

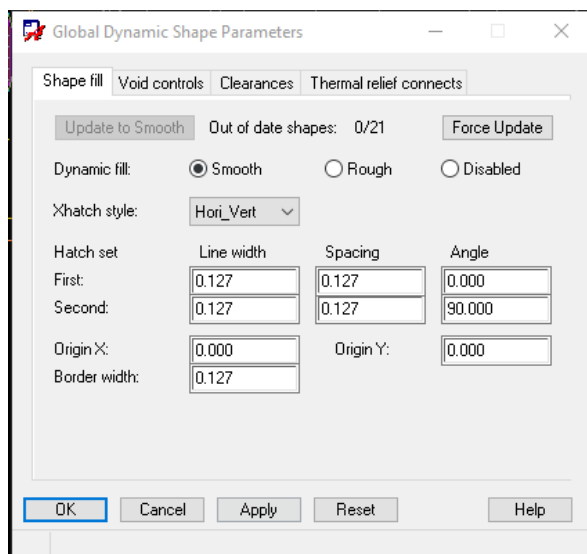


Introduction

PCB Editor offers users the ability to define how a Dynamic Copper Shape is displayed and how it reacts to items that are either connected or not connected. This technical note describes how to setup and use these settings.

Shape Settings

The setting control for Shapes in PCB Editor is effectively hierarchical. You have the ability to set controls globally, on a shape by shape basis or even at pin or via level. To setup the global shape settings use Shape – Global Dynamic Parameters. The following GUI will appear:-



Shape fill Tab

Update to Smooth - Click to automatically void and run DRC on all dynamically filled shapes, making all dynamic shapes up-to-date (Dynamic Copper Fill mode set to Smooth) and produce artwork-quality output (regardless of whether you chose Rough or Disabled in the Fill Mode field). This changes the current Dynamic Copper Fill mode to Smooth. To cancel dynamic filling of complex shapes for a large design, you can use the Esc key to stop the process, which leaves the shapes out of date. If several shapes are in the midst of dynamically filling when you invoke the Esc key, Shapes already dynamically filled remain completed. Shapes in the process of dynamically filling remain unfilled and marked out of date. Shapes whose dynamic fill is yet to be updated remain filled

but marked out of date. (This option can also be invoked from the Display – Status command.)

Out of date shapes - Lists the number of dynamic shapes for which the Dynamic Copper Fill mode has been set to Rough or Disabled.

Force Update - Updates all dynamic shapes, including those that have voiding, artwork smoothing, and thermal relief generation.

Choose one of these copper fill options that controls the update of all dynamically filled shapes globally.

- Dynamic Fill**
- Smooth:** Choose to automatically fill, void, and run DRC on all dynamic shapes and produce artwork quality output. If this field is disabled, subsequently choosing it automatically updates existing out-of-date shapes.
 - Rough:** Choose to see connectivity without full edge smoothing and thermal hook-ups in a fast fill mode to obtain true clearances around elements and resolve intersections with other voids. Artwork quality output and artwork are not created.
 - Disabled:** Choose to globally defer dynamically filling all dynamic shapes you subsequently create or modify. Use this option to edit etch for medium to large ECOs, manual ECOs or to run batch programs such as netin, gloss, testprep add/replace vias, for example, for faster performance. Shapes created under this global setting are “out of date” because they are not filled or voided, and DRC does not run, and as a result artwork cannot be produced. If this field is disabled, subsequently choosing Smooth or Rough automatically updates existing out-of-date shapes.

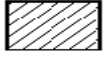
Xhatch style - Click the drop-down list and choose one of the listed styles, as shown below. The configuration of the displayed field options depends on the fill style you choose.



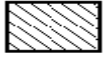
Vertical – Vertical lines only.



Horizontal – Horizontal lines only.



Diag_Pos – Diagonal +45 lines only.



Diag_Neg – Diagonal -45 lines only.



Diag_Both – Diagonal +45 and -45 Lines.



Hori_Vert – Horizontal and Vertical Lines.

Hatch Set - A hatch set is the set of parallel lines that the layout editor creates to fill a crosshatched shape. It creates one or two sets according to your choice of style, and if two sets, usually at 90 degrees to each other. You can set the width, spacing, and angle for each set independently. If you change any setting after choosing one of the styles above, the layout editor changes the choice in the Xhatch Style field to Custom.

Linewidth - Sets the width in user units of the crosshatch lines for each set. The line width must be less than or equal to the Border Width specified.

Spacing - Sets the center-to-center spacing in user units of the crosshatch lines for each set.

Angle - Sets the angle in degrees of the crosshatch lines for each set.

Origin X and Origin Y - Sets the x,y coordinates of the crosshatch lines.

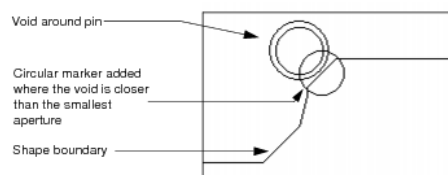
Border Width - Specifies the width of the shape boundary line. The border width cannot be smaller than the line width.

Void Controls Tab

Controls the art check routine, which verifies that the copper area can be filled properly when creating the artwork (Gerber) files for a layer. You can specify how small copper islands are treated, and affect whether a copper area is contiguous or split. To simplify clearance areas, the settings on this tab evaluate pin patterns and control the merging of multiple polygons into one copper area (if on the same net).

Artwork format - Specifies an expected artwork format (raster/vector) and determines the next displayed field option. If you choose Gerber 4x00 or 6x00, Minimum aperture for artwork fill becomes the next displayed field option; otherwise Minimum aperture for gap width displays. Cannot be overridden at the shape-instance parameter level. This MUST match the gerber format set under Manufacture – Artwork – General Parameters tab.

Minimum aperture for artwork fill - Specifies the width in user units of the smallest plotting aperture to the artwork command. For solid fill style only. This is a shape-instance value. Note - When you prepare artwork, the voids do not fill when a void and the shape boundary are closer than the smallest aperture. All points like this are marked with a circular figure on class MANUFACTURING, subclass SHAPE PROBLEMS. However, the voids are added as shown in Figure below. When you fill the shape, all the MANUFACTURING/SHAPE PROBLEMS markers are deleted.



Minimum aperture for gap width - Specifies the smallest allowable space between two voids or a void and the edge of the shape. This is a shape-instance value.

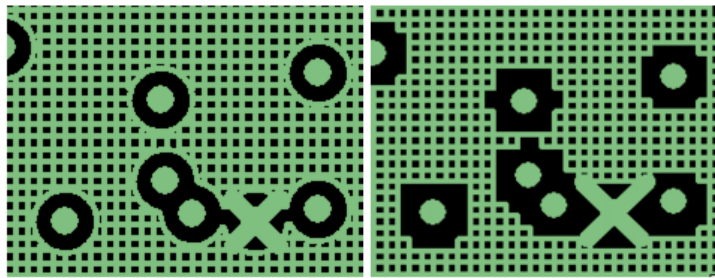
Suppress Shapes Less Than - Specifies in user units that any shape areas smaller than the area value specified here be suppressed. Dynamic voiding can split a shape into multiple shapes. Any shapes smaller than the surface area you specify in this field are ignored.

Create Pin Voids - Generates voids around a series of pads, mainly DIP patterns, either In-Line or Individually. In Line correlates to drawing one void around the entire group of pads. Individually correlates to drawing a void around each pad separately.

Distance between pins - A maximum inline distance parameter only visible when inline voiding is chosen. Replaces the av_inline environment variable.

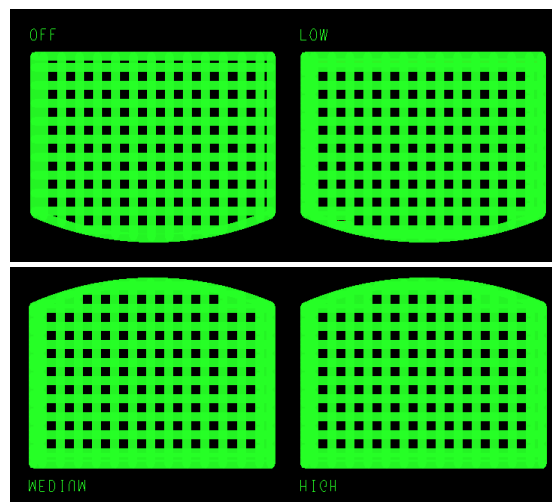
Acute Angle Trim Control - Specifies solid outline corner style (round or chamfered) for solid shapes for raster artwork formats. The minimum gap width is used for the corner radius (round) or length (chamfered).

Snap Voids to Hatch Grid - Attaches the created voids to the hatch grid. For crosshatched shapes only. The example on the right below shows how the void snaps to the hatch grid if this field is enabled; on the left, if this field is disabled.



DiffPair combined void for vias added with Return Path option - Enables creating a single oblong void for Diff Pair vias when the vias are part of a Diff Pair return path via group. Return path vias can currently only be added with Allegro PCB Designer + High Speed Option.

Fill Xhatch cells - Fill xhatch cells from low to high. The choices are Off, Low, Medium, and High. The Off does no filling whereas High completely fills a cell that has an intersection with a void or shape boundary. **Note:** Fill Xhatch cells does not work with Snap Voids to Hatch Grid option.



Clearances Tab

Specifies how far the copper is recessed from any conductive features contained within the copper area to prevent shorting. These include thru and SMD pins, vias, lines and clines, shapes/rectangles, and text. The choices are:

Thermal/Anti - Makes a void the size of thermal relief and antipad as defined in the padstack of a pin or via. This applies only to the pin clearances. If antipad clearance is smaller than the DRC values, voiding increases the clearance to the DRC value.

DRC - Makes a void sized using the DRC distance as clearance around the pad. This is the recommended option. The DRC distance is based on the Constraint Manager settings.

Oversize Value - Increases the clearance beyond the specified DRC or thermal/ antipad value for elements requiring voiding that are inside a shape boundary. Use this value as an alternative to setting up spacing constraints or if the specified DRC value is especially small. This value is not meant for use with shape keepouts.

Thermal Relief Connects Tab

Specifies how pins and vias with the same net name as the shape should be connected to the shape.

Thru/SMD Pins/Vias - Indicates how clines are to be generated. The three options are: -

Orthogonal - Connects straight up-down or left-right. The pin connects directly to the void outline or hatch lines.

Diagonal - Connects diagonally upper left to lower right and lower left to upper right.

8-Way - Connects lines from the thermal relief to the pin/via both diagonally and orthogonally.

Full Contact - Creates no voids. For crosshatched shapes, the hatch lines provide the connections, or the layout editor adds short connect lines.

Best Contact - Choose to rotate the thermal relief lines in 15 degree increments and override the chosen thermal connect style if it doesn't provide sufficient thermal connects within the specified minimum and maximum number of thermals. Use this field when the number of connected pins takes precedence over the placement (angle between thermals).

Use Fixed Thermal Width of - Controls thermal line width independently of physical constraint set mappings.

Use Thermal Width Oversize of - Adds the value you specify to the default thermal connect line width, which originates in the layout editor's Physical (Lines/Vias) Rule Set Etch Values dialog box, using min line width as the default. Consider your general routing line width strategy and thermal spoke width because the layout 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.

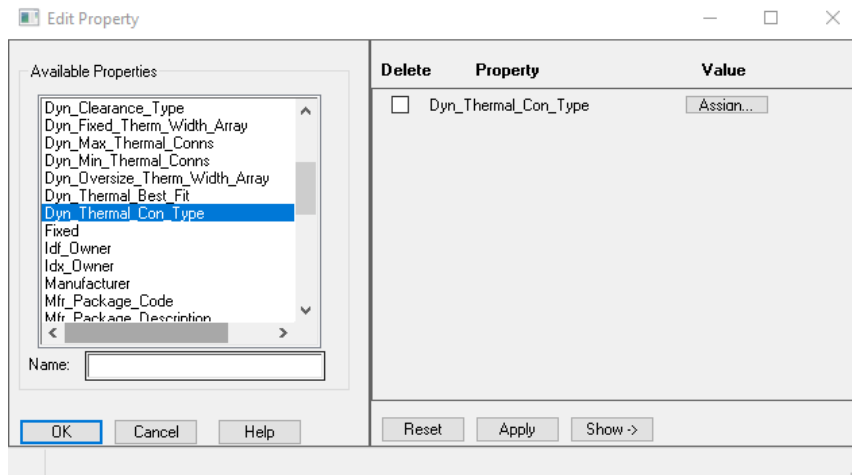
Use xhatch Thermal Width - Allows cross-hatched dynamic shapes to generate thermal clines widths based upon the shape's cross hatch width.

Minimum Connects - Specifies number of connections for thru/SMD pins and vias (choosing Full Contact disables this field) in conjunction with the Maximum Connects field to control the number of thermal connections. This ignores existing clines that connect to the shape.

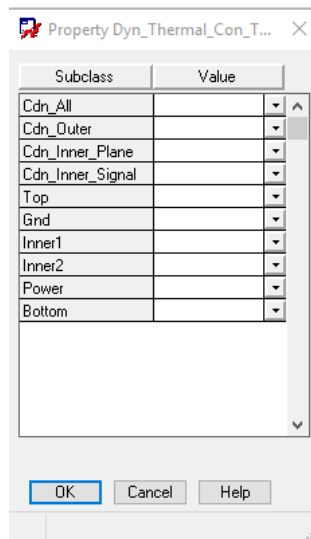
Maximum Connects - Specifies how many connect lines are created for the thermal relief. Up to four are allowed on orthogonal and diagonal; up to eight on the 8-way option.

If you want to modify a single shape rather than affect the Global Parameters use Shape – Select Shape or Isolation/Cavity (OrCAD) or Shape – Select Shape or Void/Cavity (Allegro), then LMB (left mouse button) the required shape, then RMB (right mouse button) – Parameters. The standard GUI specified above will appear but any changes made will only affect the shape you initially selected.

In many instances you may need to modify how a specific pin or via is connected to the Dynamic Copper Shapes. For these instances you will need to use the Property Editor to apply pin or via level properties to override the Global or Shape level parameters. To do this use Edit – Properties (Allegro) or Edit – Object Properties (OrCAD), set the Find Filter to just Pins or Vias then select the pin or via with the LMB. Alternatively hover over the required pin or via and use RMB – Property Edit. The following GUI will appear: -



The list of available properties is on the left. Select the property required with the LMB, if it will appear on the right hand side of the GUI, you can then set any value required based on a layer by layer setting. To do this select the Assign button and then set the values required. The actual layers listed are the layers of the PCB as defined in the Cross section (Setup – Cross Section) or you can choose the Cdn_All, Cdn_Outer etc to set different values for all layers, just the outer layers and so on.



Normally users want to change the Connection Type to the plane for a specific item so the property required is DYN_THERMAL_CON_TYPE. The available options for value are shown above but are based on the standard shape parameter options. You have the ability to modify any of the settings from the shape parameters settings above. For a full list of available properties that can be applied please look at <your_install_dir> (C:\Cadence\SPB_XX)\doc\propref\propref.pdf.

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134
Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.