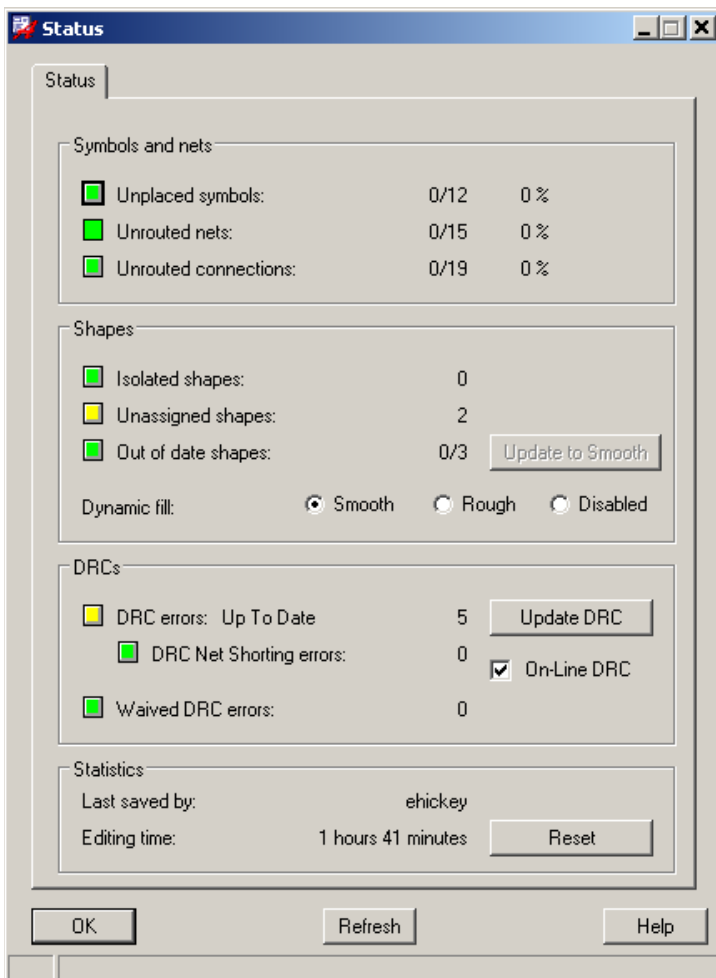
Symbol Pin "C6321.1"	10975.000	1500.000
----------------------	-----------	----------

Pins/vias partially Connected

Symbol Pin "U45.4"	9100.000	1950.000
--------------------	----------	----------

Status Tab

The Status tab quickly summarizes board condition in terms of connectivity, placement, and shape status.



For shape status:

- Out of date shapes: indicates the number shapes whose status is other than *Smooth*. All shapes must be updated to *Smooth* prior to running artwork or ODB++.
- Unassigned Shapes: indicates the number of shapes without logic assigned to them. Click the yellow box to locate them.
- Isolated Shapes: indicates the number of isolated shapes or islands in the design. Clicking the adjacent yellow box displays a dialog box indicating the extents of the shapes. The following is an example of the Shape Island Report.

(-----)

Working with Shapes

Dynamic Positive Shapes

```
(
)
(      Shape Island Report      )
(
)
(      Drawing      : demo1.brd      )
(      Software Version : 15.1A4      )
(      Date/Time      : Fri Oct 24 10:42:15 2007 )
(
)
(-----)
Total islands on design: 5

Layer = TOP
Extents: (6640.000 5830.000) (6865.000 6190.000) Net: V1_8_RAC
Extents: (6637.000 5827.030) (6745.000 6095.000) Net: P1_8V

Layer = INNER1
Extents: (7725.000 11925.000) (9800.000 12275.000) Net: GND

Layer = BOTTOM
Extents: (160.000 585.000) (515.000 1065.000) Net: BUZZER_B
Extents: (-15.000 740.000) (155.000 980.000) Net: FET_BUZZ
```

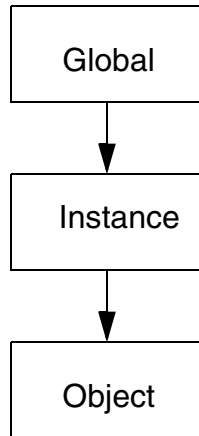
Global Dynamic Params ...

The *Global Dynamic Shape Parameters* dialog box controls parameters for all dynamic shapes. Updates to this dialog box automatically update each dynamic shape when you click *Apply* or *OK*. Parameters can be applied at three different levels:

- Global
 - ☐ Use the *Global Dynamic Shape Parameters* dialog box
- Shape Instance
 - ☐ Choose a shape, right mouse click, and choose *Parameters*
- Object Level (Pin, Via, Cline)
 - ☐ Choose *Edit – Property*: 7 dynamic properties available, all begin with DYN*

The implied hierarchy has object level properties overriding Shape Instance and Global. Shape Instance overrides Global. Overrides in the Shape Instance Parameter dialog box appear in bold blue, similar to overrides made in Constraint Manager.

Shape Parameter Hierarchy



Dynamic Fill

- *Smooth*: Produces artwork quality fill for each dynamic shape. Shapes are expected to be DRC free.
- *Rough*: Shapes approximately represent Smooth. Internal smoothing is disabled, maximum of two thermal relief spokes are drawn. Cadence recommends this option for large complex shapes or until performance metrics have been established where performance can be reasonably predicted.
- *Disabled*: Defers shape updating for all subsequent edits to dynamic shapes. Use this option if *Rough* mode performance is not acceptable.
- Xhatch parameters: Apply to static crosshatch shapes only

Void Controls

- *Artwork Format*: Optimizes shape fill for vector or raster processing
- *Minimum aperture for gap width*: Used for raster applications
- *Minimum Aperture For Artwork Fill*: Used for vector applications
- *Suppress Shapes Less Than*: Eliminates unconnected shapes less than this value when autovoicing a shape
- *Create Pin Voids*: Inline or individual options for voiding around pin fields
- *Distance Between Pins*: Used only when in-line chooseion is made

Working with Shapes

Dynamic Positive Shapes

- *Acute Angle Trim Control*: Used only when non-vector artwork format is chosen
- *Snap Voids To Hatch Grid*: Suppresses off grid clearance pads, leaving defined mesh pattern in a cross hatch shape

Clearance options

- *DRC Value*: Uses the DRC spacing values as clearance.
- *Thermal/Anti*: Clearance derived from thermal and antipad definitions in padstack. If antipad clearance is smaller than the DRC values, voiding increases the clearance to the DRC value.
- *Thru pin*: If antipad clearance is smaller than the DRC values, voiding increases the clearance to the DRC value.
- *Smd pin*: If antipad clearance is smaller than the DRC values, voiding increases the clearance to the DRC value.
- *Via*: If antipad clearance is smaller than the DRC values, voiding increases the clearance to the DRC value.
- *Oversize*: These values are incremented to default clearance settings.

Thermal relief connects

- Options for Pins (Thru and Smd) and Vias
- Min/Max Thermal connect values
- *Best Contact*: When enabled, if pin or via fails to meet minimum requirement, spokes populate in 22.5 degree increments.
- *Thermal Width Oversize Value*: Value is incremented to default thermal connect width lines
- *Fixed Thermal Width*: Overrides physical constraint set values

Best Practices and Performance Guidelines

Getting Started with Shapes

You are starting a new board and your stack-up construction calls for several negative planes and etch layers containing positive shapes. Outer layers are mostly pad only and are to be

Working with Shapes

Dynamic Positive Shapes

filled with a copper ground shield. If your board outline did not come in from IDF or DXF, choose *Shape – Polygon*, *Shape – Rectangular*, or *Shape – Circular* to draw your outline.

After setting the class/subclass to Board Geometry/Outline in the *Options* tab, your shape fill type automatically sets itself to *Unfill*. Do not attempt to change this, Allegro PCB Editor does not allow dynamic fill on non-electrical subclasses. Take advantage of the v shape grid option on the *Options* tab. Enter as many as five different shape grids to be used for outlines, areas, or copper shape boundaries. These settings are volatile and cleared when exiting Allegro PCB Editor. If adding component and etch keepout areas, follow the same procedure, but this time choosing a keepout subclass sets the shape fill type to *Static Solid*.

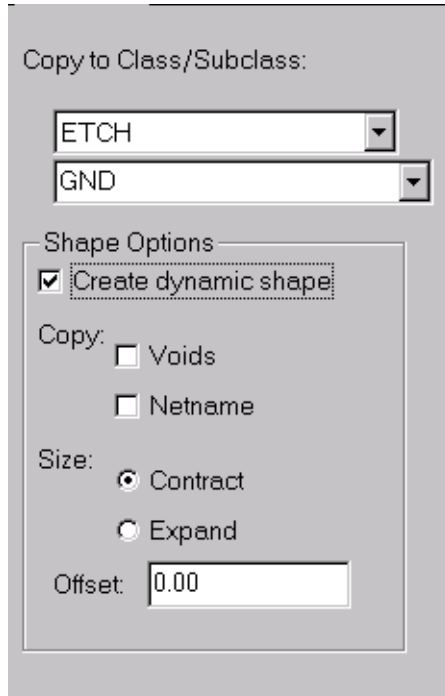
Set up negative planes using *Edit– Z-Copy* since the copper geometry of a negative plane tends to be an exact contour of the board outline minus a contraction distance. *Edit– Z-Copy* is available in all tiers of Allegro PCB Editor. *Edit– Z-Copy* can be used to copy a shape to multiple layers at once provided the target subclass names are named sequentially like GND, GND1, GND2. When choosing the subclass name to copy to, intercede and enter an asterisk after the name as shown in the graphic below. Once the shapes are copied, assign a net name by first choosing the shape using the arrow like icon in the shape toolbar or from the shape menu choose shape or void. After the shape is chosen, change the net name by picking from the pull down list in *Options* tab under *Assign Net Name*, provided your power and ground signals have a voltage property assigned or from the full net browser to the right of the pull-down. Only one shape can be chosen at a time.

If your board has several power and ground planes, use *Edit– Z-Copy* to copy only to one subclass layer. After a net name is assigned to the shape, use *Edit– Z-Copy* once again, but

Working with Shapes

Dynamic Positive Shapes

enable the *Copy Netname* option in the *Options* tab to preclude manually assigning a net name to many replicated shapes.



Use the *Create Dynamic Shape* option to create new dynamic shapes; otherwise, static shapes are created.

Negative Planes – Dynamic or Static Fill?

Use dynamic filled shapes on internal power and ground plane layers, especially on boards with split planes. In earlier releases (pre 15.0), the intersection between the split planes known as Anti-Etch created false DRC conditions when obstacles like vias or pins encroached upon them. In addition, when using *Route – Slide* to move a via across the Anti-Etch area, the via denoted as a DRC condition did not move across fluently, but jumped erratically, causing undesirable results. Negative dynamic shapes are useful because you eliminate the need to void pads close to edges, or fix chopped edges where pads were voided, but have moved. The dynamics of the negative shape only operate on the edges, which make split planes much friendlier.

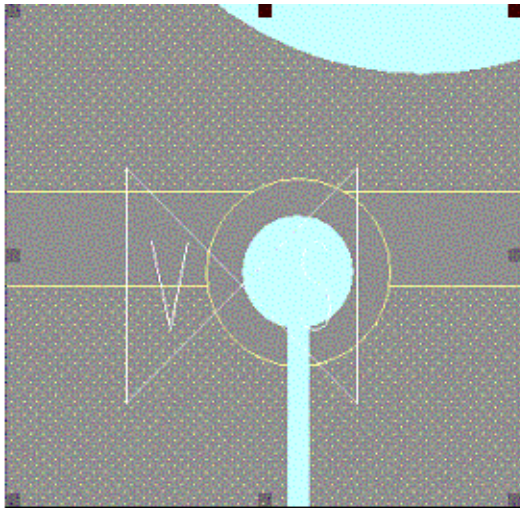
You do not have to change shapes to dynamic, you can leave them static.

If you have a pre-15.0 board with copper pour areas customized by manually adding void areas, then it is not advisable to change those shapes to dynamic unless you manually re-add the void(s) in the new dynamic shape. This is true for negative and positive because Allegro PCB Editor, with static shapes, cannot differentiate between voids created via the autovoid process and user created. Once a shape is dynamic, Allegro PCB Editor can track user-created voids versus autovoid-created voids.

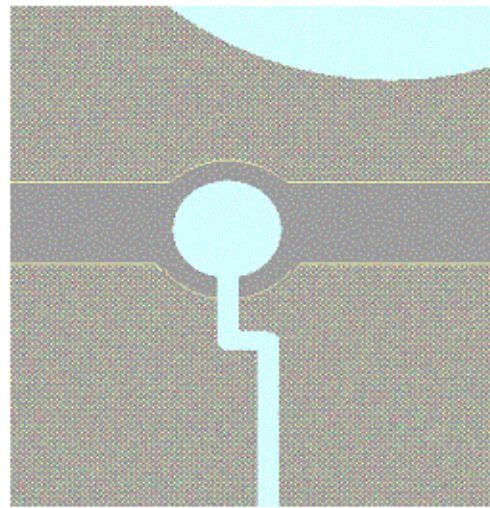
Working with Shapes

Dynamic Positive Shapes

The conversion from static to dynamic shapes was developed to aid those customers who wanted to convert pre-15.0 builds to use dynamic shapes. Additional manual work may be required in the conversion process to add user voids and to smooth the boundary. If the voids you have are based on route keepouts, dynamics generate the new voids for you, and they would then move if keepouts changed later.



14.2 False DRC condition



15.0 Dynamic Neg Planes

Global Parameter Considerations for Dynamic Shapes

Dynamic shapes are recommended for most electrical layer applications in Allegro PCB Editor. Before drawing shape boundaries, set parameters on the *Global Dynamic Shape Parameters* dialog box, which then automatically update to all dynamic shapes unless a shape has instance- or object-level properties.

When using dynamic shapes initially, Cadence strongly recommends setting *Dynamic Fill* to *Rough* mode. The major benefit of using this mode directly affects the performance of auto-voiding the shape. The *Rough* mode option produces approximate artwork quality shape results, but within a fraction of the time required using the *Smooth* mode. Internal to Allegro PCB Editor, the smoothing algorithm is disabled during auto-void and thermal spoke quantity is maximized at two. Eventually all shapes must be updated to *Smooth* to output artwork; however, you may want to do this in a batch job during non-work hours. If your shapes are confined to a limited area only and do not have an excessive amount of obstacles to void, it may be safe to choose *Smooth*; however, test the process using *Rough* mode first. Save a copy of your current database before updating shapes to *Smooth* if you decide to terminate the process.

Working with Shapes

Dynamic Positive Shapes

Important

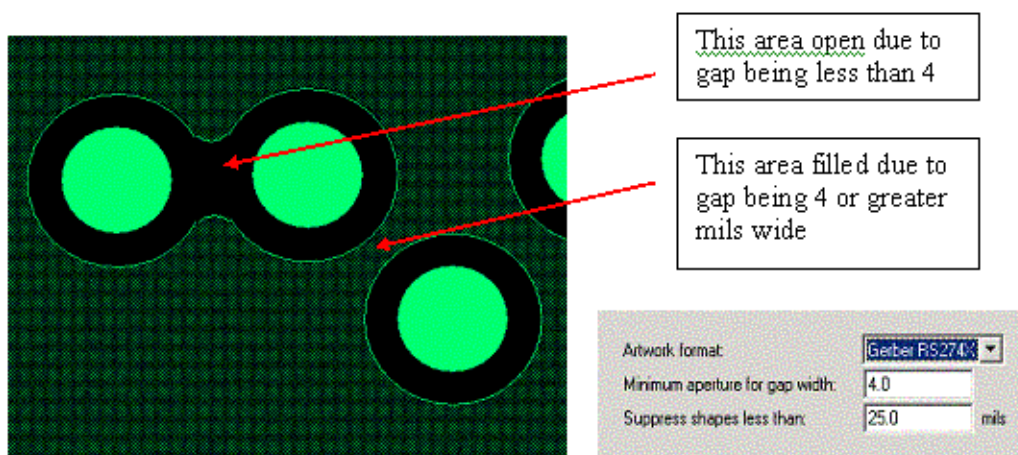
It's important to understand the impact of choosing global parameters. The following options can only be set at the global level and cannot be overridden by instance or object level properties. They are grayed out in the *Shape Instance Parameters* dialog box.

- Dynamic Fill: *Smooth*, *Rough*, or *DISABLED*
- Artwork Format: Raster or Vector options
- Minimum Aperture for respective artwork chooseion

The *Disabled* dynamic fill option essentially disables the auto-voiding algorithm. If wholesale editing occurs, and the performance of *Rough* dynamic fill mode is unsatisfactory, using the *Disabled* dynamic fill option may prove to be the fastest method of making numerous changes. Once changes are made, update to either *Rough* or *Smooth*, depending on the time available for this update.

Artwork Format

The *Artwork Format* field on the *Global Dynamic Shape Parameters* dialog box aligns and improves voiding quality with the artwork style you output. Many formats are offered, but effectively there are only two: raster and vector. Vector based artwork, such as 274D, always uses a circular line for the display and chooses the artwork draw from an aperture list for shape filling. It cannot draw a perfectly squared edge because it is limited to filling with a circular aperture. Raster formats, such as 274X, can have edges smoothed with chamfered or round styles since no aperture is needed.



Using Min Aperture for Gap Width Control

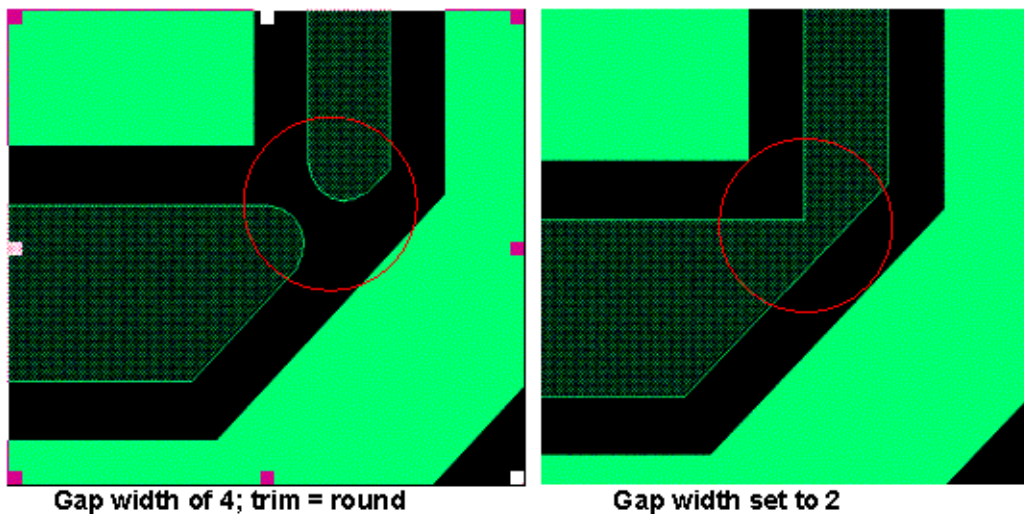
Working with Shapes

Dynamic Positive Shapes

Trim control

The *Acute Angle Trim Control* field is used with raster sections only and cuts out a sharp acute angle and flattens it so that you can draw with an aperture relative to the *Minimum Aperture For Gap Width* setting.

Gap width is defined as the distance from the edge of a void to the edge of a void. A larger value produces a greater number of clearances between objects; a lesser value produces greater continuity within the copper but a higher number of potential slivers. Gap width should be in-line with the minimum trace size your PCB fabricator can etch.



It may be necessary to re-void shapes to achieve a contiguous flow of copper throughout. Islands or unconnected fragments of copper may develop as a result of certain parameter settings, including gap width and clearances. Fragments within the shape can be suppressed or eliminated automatically using the *Suppress Shapes Less Than* option on the *Void Controls* tab. The entered value represents an area setting that controls what is removed from the shape. For example, if a value of 25 is entered, any shape fragment with an area less than 625 mils is deleted. Use *Display – Element* on the shape to obtain the shape area value. Use *Shape – Delete Islands* to detect and remove any remaining fragments. Use shadow mode control with a very low intensity to detect any additional islands throughout a shape.

Clearance values

Pins now separate into thru hole and smd categories. The *Use Thermal Width Oversize* of option lets you add more clearance around objects within the shape boundary.

Working with Shapes

Dynamic Positive Shapes

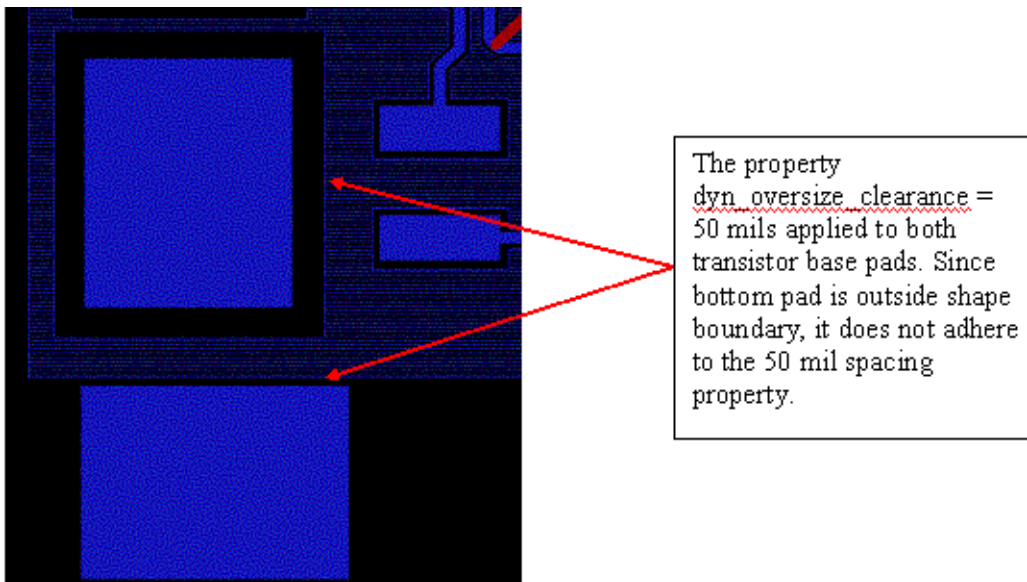


Tip

If round-off errors occur after auto-voiding, update the *Clearances* tab of the *Global Dynamic Shape Parameters* dialog box. Add a uniform small oversize value to all fields under *Oversize Value*. Cadence recommends a value of twice the smallest unit/accuracy of the design to eliminate small spacing problems.

For example, if the database were set up as mils with one place of accuracy, and the DRC rule equals 5.0 with a roundoff DRC error of 4.9, then enter 0.2 as the oversize to void. For a database with two places of accuracy, enter 0.02. In parallel, when designs require a global update, and Gerber 4x or 6x is the artwork format, Cadence recommends increasing the *Minimum Aperture For Artwork Fill* value by the same number used in the *Oversize Value*. (The aperture in the wheel should remain unchanged.)

For instance-based applications, use the property `DYN_OVERSIZE_CLEARANCE`. The following graphic explains the use of pin-level properties applied to the two large component pads. Each has been assigned a value of 50 mils. Since the bottom pad is outside the shape boundary, the property is not applicable.



Thermal relief connect lines

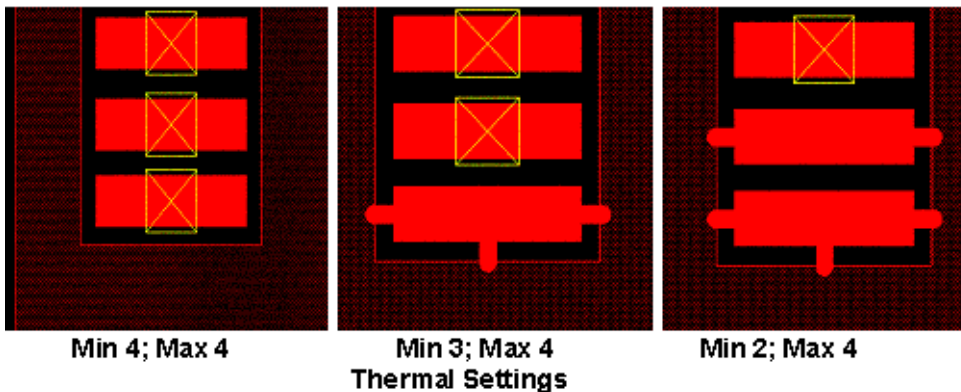
Thermal relief connect lines can be automatically constructed on pins and vias. Pins have been classified into Smd and Thru Hole. Additionally, a minimum and maximum setting can be applied to thermal connects as long as the option is not set to direct, in which case, Allegro PCB Editor sets min and max equal to 0. If Allegro PCB Editor cannot populate to the

Working with Shapes

Dynamic Positive Shapes

minimum thermal requirement, it leaves a via or pin free of any thermal connects. Consider using a min value less than the max to achieve the best coverage. Review current carrying requirements with the electrical engineer to determine if this is an acceptable practice.

In the following example, SMT pins require a connection to GND. With in-line pins, it is common to populate two spokes on inner pins and three on the outer if pin-escaping vias are not used. In the example below, the min/max setting controls thermal connections. The graphic on left is free of thermals, due to the requirement of four minimum connects. Since four is not possible, no thermal connect is applied. The middle graphic reduces the minimum connect to three where it is possible to populate but only on the end pin. With the minimum value set to two, the inner pins now properly connect while the end pin remains at three as the right graphic shows.

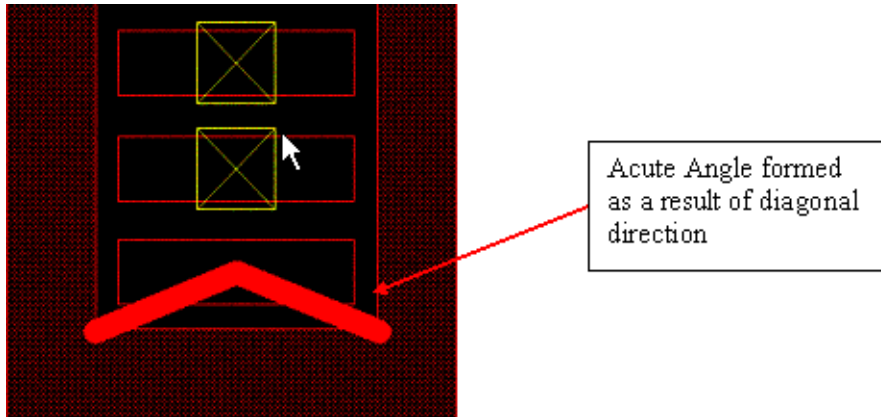


When applying thermals to SMT pins, use the *Orthogonal* rather than *Diagonal* setting, at least for oblong pins. Allegro PCB Editor populates thermals on a polar axis with each 22.5 degree interval a possible location. Because a circle is 360 degrees, 16 locations exist to potentially place a thermal connect line. In addition, Allegro PCB Editor does not account for pad exit directions for clines; therefore, the probability for acute angles becomes greater. In the same example, changing the type to *Diagonal* causes the thermal connect on the outer

Working with Shapes

Dynamic Positive Shapes

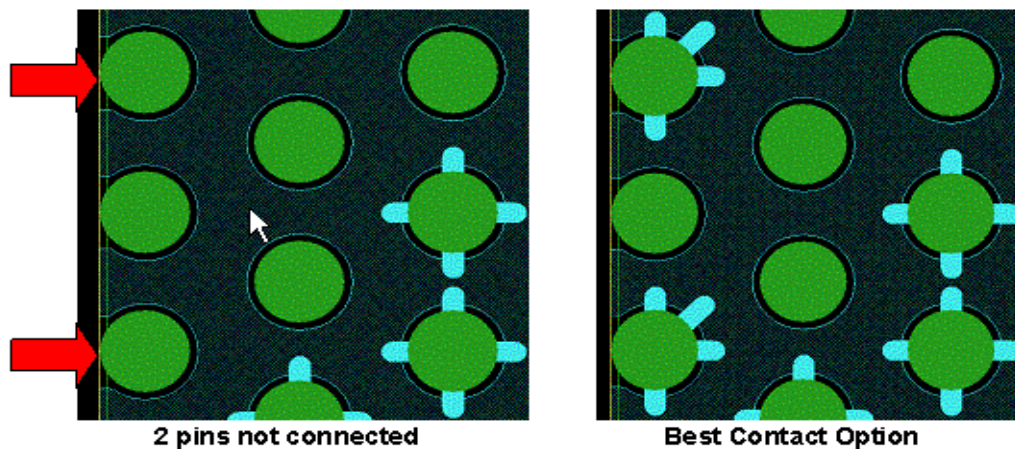
pin to produce acid trap violations, and the adjacent pin is not populated due to DRC conditions.



Best Contact

The *Best Contact* option help populates thermal spokes by adding clines where possible. Odd angle connects may result, but the intent is to meet the minimum requirement and override the connect settings to do so. In the example below, the left graphic shows a boundary placed close to a column of connector pins. This proximity prevents the pins with the adjacent red arrow from making a connection when thermal connects call for a minimum and maximum of 4.

Using the *Best Contact* option as shown in the right graphic, a cline added in the 45 degree location satisfies the requirement. When the *Best Contact* option is chosen, Allegro PCB Editor examines the pin or via object and determines the best location for populating thermals: 45 degree locations take priority over 22.5 degree locations.



The *Use Thermal Width Oversize of* field adds the value you specify to the default thermal connect line width, using `min line width` as the default. Consider your general routing line width strategy and thermal spoke width because Allegro PCB Editor uses one source for both applications. It may not be practical to set up GND for 50 mils and also use thermal spokes in a non-direct contact mode. Using direct contact for thermal applications mitigates this issue.

The *Use Fixed Thermal Width of* field controls thermal line width directly from the *Shape Instance Parameters* dialog box and independently of physical constraint set mappings. Prior to 15.2, thermal line width derived only from the physical constraint set, hampering control of power/ground routing and thermal line width using a single set of constraint values. For example, you may want GND routing to be 25 mils, but spoke width of 10 mils. In this scenario, the physical constraint set for GND routing would have a min line width set to 25 mils; on the *Shape Instance Parameters* dialog box, the fixed thermal width be set to 10 mils.

The *Shape Instance Parameters* dialog box defaults to an oversize value of 0 mils to avoid uprev problems. The user must choose the *Use Fixed Thermal Width of:* option to control thermal line width from the *Shape Instance Parameters* dialog box.



Tip

Direct contact thermals help improve dynamic shape auto-voiding performance. The PCB Industry generally accepts direct contact of copper on vias.

Shape Instance Parameters

It may become necessary to override global parameters on a shape-instance level. You may want extra pin or via clearance or different thermal connect styles for a particular shape. There are two methods to accomplish this:

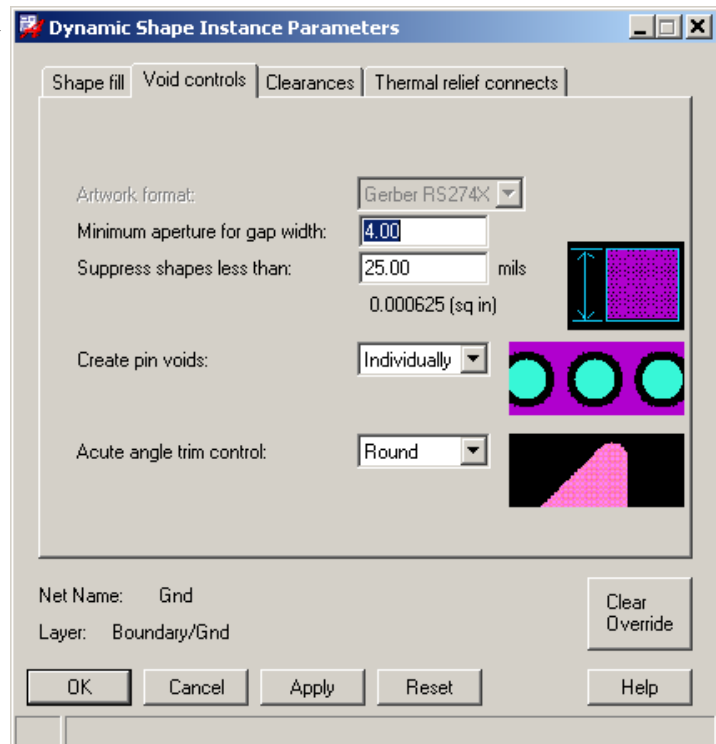
- by adding the new shape, right mouse clicking, and choosing *Parameters* from the popup menu, which displays the *Shape Instance Parameters* dialog box. Changed values appear in a bold blue, which follows the lead from Constraint Manager.

Working with Shapes

Dynamic Positive Shapes

- by choosing an existing shape, right mouse clicking, and choosing *Parameters* from the popup menu.

Header reflects
instance-based
parameters



Object Level Properties

Use *Edit – Property* to apply the third level of shape parameters to objects. Once applied, these values override instance and global parameters. Object-based properties can be added in the Allegro PCB Editor session and also at the library level, where it may be best to add certain properties routinely designed into all boards designed in your group. For example, you may want to associate a thermal connect of a particular size with an inductor pin or to limit thermal connects to two on a 0603 package type, but maintain four thermal connects on all 1210 packages. (These package types are used for resistors and capacitors, and the numbers represent the Width x Length of the geometry.)

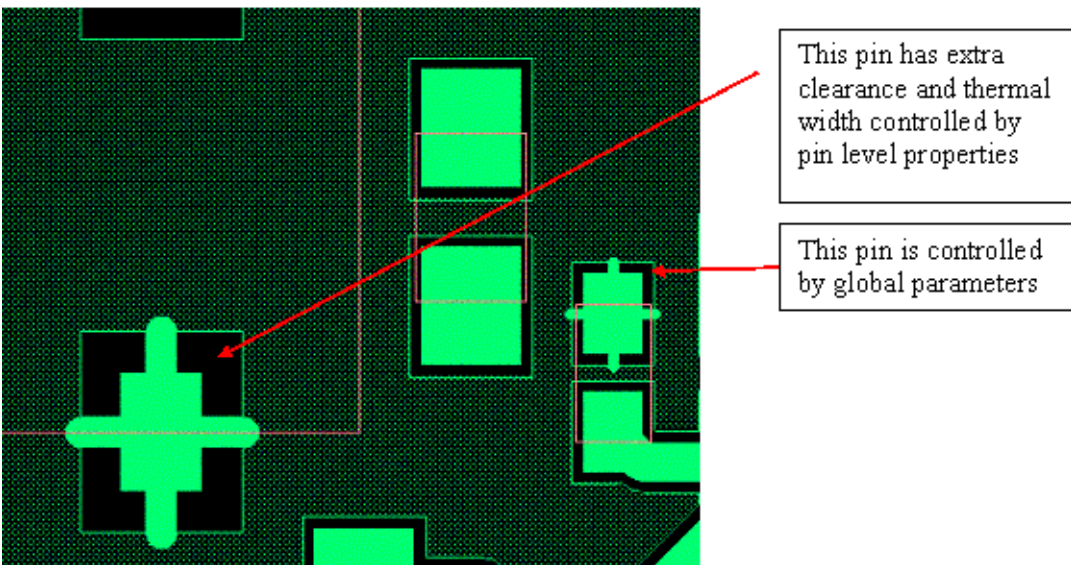
Working with Shapes

Dynamic Positive Shapes

There are seven properties associated with dynamic shapes:

Delete	Property	Value
<input type="checkbox"/>	Dyn_Clearance_Oversize	<input type="text"/>
<input type="checkbox"/>	Dyn_Clearance_Type	<input type="text"/>
<input type="checkbox"/>	Dyn_Max_Thermal_Conns	<input type="text"/>
<input type="checkbox"/>	Dyn_Min_Thermal_Conns	<input type="text"/>
<input type="checkbox"/>	Dyn_Oversize_Therm_Width	<input type="text"/>
<input type="checkbox"/>	Dyn_Thermal_Best_Fit	<input type="text"/>
<input type="checkbox"/>	Dyn_Thermal_Con_Type	<input type="text"/>

The following graphic displays the use of the properties `DYN_CLEARANCE_OVERSIZE` and `DYN_OVERSIZE_THERM_WIDTH`, added using *Edit – Property* with the *Find Filter* set to *Pin*. Once added, these properties remain on the pin even if the shape is deleted and re-added.



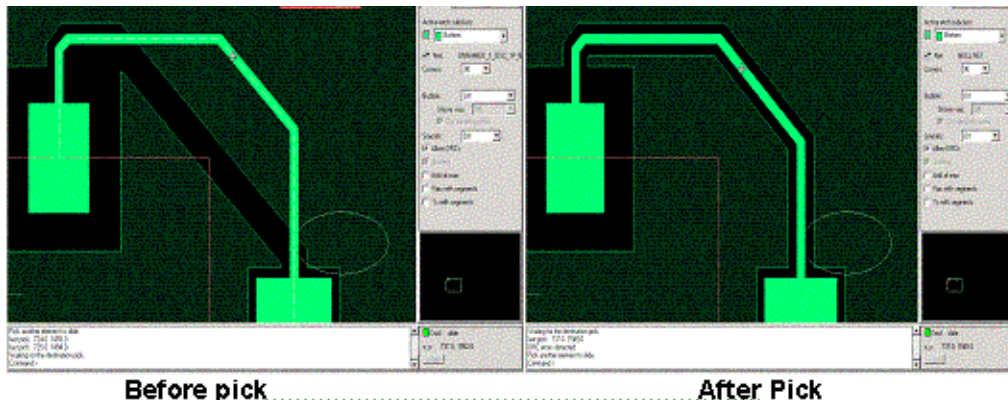
Interactive Etch Editing

A benefit of dynamic shapes is using the interactive and automatic routing tools without deleting and re-adding the shape. Routing a cline within the shape is commonly known as plowing, while the re-pouring of copper in areas vacated by a cline or via is known as healing. Allegro PCB Editor's interactive etch editing tools can be used to add clines or vias within the shape while having the shape dynamically auto-void after each change. The actual database updating does not occur until you left mouse click.

Working with Shapes

Dynamic Positive Shapes

The following graphics show *Route – Slide* moving a cline. On the left, the cursor has moved the cline, but a left mouse click has not occurred yet. On the right, the cline is instantiated and committed to the database.



Before using *Route – Connect*, *Route – Slide* or *Edit – Vertex*, understand the limitations.

- Outside the shape: While performing edits outside the boundary of a dynamic shape, any resulting bubbling external to the shape does not compromise the integrity of a dynamic shape. You cannot shove etch into a dynamic shape intentionally or accidentally. To slide a cline that is outside the shape, disable bubble mode, then slide the cline.
- Cutting Corners: When a cline cuts through a shape, the resulting action produce either an island or a conducting fragment. Either type of fragment may become suppressed pending the value you have assigned to the *Suppress Shapes Less Than* option. If suppressed, the boundary remains unchanged allowing the suppressed area to heal in the event the cline is removed. The graphic on the left shows a cline being added and cutting the shape resulting in the formation of an island. The graphic on the right has the cline retracting from within the shape resulting in the healing of the voided area.

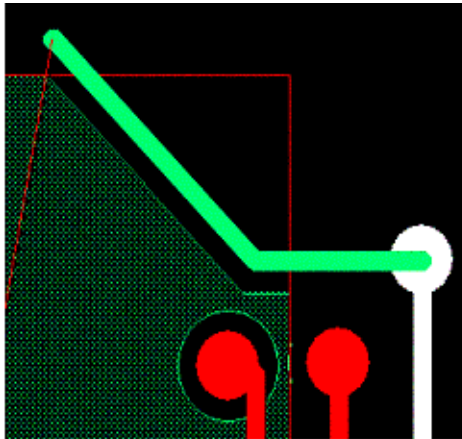
Working with Shapes

Dynamic Positive Shapes

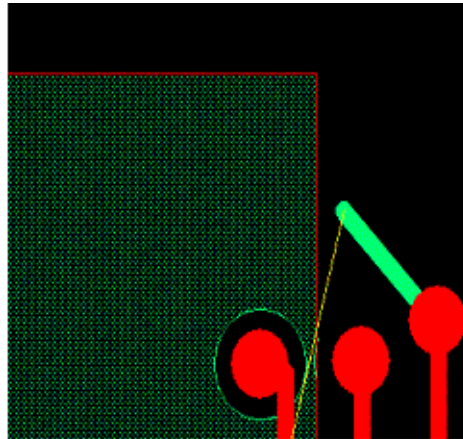


Tip

If routing into a shape from outside, it may be best to disable the bubble options. *Hug preferred* does not allow a cline to enter a shape as it tries to hug around it; with *shove*, multiple instantiations may be necessary to control the path.



Cutting a Shape



Retreating back to the Boundary

Performance Considerations

Etch editing within a dynamic shape triggers the auto-voiding algorithm; however, it is localized to where the dynamics occur. The complexity of the shape is directly proportional to the performance. If your shapes are *Smooth* and etch editing performance is sluggish, consider changing the *Dynamic Fill* mode to *Rough*. If wholesale editing occurs, choose the *Disabled* mode, then update later.

Interface to Allegro PCB Router

When dynamic shapes pass to Allegro PCB Router from Allegro PCB Editor, vias can pop through the shapes; however, routing is not be allowed through them. If via popping is not preferred, convert the shapes to static in Allegro PCB Editor to prevent this.

Outputting Manufacturing Data

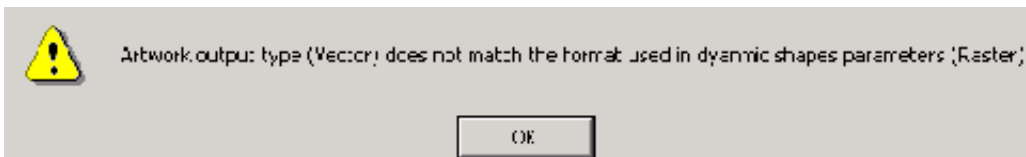
Before artwork can be created, all dynamic shapes must be updated to *Smooth*. Shapes not created as *Smooth* are labeled as *out of date shapes* as shown in the *Status* dialog box below. Use the *Update to Smooth* option to update all out of date shapes. Failure to do so prevents you from generating artwork data.

When dynamic shapes are out-of-date, Allegro PCB Editor and APD display a *Dynamic Shapes Need Updating...* button on the *Artwork Control Form* dialog box. If you try to use the *Create Artwork* button on the *Artwork Control Form* dialog box, an error message appears:

Dynamic Shapes are out of date, please update them.

Click *Dynamic Shapes Need Updating...* to open the *Status* tab of the *Status* dialog box, which becomes active, blocking any use of the *Artwork Control Form* dialog box until you update dynamic shapes or DRCs before proceeding with artwork.

A warning also appears if the shape artwork format does not align with the artwork device type setting. This is limited to comparing raster against vector and not individual types.



The shape boundary should not be part of any film records. Although it has a zero width outline, it could be output accidentally and assigned an aperture width.

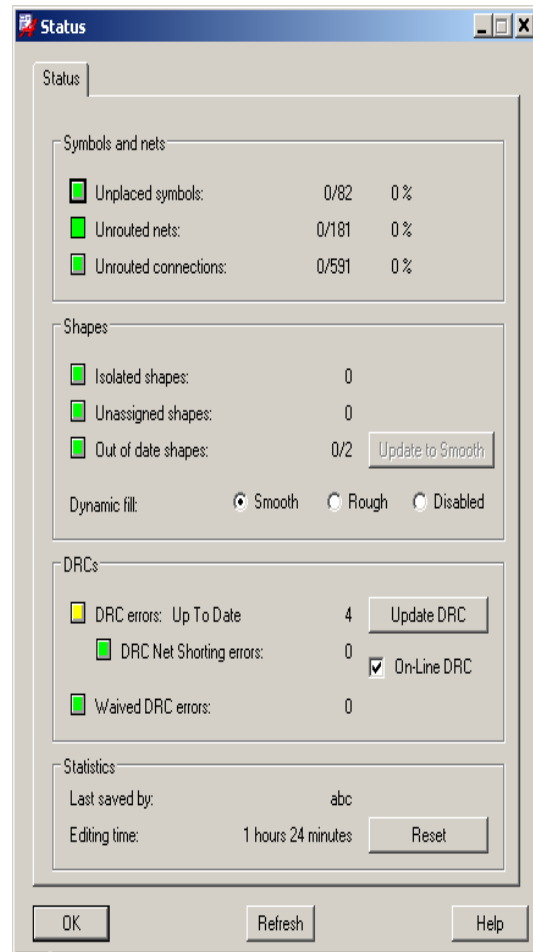
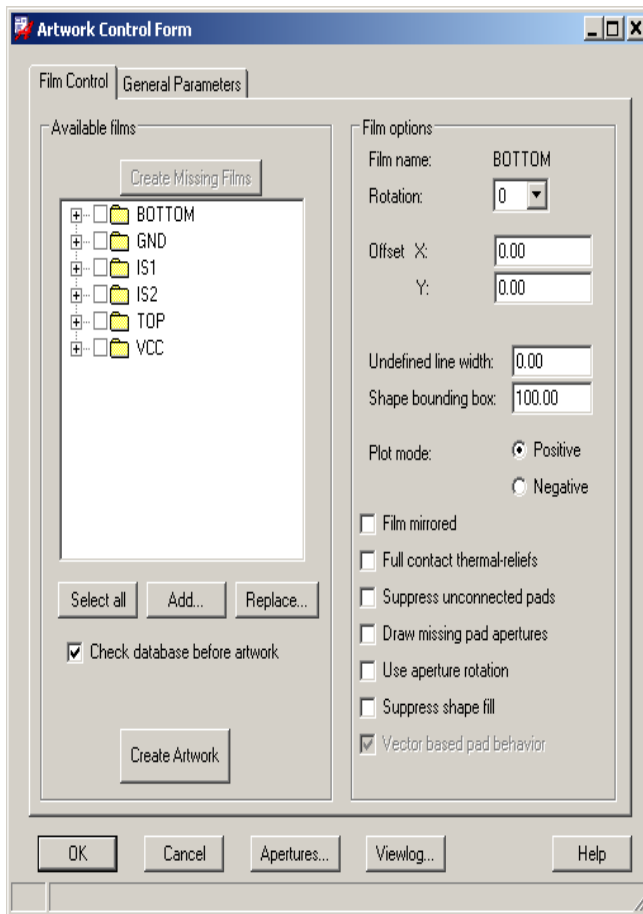
Working with Shapes

Dynamic Positive Shapes



Tip

DO NOT include the boundary class in artwork film records. Although Allegro PCB Editor only assigns pin, via, and etch classes, the user may add more class information. Adding a boundary class could create artwork continuity problems.



User Preference Variables

At present, there are eight shape related variables. Choose *Setup – User Preferences* and choose the *Shape -General/Voiding* category. Variables beginning with *av_* are for static applications only.

Working with Shapes

Dynamic Positive Shapes

- *av_endcapstyle*: the void around a cline appears in this style. Defaults to round. Takes effect when pad/via/pin is less than the connect line width.
- *av_inline*: controls the distance between pins during autovoid processing to determine if pins are voided as a group.
- *av_thermal_extend*: Controls the distance the thermal connect cline infringes into the shape. Default is 5 mils.
- *highlight_shape_net*: Highlights the net associated with the chosen shape.
- *no_shape_fill*: Shapes are drawn without fill or skeleton if set
- *old_shape_fill_style*: 15.0 introduces a transparent stencil fill pattern. Set this variable if you desire the 14.x solid fill style.
- *dv_endcapstyle*: applies to dynamic shape voiding. It affects the geometry of voids created around cline end points. It allows shape voiding if the default mode (circle) shows problem.
- *pad_drcplus*: If small round-off errors occur between pin and shape, set this variable with a numerical value that adds additional clearance on all auto-voids from pin to shape.
- *shape_add_filltype*: determines the initial shape type used for adding shapes on Etch. Default is dynamic.
- *shape_arcmode_nonsticky*: controls the arc behaviour of the edit shape/void boundary commands. This variable on setting will automatically changes the segment type to line after an arc has been added.
- *shape_drag_move*: If enabled, allows you to move a shape using the LMB drag. You must be in shape edit mode to drag a shape.
- *shape_local_temp*: if design is located on network file server. This variable uses standard OS TEMP or TMP as the location for these files.
- *shape_merge_props*: merging properties also on merging two shapes.
- *shape_no_ripupthermal*: changes the default response for the static shape thermal rip-up question form from Yes to No. In this case, thermals will not rip up.
- *shape_no_clip_rk*: adding an etch shape that crosses the route keepin, will be clipped to the route keepin. Shapes added outside of the keepin are not clipped and will not generate DRCs. Restores pre 15.5 functionality where no clipping was done.
- *shape_rki_autoclip*: allows automatic update of dynamic shapes to the route keepin. By default, dynamic shape boundaries, are clipped to the route keepin.

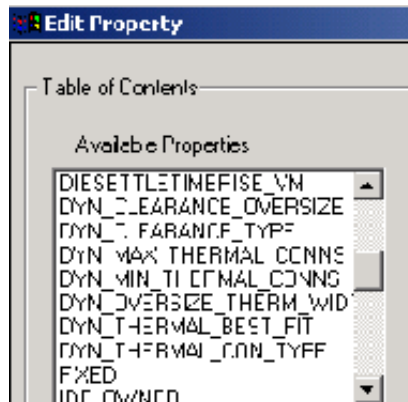
Questions and Answers

Is Smooth the same as WYSIWYG?

WYSIWYG became Smooth in 15.1 to better represent its functionality.

Can you change the thermal relief for one pin only ?

Yes, use *Edit – Property* with the *Find Filter* set to *Pin*, choose a pin, then a dynamic shape property. Once applied, these objects that inherit these properties take priority over global and instance level parameters.



Do oversize clearances accept negative values?

No.

Wouldn't the thermal relief be based upon the thermal model for the pin?

The parameter settings or object-level properties control thermal reliefs for dynamic positive planes. The `.fsm` file associated with the padstack controls thermals on negative planes.

When you change the global parameters, do existing shapes automatically update?

Global dynamic shape parameter changes update existing shapes unless they have overriding values set.

Can you specify a Gerber output on the *Artwork Control Form* dialog box that differs from that chosen on the *Global Dynamic Shape Parameters* dialog box?

A disparity can exist between shape and artwork settings. When you attempt to run artwork, a warning appears that identifies any difference between vector and raster settings.

How would you define a spacing constraint where copper must be kept away from pads, clines, etc. by the cline width?

Use the spacing constraint set to control shape-to-pin, -via, -line, or -shape spacing with a cline value. When setting shape parameters, use the *DRC* option in the *Clearances* tab. No absolute mode for clearance values exists. If a common line width cannot represent shape-to-object clearance values, consider offsetting with the oversize clearance option using a positive not a negative increment.

Why does the shape not merge with the other GND?

Shapes will not merge automatically. Use *Shape – Merge Shapes* if shapes are assigned to the same net.

The minimum number of spokes was defined as two; the maximum, four: Why is the default two? Wouldn't two be populated only if four were not possible?

In *Rough* mode, a maximum of two thermals on the same axis will be populated to expedite shape processing. In *Smooth* mode, PCB Editor tries to meet the maximum spoke requirement. If the minimum is not met, no thermals are populated. Consider using the *Best Contact* option on the object, which adds more thermal spokes on angles down to 22.5 degrees.

Is overlapping shapes cause DRCs, even with the same net?

Yes, overlapping shapes of the same net report a DRC in 15.x. DRCs are not reported if dynamic shapes are in *Rough* mode.

What is the difference between dynamic and static shapes?

Static is analogous to pre-15.0 shapes that require interactive voiding. Dynamic shapes automatically void when the shape is edited, for example, routing or moving components.

Editing global parameters for dynamic shapes does not change the `env` file, but edits to the settings on the *User Preference Editor* does. How can the edits made on the *Global Dynamic Shape Parameters* dialog box become the norm, that is, in the `env` file?

Shape parameters are not written to the `env` file. The *User Preference Editor* controls settings apart from those on the *Global Dynamic Shape Parameters* dialog box.

Do global dynamic shape parameters update automatically, or do you have to edit shapes to update them?

Out-of-date shapes update to *Smooth* without requiring shape modification. Changing dynamic fill mode from *Smooth* to *Rough* requires editing of the shape, such as moving the shape a distance of 0 to trigger an update.

When creating a shape, can you automatically identify islands?

Run the `island delete` command to detect islands, which you can delete by instance or globally.

If you use different parameters for different shapes, is there any way to understand the parameter setup for that shape after the changing the parameters?

Choose the shape, right mouse click ,and choose *Parameters* or *Reports* from the popup menu that displays. For a summary of all shapes, use the Shapes Report, available by choosing *Tools – Reports*.

If you have many complex shapes, will performance suffer? Is there a better way to approach this?

We recommend maintaining *Rough* mode or even *Disabled* mode until the design is nearly finished when using shapes that tend to flood layers. With the 15.0 dynamic shape fill, consider reverting to the solid fill as in 14.2 by changing the variable in user preference. There should be a 20 to 25% performance increase in 15.1.

Why don't the voids of the Cross-Hatched shapes have the same border as the boundary?

They are consistent.

Is there supposed to be a border line around the void areas?

Yes, enable the boundary subclass in the *Color* dialog box, and choose *Stackup*.

Is there autovoicing for static shapes?

Use *Shape – Manual Void/Cavity– Element*; choose a shape, then choose an area to void by window.

Are complex shapes created in the Harmony software compatible with 15.1 when changed to dynamic shapes in Allegro PCB Editor? Software crashes occur when dynamic shapes are changed to *Smooth*.

We do not test Harmony internally as this is an OHIO Design product. Send us a testcase and script if possible that recreates a crash condition.

Can an aperture be changed for a specific shape instance?

Apertures can only be controlled at a global level.

Can you erase spokes if you didn't need all of them on one pin?

Spokes generated in dynamic shapes cannot be deleted or moved. You must use instance- or object-level properties to customize.

Do these object-level properties extract when you export a netlist with properties?

Shape properties cannot be assigned at the net level. They must be assigned to pins, vias, or clines.

Is there a trick to using copper pour and keeping the database small?

None of which we are aware.

If you delete an island, and subsequently want to add it again manually, can you?

A manual void represents the island. Use the *Shape – Manual Void/Cavity– Delete* command, and choose the island boundary.

Does *Shape – Compose Shape* use the centerline for the finished border or the outside edge of the line?

It uses the center line.

Do negative planes still have the problem creating a complex plane, that is, one net inset entirely within another net?

The 15.0 shape environment in this regard is similar that of 14.2. Your artwork choice factors into the design of complex planes. The embedded shape problem is a limit in vector artwork output. Dynamic shapes do not change the requirement to suppress shape fill and add your own fill lines for negative shapes within shapes when using vector artwork.

Can shapes be disabled before passing to Allegro X PCB Router?

Shapes pass to Allegro X PCB Router as wire keepout areas, allowing vias to be added through shapes.

Does the line width matter for creating the border of the shapes?

The shape boundary width is not user controlled.

Can thermals be mapped to flash symbols?

No, a separate thermal line width control exists in the *Shape Instance Parameters* dialog box. Currently physical constraint sets handle this issue.

Can dynamic shapes be added to soldermask layers?

No, only to electrical subclasses.

Are there plans for additional shape controls from the right mouse button?

Preview the 15.2 popup menu by running the *Shape – Select Shape or Void/cavity* command and then right mouse clicking.

Can I control a thermal by direction, for example, N S E W?

No, available properties appear as in the above screen capture.

Working with Shapes

Dynamic Positive Shapes

Why don't shapes obey anti etch?

Initially they do when using the `split plane create` command; however, any modifications to the anti etch line requires re-execution of the `split plane create` command.