Padstack Designer

Dialog Boxes | Procedures

The Padstack Designer lets you create and edit padstacks and save them to your design, to a library, or to both at once. You define the pad size and shape for all etch/conductor and non-etch/conductor mask layers in the Padstack Editor.

Default routing layers are BEGIN layer, DEFAULT INTERNAL, and END layer. The DEFAULT INTERNAL padstack definition is used by default when you add more layers in your design. When the padstack is placed in the footprint, the BEGIN layer is mapped to the TOP substrate layer, and the END layer is mapped to the BOTTOM substrate layer.

Non-etch/conductor mask layers include fixed and user-defined mask layers. The fixed mask layers are SOLDERMASK_TOP, SOLDERMASK_BOTTOM (for soldermask artwork) and PASTEMASK_TOP, PASTEMASK_BOTTOM (for solder paste artwork). An extra layer pair named FILMMASK_TOP and FILMMASK_BOTTOM is available for use in whatever means you wish. These two layers are optional and do not have to be used or defined. You can also define up to a maximum of sixteen user-defined mask layers, which you add according to your design flow requirements. This may include, but not be limited to, applications associated with via plugging, hard gold, soft gold, silver, or other soldermask alternatives.

For an overview concerning padstack creation, see the *Defining and Developing Libraries* user guide in your documentation set.

Padstack Designer Dialog Box

The Padstack Designer comprises the *Parameters* tab, the *Layers* tab, and the Menu bar.

Parameters Tab

Click on the *Parameters* tab to view the current padstack type in the Summary group box.

Type

Through	Specifies that the padstack penetrates all layers. This is the default padstack type. You must specify the drill hole parameters for the through padstack type.
Blind/Buried	Specifies that the padstack is blind (navigates the surface and internal layers) or buried (traverses internal layers). When creating a blind or buried padstack, layers to be excluded must have their pads removed. Pad information must exist on a given layer to be considered drilled to that layer. The criterion for connectivity is the existence of a pad definition on the subclass. The connectivity model for any padstack reads all/any type pad information defined in the padstack, and both the model and the add connect command interpret the existence of a pad of any type (thermal, antipad or regular) as a legal layer for a connection. When an outer subclass is missing a regular pad, it is still considered a thru drill when it contains some pad information. If all pad data is missing, then the layer is not considered drilled, and it is a candidate for blind/buried via status. No connection is available on layers with no pad definitions.
Microvia	Enabling this option classifies a blind or buried via as a microvia, defined by the IPC - Association Connecting Electronics Industries as less than or equal to 0.15 mm or 5.91 mils in diameter, with a pad diameter less than 0.35 mm or 13.8 mils formed by laser or mechanical drilling, wet/dry etching, photo

imaging, or conductive ink-formation followed by a plating operation.

Note: In Allegro PCB Design L, OrCAD, and Allegro PCB Performance option L, a padstack remains classified as a microvia, but any action based upon this padstack type is ignored, as `pad_designer` is an unlicensed tool. DRC treats microvias as blind and buried vias and uses blind and buried rules for reporting, possibly resulting in more DRCs if the blind and buried rules spacing requirements are larger than those for microvias. If a design with microvia DRCs is opened in a non-microvia tier, those DRCs are maintained and reported as microvia DRCs. Any operations that affect those DRC'ed objects (for example. `drcupdate`) deletes these DRC and possibly replaces them with a blind and buried via DRC. This DRC model follows the current electrical DRC tiering model.

Single Specifies that the padstack is for only one layer. You usually use this option for SMT pads.

Mask Refers to a padstack with no drill hole defined and pads defined only on mask layers.

Undefined Specifies that the padstack has no pads (etch/conductor or mask) or drill hole defined.

Units

All of the fields that use measurement refer to this value. You should choose the units and accuracy to suit your design environment, but remember that these units are converted to board units and accuracy, which may result in rounding when you apply the units to a design.

Units Specifies the unit of measurement for the padstack. Choices are Mils, Inch, Millimeter, Centimeter, and Micron.

Decimal Places Specifies the precision of the units of measurement. Choose a number from 0 through 4, which sets the number of decimal places.

Usage Options

Microvia Enables the padstack to be used for HDI features, including advanced DRC. Use on HDI boards where spacing rule sets differ between HDI and conventional blind/buried or core vias. Only applicable to a blind/buried via type, which should be used to represent mechanical drilled vias, such as the core via on an HDI design. The two via types are supported in the Spacing and Same Net Spacing Constraint domains of Constraint Manager.

Suppress unconnected int. pads;legacy artwork

Enable to suppress unused inner layer pads, thereby making them unavailable for DRC, routing, and display, with the *Suppress Unconnected Pads* option on the *Film Control* tab of the Artwork Control Form, available by choosing *Manufacturing* - *Artwork* (`film param` command) or with dynamic unused pad suppression, available by choosing *Setup* - *Unused Pad Suppression* (`unused pads` command). Otherwise, you cannot suppress any pads during photoplotting.

Mech pins use antipads as Route Keepouts;ARK

Enable to use antipads defined on ETCH/CONDUCTOR layers and associated with mechanical pins as an implicit route keepout area. This may reduce the need to draw keepout shapes associated with the mechanical symbol. Use in combination with design-level mechanical hole DRC.

Multiple Drill

Choosing a *Hole Type* of *Oval Slot* or *Rectangle Slot* for non-circular slot holes disables this section, as multiple slot holes are not supported in a padstack.

Enabled Choose to set the number of drill holes in a multi or plural via padstack by entering the number of rows and columns. Total drill holes cannot exceed 50.

Staggered Choose to offset the first hole of the even-numbered row in a multiple drill pattern, by half the hole *Size X* value plus half the *Clearance X* value. The bottom row constitutes row 1of a multiple drill pattern.

Rows Specifies the number of rows in the multiple drill pattern. Acceptable values are 1 through 10.

Columns Specifies the number of columns in the multiple drill pattern. Acceptable values are 1 through 10.

Clearance X Specifies the spacing between columns of holes for a multiple drill array. The hole shown in the padstack graphic reflects any clearance differences. Negative values are supported if via overlapping is required. Set the value to zero to support via tangency.

Clearance Y Specifies the spacing between rows of holes for a multiple drill array. The hole shown in the padstack graphic reflects any clearance differences. Negative values are supported if via overlapping is required. Set the value to zero to support via tangency.

Drill/Slot Hole

Specifies your drill hole and drill symbol information before generating drill drawings. The drill figure and characters do not display when a padstack initially loads to a design.

Hole Type Choose a hole type of *Circle Drill* for circular holes. Choose *Oval Slot* or *Rectangle Slot* for non-circular slot holes.

Plating Specifies the type of plating: *Plated, Non-Plated,* and *Optional.* Defaults to *Plated*.

Drill Diameter Specifies the hole drill size in the padstack's unit of measurement and appears only if you chose a *Hole Type* of *Circle Drill* and used `padeditdb` or `padeditlib` to launch Padstack Designer.

Slot Size X Specifies the hole drill size in the padstack's unit of measurement if you chose *Oval* or *Rectangle Slot* as a *Hole Type*.

Slot Size Y Specifies the hole drill size in the padstack's unit of measurement if you chose *Oval* or *Rectangle Slot* as a *Hole Type.*

Tolerance +/- Specifies positive and negative tolerance for each padstack hole size, which then appear in the Drill Legend output. as separate drill sizes and in NC Drill output as separate drill sizes and tool bits.

The Drill Legend displays these tolerance values in one column, as `+<value>/-<value>`. Each NC Drill output file header, where the drill tools and sizes display, contains any defined positive and negative tolerances.

Offset X, Offset Y Specifies the x and y coordinates that indicate the distance the drill hole shifts from the pad center (0,0). The drill hole should not be offset for electrically connected pins, and any offsets should be applied to the pads, and not away from the drill so that no copper exists for connectivity.

The connect point of the pin is derived from its origin, and pin origin is the location that the connectivity model uses to determine if nets are connected or

not. Pad data shifted or deleted has no bearing on connectivity. A thermal, anti pad, or regular pad indicates a pad exists at that location on that layer. The pin loads a padstack and then applies offsets based on the padstack definition. The origin of the pin remains unchanged. Shape voiding also uses the pin origin for connections.

Non-standard Drill Choose a non-standard drill manufacturing method of *Laser, Plasma, Punch, Wet/dry etching, Photo Imaging, Conductive Ink Formation*, or *Other*. The default selection of <blank> indicates standard NC mechanical spindle-type drilling of the hole.

Circle Drill holes manufactured with a non-standard drill are named `<design name>-<type><n>.drl`, where `<type>` is `laser, plasma, punch, etch, photo, condink, or other` in the NC Drill output filename. For example, `ncdrill-laser1.drl`).

Drill/Slot Symbol

Figure Specifies the geometric shape that identifies each hole size (Null, rectangle, square, circle, octagon, cross, diamond, oblong, hexagon X, hexagon Y, or triangle) if you chose *Circle Drill* in the *Hole Type* field. Otherwise, this field is read only and set by Padstack Designer to *Oblong X, Oblong Y, or Rectangle,* depending on whether you specified *Oval Slot* or *Rectangle Slot* in the *Hole Type* field. You must specify the same figure for all holes of the same size and plating type.

Characters Specifies up to three optional, printable characters to define a drill size and its respective symbol codes. The characters' height fits the given width and height of this symbol.

NOTE: The combination of the drill figure and the optional characters must be unique for the specified drill size and hole plating. The drill figure and characters do not display when a padstack initially loads into a design.

For example, all padstacks with 39 mil holes might have a square drill figure with a drill character of A. All padstacks with 25 mil holes could then have a square figure with a character of B or they could have a triangular figure with a character of A. Either of these combinations would differentiate the 25 mil hole from the 39 mil hole on a drill drawing.

Width Specifies the width of the drill figure in the padstack unit of measurement, if you chose a *Hole Type* of *Circle Drill.* Otherwise, the field is read only, and its value defaults from the *Slot Size X/Y* field automatically.

Height Specifies the height of the drill figure. In the unit of measurement that you set for the padstack if you chose a *Hole Type* of *Circle Drill.* Otherwise, the field is read only, and its value defaults from the *Slot Size X/Y* field automatically.

The width and height for circles and squares are automatically the same. For example, if you enter 5 in this field, 5 appears in the width field.

Top View Displays the padstack extents, drill hole size, and the drill hole offset, or a top down view of the padstack. The drill hole is gray, and the pads appear in different colors to highlight overlapping pads.

Layers Tab

This tab defines both the number of layers in the padstack and the individual layer type. When you define a padstack for your library, you only see the layers that you define in the table, but when you open a padstack from the design, you see all of the layers that are related to that

padstack.

The top section displays all padstack layers with the corresponding *Regular Pad*, *Thermal Relief*, and *Anti Pad* definitions. A tabular format represents padstack layers with each row representing a single pad layer. For library padstacks, you can change the padstack layer name in the Layer column of any highlighted layer. For layout padstacks, the layer name is not editable in the Padstack Designer. To edit a layer name in a layout padstack, you must use the Layout Cross-section.You can edit any of the white fields and Insert, Delete, Copy, and Paste layer definitions.

You do not need to fill in all of the options to save the padstack. To save a padstack file, you must define a pad for at least one layer.

You may define a default pad set to be used on internal layers for DRC (design rule checking) of line-to-pad spacing. The INTERNAL_LAYER pad layer allows you to define a single pad layer set that maps to all internal layers when the padstack is loaded into the board. Layers defined specifically by name override the INTERNAL_LAYER pad during board mapping. The pad definition triggers DRC.

If no pad definitions exist on internal layers, this particular check does not occur. This may cause etch/conductor to be placed in the path of a drill hole without DRC flagging it. You define the pad size smaller than the drill hole so the pad will be drilled out during manufacturing. As long as a pad is defined, DRC uses the larger of the drill hole or pad to check spacing. Do not specify Null pad definitions on internal layers of through-hole padstacks because those definitions cannot be connected.

For multiple drill padstacks (multi or plural vias), the overall array of the drill holes must fit within all of the individual pads.

Do not use pad sizes equal to the hole size. Drilling is never perfectly centered, and off-center holes leave crescents of etch/conductor that can detach and move, causing shorts.

The bottom section displays the pad definition areas for *Regular Pad*, *Thermal Relief*, and *Anti Pad* for all conductor/etch layers and the Regular Pad definition for mask layers. A pad may be used on a routing layer or it may be used on a plane layer. For planes, based upon your design environment, the pad may be used on a negative plane or on a positive plane. Therefore, it is usually best to define all of the regular, thermal and anti-pad definitions for the Begin Layer, Default Internal and End Layer when creating the initial padstack. For each of these definitions, you must define the pad shape as circle, square rectangle, oblong, octagon, or shape. A shape (polygon) is used for any definition that is not a circle, a square, a rectangle, oblong or an octagon. A Shape symbol for the geometry of the pad must be created manually using the Symbol Editor.

Positive Etch/Conductor Layer Pad Requirements

Padstacks requirements differ depending on whether artwork is positive or negative. Positive etch/conductor layers primarily use the regular pad. Pad shape symbols must be generated and used for positive regular pads when you require complex pad geometries. This solid shape displays, and DRC runs against the actual geometry. Pad shapes are limited to one solid shape, typically used for:

- SMD pads where corners must be chamfered,
- octagonal shaped pads when using versions prior to 15.5, or
- in a unique situation where pad boundary editing changes a pad geometry.

Flash names may also be used as regular pads. Pad flashes are used primarily as negative

thermal reliefs and in limited applications when defined in the regular pad. (Prior to the support of pad shapes, pad flashes were used to generate irregular pad geometries in artwork.) Regular pads cannot contain voids and are limited to a single geometry. A flash is not limited to one outline and can contain multiple shapes, which may solve requirements for multi-shape pads on soldermasks or outer-layer footprints.

When using a flash, the display, DRC, and connectivity use the regular pad geometry and extents. The flash is a label. Ensure you specify a geometry and size that accommodates the size of the desired flash aperture. A multi-shape flash might not contain copper at the connect point (pin 0,0 location), and the pin may not physically connect if the pin center is not matched to copper in the flash. This application requires careful review of the connectivity in the board and in the artwork output.

In regards to pad flashes and artwork in positive data, if a regular pad is used, raster artwork searches PSMPATH for flash geometry (`.fsm`) file data. The flash name supersedes the geometry definition in the padstack if you chose a flash name in the padstack. The shape data in the flash symbol is embedded in the output artwork file, and displays when you run *File - Import - Artwork* (`load photoplot` command). Vector artwork, however, searches for a matching flash name in the aperture list. It does not use the library flash symbol. Running *File - Import - Artwork*, where vector artwork contains flashes in the aperture file, searches ARTPATH for the flash geometry (`.fsm`) file data. If found, the etch/conductor geometry displays; otherwise, a triangle with the flash name displays as an indicator of an aperture flash. The actual flash geometry must be provided with the aperture file to the vendor generating the artwork.

Negative Etch/Conductor Layer Pad Requirements

Negative layers use shapes for connectivity. Based on the pad information from the padstack's thermal relief and anti-pads definition, the shapes are drawn on the etch/conductor subclass, assigned to a net, and flash the thermal pad shape. Negative planes should contain shapes that reference, display, and DRC check to the thermal relief and anti-pads geometries defined in padstacks. With no expectation of autovoiding, the intent is to quickly generate power and ground planes without processing all DRC rules, and instead create clearances-based padstack data.

To visually determine the sizes of thermal and antipad that artwork flashes or embeds as apertures (depending on artwork output type), enable *Thermal Pads* on the *Display* tab of the *Design Parameter Editor*, available by choosing *Setup - Design Parameters* (`prmed` command).

Older databases using the older pad flash process, in which the variable `old_style_flash_symbols` is enabled in the *Misc* category of the User Preferences Editor, available by choosing *Setup - User Preferences* (`enved` command), draw a thermal pad using the geometry and extents of the thermal pad listed in the padstack. The flash name is used for artwork, and its geometry is not displayed. A crosshair centered in that pad indicates connectivity. New databases using this old style methodology display this geometry and not the information in the flash symbol itself.

New databases (in which the `old_style_flash_symbols` variable is disabled) draw the actual flash symbol `.fsm` geometry when you enable *Thermal Pads*, using true display of flashes and older data that has been updated to exploit this feature. See the `flash_convert` utility.

When the negative planes setup is correct and contains flash names for thermal pads, artwork output will not differ between the old and new style pad flashes. Old and new style flashes require a flash symbol for raster artwork to embed in the output file. The flash is either in the `.brd` file for new style flashes, or in the PSMPATH library path for old style flash data.

Artwork fails when no flash appears in the PSMPATH library path, and the following error appears in the `photoplot.log`:

```
ERROR: aborting film - undefined aperture symbol - cannot continue

Load photoplot will read and display the raster artwork flashes.
```

Vector artwork relies on the aperture file to match pad flashes. Therefore, old and new style flash data write identical artwork output files. When you run *File - Import - Artwork*, (`load photoplot` command) PSMPATH is searched for flash symbols. If found, the flash draws that geometry; if not, a triangle figure is placed as the flash indicator and text containing the flash name inserted.

Negative pads in artwork

If thermal flash names are not defined in padstacks, warnings and errors occur if flash names are missing from the thermal relief when processing negative layers. Artwork expects a pad flash in this case. Without a flash name, raster artwork uses the regular pad size and issues this warning in the `photoplot.log`:

```
WARNING: No thermal flash for padstack "PADNAME", regular pad used.
```

Without a flash name, vector artwork reports this as an error:

```
PADSTACKS MISSING THERMAL/ANTIPAD DEFINITIONS:

VIA

     PAD60CIR36D
```

In both cases, Cadence advises adding a thermal flash name to the padstack.

All other pads on a negative layer flash the antipad. Positive shapes create voids based on the thermal and antipad sizes if DRC is not selected as the voiding rule. The true intent of these pads is for negative layer flashes. Therefore, Cadence does not recommend using thermal and anti pads for positive shape voiding but rather DRC rules. The thermal relief can use any standard geometry, or point to type Flash, where a library flash symbol representing the true thermal geometry can be assigned.

This can be used in old style flash symbol mode where raster artwork uses the geometry in the artwork file, but the display contains just cross hair indicators for thermal connectivity. In new style flash symbol mode, the display, negative plane DRC, and the artwork file integrate the true flash geometry in all areas. Flash symbols allow multiple shapes, but no voids. These will not flash on positive layers, nor display on positive layers.

Non-plated holes can be defined with small spotting pads that are drilled away during fabrication, or the pad can be defined as null, meaning no pad will be drawn. Pads add benefit when dimensioning or using tools to locate the pins for viewing or checking. Adding adequate route keepout areas around these small pads prevents etch/conductor clearance issues. Dynamic shapes void around these large areas. Using keepout areas for DRC clearances on non-plated holes is recommended in conjunction with voiding to DRC rules.

Views Displays the padstack in cross-section or as a top view.

Padstack layers

Layer Displays all the layers of your padstack, with the corresponding *Regular Pad*, *Thermal Relief* and *Anti Pad* definitions. You can edit any of the white fields. Bgn (Begin) and End layers are the outer layer pads: in a `.brd` file, the Top and Bottom; in an `.mcm` file, Surface and Base. Default internal is used for through-hole inner layer pad data.

Regular Specifies a positive pad with a regular shape (circle, square, rectangle, oblong, octagon), flashed on positive layers only. Through hole pads require regular pad geometries be defined for every board layer. Surface mount pads require only Top and related Top mask information. Non-standard shaped pads are available as pad shapes and as pad flashes. If you are using your own custom shape, choose shape, which is used for any definition that is not a circle, a square, a rectangle, oblong or an octagon. A Shape symbol for the geometry of the pad must be created manually using the Symbol Editor.

Thermal relief Specifies either a positive or a negative pad. A positive thermal relief connects pins to a positive copper area or to an embedded plane that prevents heat from concentrating near a pin or via during the unsoldering process. The thermal relief definition also may connect a pad to a copper area created on a routing layer, such as an external shield.

A positive plane thermal relief comprises a "spoked wheel" pattern, which combines the regular positive pad, void areas, and tie bars, the latter two of which are defined in the Shape Parameters dialog box.

A negative thermal relief is defined by a flash symbol used to connect pins to a negative copper area. The thermal relief may be plotted as a regular pad flash combined with the thermal values used as the shape-to-pad clearance.

Anti-pads Specifies a negative pad (clear, surrounded by black), where the size and type determine pad flashes for negative planes, or pad sizes for clearances when specified in positive shape parameters. Used to disconnect pins from a surrounding copper area.

Geometry Specifies the standard shape of the pad. You have a choice of Null (no shape), Circle, Square, Oblong, Rectangle, Octagon, or Shape (custom pad).

Shape Specifies an irregularly shaped (custom) pad created with the Symbol Editor. You must specify the Shape symbol name (`.ssm`) file in the Geometry field.

Flash Specifies the flash name referencing flash symbols displayed on WYSIWYG negative planes, or a user-defined name of an aperture for Gerber flashing of a unique pad shape. Define a flash symbol in the Symbol Editor if you are going to create a negative plane in your design. If you plan to use only positive planes, you do not need to create flash symbols.

Width Specifies the width of the pad if it is a Square, Oblong, Octagon, or Rectangle. If the pad is a Circle, enter the diameter in the unit of measurement set for the padstack.

If you specify a Square, Circle, or Octagon in the Geometry field, the value you enter here automatically sets the Height field to the same value. For Square and Octagon geometries, the specified size defines the horizontal distance between the two vertical sides/facets of the geometry.

Height Specifies the height of the pad if it is a Square, Oblong, Octagon, or Rectangle. If the pad is a Circle, enter the diameter in the unit of measurement set for the padstack.

If you specify a Square, Circle, or Octagon in the Geometry field, the value you enter here automatically sets the Width field to the same value. For Square and Octagon geometries, the specified size defines the vertical distance between the two horizontal sides/facets of the geometry.

Note: The width and height for circles, squares, and octagons are automatically

the same. For example, if you enter 5 in this field, 5 appears in the width field.

Offset Specifies the *x* and *y* coordinates that indicate how far the pad origin shifts from the pad center (0,0).

Select Padstack Template

This standard library browser lets you choose a padstack template. All objects appear in alphabetical order.

To choose an element, type the name in the search field, or highlight it in the list box. To narrow the list, enter a search string in the search field and click OK. The asterisk (*) displays the complete list. For example, a search string of MTG* returns all objects beginning with MTG. Your last search is remembered.

Menu Commands

The commands on the Padstack Designer menu bar differ depending on whether you started the Padstack Designer as a standalone tool (using the `pad_designer` command) or from within a design (using the `padeditdb` or `padeditlib` command).

File - New Displays the New Padstack dialog box where you can define a new padstack or edit an existing one using the following fields:

Directory: Displays the current working directory.

Padstack Name: Enter a name for the new padstack. A padstack filename can consist of the characters A to Z, 0 through 9, dash (-), and underscore (_).

Browse: Click to choose an existing padstack from the file browser that appears. Doing so overwrites the existing padstack.

Template: Click to create a new padstack based on a template containing default `.pad` definition information. For example, a padstack template might contain user-defined mask layers as mandated by corporate standards. The Select Padstack Template library browser displays, whose contents are derived using the PATH variable WIZARD_TEMPLATE_PATH. Populated by your CAD administrator, the CDS_SITE wizard directory location is CDS_SITE/pcb/templates.

OK: Click to create a new padstack.

Cancel: Closes the New Padstack dialog box without creating a padstack.

If you are currently editing a padstack, you are asked if you want to save it. When you have saved the current padstack, the new file browser appears with the filter set to **.pad*.

File - Update to Design Available only when Padstack Designer is invoked within a design using `padeditdb` or `padeditlib`. Adds the current padstack definition back to the design, if there are no errors. If the padstack is new, it is added to the design. If the padstack already exists in the design, the design padstack is updated with the new information from the Padstack Designer. If there are errors in the padstack, the update fails and the errors are displayed in the Editor's status area.

Note: This command does not save the new definition. Choose *File - Save to File* or *File - Save As* to save the data.

File - Recent Padstacks - Recent File List

Available only when Padstack Designer is invoked as a stand-alone tool. Opens

a list (up to 20) of your most recently used (MRU) padstacks with their specified path names. The default number of padstacks is 10. When you choose a padstack from the list, the editor opens the file and changes the current directory to the directory where the specified padstack resides.

File - Save This command appears in the Padstack Designer when you open the application in standalone mode. Saves the current padstack to your current working directory. If there are errors in your padstack, you are warned about them. If a padstack of the same name already exists in your working directory, you are warned that the padstack file will be overwritten. If you do not want to overwrite the padstack, choose *File - Save As* to save the padstack with a different name.

File - Save As Saves the current padstack definition under a different name. When you choose this option, s the current padstack is checked for errors and the *Save As* browser displays. Enter the new name in the filename field. If a padstack already exists with the same name, you are warned that it will be overwritten. Enter a unique name to avoid overwriting another file.

File - Check Checks the current padstack definition for any potential problems. Any problems are displayed in the Pad Stack Warnings window.

Note: This command does not save the padstack.

File - Save Padstack to 16.01

Available only when Padstack Designer is invoked as a stand-alone tool. Revises a design database containing new functionality, so that you can open the design in an earlier version of the same base release.

File - Properties Lets you set an optional password-protected lock from the File Properties dialog box so the file is marked as read-only in the database. For details, see File Properties Dialog Box in the `file_property` command.

File - Script Available only when Padstack Designer is invoked as a stand-alone tool. Starts the scripting process to record or replay padstack scripts. Recording a script lets you automate the padstack definition process. Replaying a script cuts the time that is involved when you define new padstacks that share similar definitions.

File - Exit Exits the Padstack Designer. You are prompted to save all data and update the design.

Reports - Padstack Summary

Displays a summary of the current padstack data.

Reports - Show Instances

Displays all the pins and vias that use the current padstack.

Reports - Library Drill Report

Contains the drill-related information for all the available library padstacks, whether the padstacks are used in the current design or not. While editing padstacks, you can assess if the hole you are defining in the padstack is already in use elsewhere in the library, and ensure other drill information (such as drill symbols for example) is synchronized. Functions as a reference aid, as information is unrelated to the current design.

Procedures

The following padstack procedures are addressed:

Prerequisites to Defining New Library Padstacks

- Check your manufacturer specification sheets and verify your design requirements.
- Gather the dimensions, physical data, logical data, manufacturing requirements, and documentation requirements for the padstacks that you want to define.
- Check that your padstacks are not already in the library.
- Determine your photoplot requirements, such as flash names, NC Drill data, and offset requirements.
- Create any custom (unique) pad shapes that you need.

Padstack Definition Guidelines

- Avoid specifying Null pad definitions on internal layers of through-hole padstacks.
- Define a pad size for every internal layer so that design rule checking (DRC) checks the line-to-pad spacing. DRCs occur on line-to-pad spacing only if pad definitions exist on internal layers. Therefore, an etch/conductor layer can be placed in the path of a drill hole without being flagged by DRC.
- Define the pad size to be smaller than the drill hole, so that the pad is drilled out during manufacturing, and the DRC can verify the spacing. As long as a pad is defined, the DRC uses the larger of the drill hole or pad to check your spacing.
- Define all layers through which the padstack passes for editing when you define blind/buried padstacks. Because undefined layers do not reveal information about the objects on them, it may appear that data is lost because the layer is undefined.

Note: When creating a blind or buried padstack, layers to be excluded must have their pads removed. Pad information must exist on a given layer to be considered drilled to that layer. The criterion for connectivity is the existence of a pad definition on the subclass. The connectivity model for any padstack reads all/any type pad information defined in the padstack, and both the model and the add connect command interpret the existence of a pad of any type (thermal, antipad or regular) as a legal layer for a connection. When an outer subclass is missing a regular pad, it is still considered a thru drill when it contains some pad

information. If all pad data is missing, then the layer is not considered drilled, and it is a candidate for blind/buried via status. No connection is available on layers with no pad definitions.

Starting Padstack Designer

Padstack Designer can be used independently of other Cadence tools, or can be launched from them.

As a stand-alone tool:

On a PC

Choose *Start - Programs - Cadence - PCB Systems - Padstack Designer*

or

1. Click *Start - Run*
2. In the Open field of the Run dialog box enter the following:

```
pad_designer.exe
```

The Padstack Designer dialog box appears.

On a Unix Workstation

→ From your system prompt, type the full path name to the directory in which your Cadence tools are installed, and invoke the Padstack Designer:

```
pad_designer
```

Defining Padstacks

Whether you are defining a padstack for your design, your library, or both, you must set both the padstack parameters and the pad definitions for your padstack layers.

Defining Parameters

As you define the padstack layer parameters, the Top View shows the extents of your data. The layers are in different colors to show any overlap, and the drill hole is gray.

1. Select *File - New* from the Padstack Designer main menu.

The New Padstack dialog box displays.

2. Enter a name for the new padstack. A padstack filename can consist of the characters A to Z, 0 through 9, dash (-), and underscore (_).

-or-
Click *Browse* to choose an existing padstack from the file browser that appears and click OK.

-or-
Click *Template* to create a new padstack based on a template containing default `.pad` definition information and click OK.

3. Click on the *Parameters* tab in the Padstack Designer.
4. Note that the current padstack type displays in the *Summary* group box. It will be one of *Through, Blind/Buried, Microvia, Single, Mask*, or *Undefined*. The Padstack Designer software continuously evaluates and sets the padstack type based upon the current padstack geometries. When you begin editing a new padstack, the type will initially be Undefined, indicating that there are no pads or drill hole defined in the padstack.
5. Set the *Allow Suppression of Unconnected Internal Pads* checkbox in the *Usage Options* group box to control how unconnected pads on internal layers are handled during photoplotting

6. Choose the units for the padstack from the *Units* list box. These units apply to any measurements in the padstack definition.
7. Enter a number from 0 to 4 in the *Decimal Places* to indicate the units precision.
8. Provide the *Drill/Slot Hole* information to generate drill drawings.
9. Specify the *Drill/Slot Symbol* information that you want to appear in the NC drill legend.

Note: The drill figure and characters do not display when a padstack is initially loaded into a design.

The combination of the drill figure and the optional characters must be unique for the specified drill size and hole plating. For example, if all padstacks with 39-mil holes have a square drill figure with a drill character of A, all padstacks with 25-mil holes can have a square figure with a character of B, or they can have a triangular figure with a character of A. You can use either of these combinations to differentiate the 25-mil hole from the 39-mil hole on a drill drawing.

Defining Pads for a Layer

1. Click on the *Layers* tab.
2. Click each layer that you want to define. For example, if you are defining pads for the top/surface or Begin Layer, highlight that row in the *Layers* table.
3. In the lower section of the *Layers* tab, specify the information for each pad type (REGULAR, THERMAL RELIEF, and ANTI-PAD) for the selected layer.
4. Repeat steps 2 and 3 for each layer of pads that you need to define.

Inserting Layers in a Padstack

When you add a new layer to a padstack, the Padstack Designer adds that layer above the layer you are working on when you start the process. Note that you can add layers in a library padstack, but not in a layout padstack.

1. Start the Padstack Designer.
2. Choose *File - New* from the Padstack Designer main menu.

The New Padstack dialog box displays.

3. Click *Browse* to choose an existing padstack to edit from the file browser that appears.
4. On the *Layers* tab of the Padstack Designer, right-click next to the layer above. Choose Insert from the pop-up menu.

A new layer field (UNNAMED1) opens above the layer that you are working on. The text defaults to uppercase in the Layer field regardless of how you typed the name.

5. In the *Layer* column, highlight UNNAMED 1, enter the name of the new layer, and press Enter.
6. Enter the pad information for the new layer by either:

→ Copying pad data from another layer

→ Defining the layer and padstack data in the layer definition fields

Copying Pad Definitions to Other Layers

Note: When you copy pad definitions, only the pad data is copied and not the layer name.

1. On the *Layers* tab of the Padstack Designer, right-click next to the layer from which you want to copy data.
2. Choose *Copy* from the pop-up menu.
3. Right-click on the next to the layer to which you want to copy data.

4. Choose Paste from the pop-up menu.

The data that you have copied is applied to this layer.

Deleting Layers in a Padstack

1. On the *Layers* tab of the Padstack Designer, right-click next to the layer that you want to delete.
2. Choose Delete from the pop-up menu.

The layer is deleted.

Single Layer Mode

When you select the *Single Layer Mode* checkbox, only the first conductor/etch layer with pads defined on it will be displayed. If no pads have yet been defined, the Top layer displays. To view all ETCH/CONDUCTOR layers, deselect the checkbox.

Adding Custom Pad Shapes to a Padstack

When you create a new padstack or edit an existing one, you can add custom pad shapes to any of your definitions. Make sure that you have defined the shapes prior to adding them to your padstack.

1. Start the Padstack Designer.
2. Select *File* - *New* from the Padstack Designer main menu.

The New Padstack dialog box displays.

3. Click *Browse* to choose an existing padstack from the file browser that appears and click OK.
4. Click the *Layers* tab to see the layer information for the padstack.
5. Click the layer to which you want to add your custom pad shape to (for example, BEGIN LAYER).

The layer information displays at the bottom of the screen.

6. In the *Geometry* field of the pad definition, choose *Shape*.
7. In the *Shape* field, enter the name of the custom pad shape symbol.

If you do not know the name of the custom pad, click the button to the right of the Shape field to display a browser. The browser has check boxes that allow you to choose lists of library or design padstacks. Choose the custom pad shape from your library or, if you are editing a design padstack, from the design.

The width and height of the pad shape symbol display automatically in the *Width* and *Height* fields for the pad type.

8. Choose *File* - *Save* from the menu bar to save the padstack.

The custom pad shape is now the geometry for that padstack layer.

9. Choose *File* - *Exit* to close the Padstack Designer.

Adding Flash Shapes to a Padstack

When you create a new padstack or edit an existing one, you can add flash shapes to any of your definitions. Make sure that you have defined the shapes prior to adding them to your padstack.

1. Start the Padstack Designer.
2. Choose *File* - *New* from the Padstack Designer main menu.

The New Padstack dialog box displays.

3. Click *Browse* to choose an existing padstack to edit from the file browser that appears.
4. Click the *Layers* tab to see the layer information for the padstack.
5. Click the layer to which you want to add the custom pad shape (for example, BEGIN LAYER).

The layer information displays at the bottom of the screen.

6. In the *Flash* field, enter the name of the flash shape that you want to use.
7. Choose *File - Save* from the menu bar to save the padstack.
8. Choose *File - Exit* to close the Padstack Designer.

Checking Padstack Definitions

You can check the validity of any padstack as you define or edit it.

1. Start the Padstack Designer and define a new padstack or modify a padstack.
2. Choose *File - Check* from the Padstack Designer menu bar.

The Pad Stack Warnings window appears with the list of potential problems that are associated with your padstack definition.

3. To close the Pad Stack Warnings window, click *Close* on the window menu.
4. Correct the padstack definition if necessary before saving the padstack.

Creating a Padstack List File

Padstack list files are ASCII text files that contain the names of padstacks that can be updated using `refresh padstack`.

1. Use a text editor to create the padstack list file. The default file extension is `.lst`.

The following file format conventions apply:

- Provide only one padstack name on each line.
- You can use either uppercase or lowercase letters.
- any leading or trailing white space IS REMOVED.

2. Save the file in the current working directory.

The following example shows the format of the padstack list file:

```
pad93cir58d

soj

via

smd25-94
```

Creating and Saving Library Padstacks

→ To save a padstack to a library, choose *File - Save As* from the Padstack Designer and use the file browser to specify the name of the new padstack definition.

If you have edited an existing padstack and want to save the changes to the same, choose *File - Save* (if Padstack Designer is running in standalone mode) or *File - Save to File* (if Padstack Designer is running within your tool user interface) to update the previous padstack definition with the new one.

The padstack definition is checked before saving it to a library file. If any problems occur, the Pad Stack Warnings window appears. You can save the padstack with or without warnings. However, saving a padstack with warnings may result in post-processing errors or problems when you use the padstack.

Important When saving a padstack with the Single Layer Mode checkbox selected, all conductor/etch pads on layers other than the single layer displayed will be deleted from the padstack before saving and cannot be recovered. In this case, a confirmer is issued informing you of this condition and allows you to cancel the save operation.

Displaying Derived Padstack Names

You can use the following methods to display the derived padstack names in your design:

Method 1

1. Run `padeditdb` in your tool user interface.

The *Options* tab reconfigures the parameters.

2. Click *Derived*.

The derived padstacks in your design appear in the Available Padstacks listing.

Method 2

→ Use the Constraint Manager.

Modifying Padstacks in a Library

1. Run `padeditlib`.
2. Choose the library padstack that you want to modify from the file browser and click OK.

The Padstack Designer dialog box appears with the padstack definition loaded. The banner of the Padstack Designer lists the name of the padstack that you are modifying.

3. Edit the padstack parameters and layers as appropriate.

→ To save the padstack to the library, choose *File* - *Save* to File to overwrite the current padstack definition with the newly modified one, or choose *File* - *Save As* to specify a new padstack name.

→ To load the padstack into your design, choose *File* - *Update To Design*. This option is available only if you invoke the Padstack Editor from your tool's user interface instead of using the Padstack Editor as a standalone program. For more information, see Modifying Padstacks in the Design, below.

Modifying Padstacks in the Design

You can edit both the padstack definitions and the padstack instances in the design that you are currently working on.

1. Run `padeditdb`.
2. Click on the *Options* tab of the Control Panel.
3. Click *Definition*.
4. Choose the padstack definition that you want to edit from the list in the *Options* tab.

The padstack information appears in *Name* field.

5. Click *Edit.* You can also choose Edit by right-clicking in the design window and selecting it from a pop-up menu.

The padstack designer opens with the padstack definition loaded.

6. Modify the padstack parameters and pad layers as necessary.
7. Choose *File* - *Update* to Design from the Padstack Designer to update all of the padstacks in the design that have the current name. DRC executes on all of the changes to your padstacks in the layout. All of the padstacks in the design refresh unless errors exist in the

padstack definition, in which case, a warning message appears and you can correct the errors in the Padstack Designer. If the padstack exists in the design (that is, you have not renamed the padstack), you are prompted to overwrite the existing padstack.

You can also choose *File* - *Save* from the Padstack Designer to save your padstack data to a library.

Editing Padstack Instances in your current design

1. Run padeditdb.
2. Click on the *Options* tab of the Control Panel.
3. Click *Instance*.
4. Specify a symbol name, pin name, or reference designator in the appropriate fields. You can also click on the padstack instance in the design window that you want to modify. For example:
   - To edit instances of pins of a part symbol, that reference the same padstack, type the symbol name in the Symbol field and the pin number specification in the Pin field.
   - To edit a specific pin number on a specific reference designator, type the pin number in the Pin field and the reference designator in the Ref Des field.

   For example, to edit pin 1 on Z18, enter 1 in the Pin field and Z18 in the Ref Des field.

Notes:

If you specify an asterisk (*) in the field, then all pins of the specified symbol (or refdes) get the new padstack.

If you specify an asterisk (*) in the Symbol field, the Ref Des field automatically changes to an asterisk (*) and all of the pins that you specify in the Pins field get the new padstack.

IF you specify an asterisk (*) in the *Ref Des* field, the *Symbol* field automatically changes to an asterisk (*), and all of the pins that you specify in the Pins field get the new padstack.

If you are modifying a design padstack in the Symbol Editor, you do not see the Ref Des or Symbol fields in the *Options* tab.

5. Click *Edit.* You can also choose Edit by right-clicking in the design window and selecting it from a pop-up menu. The padstack designer opens with the padstack instance loaded. The banner on the Padstack Designer says

```
"Editing Design Instance <name-".
```

The tool finds the pins that match the information in the Symbol, Pin, and Ref Des boxes, and then finds the padstack name that is used by those pins. If the combination specifies pins that have more than one padstack name among them, the following error message displays:

```
E - More than one pad stack specified

Enter a different combination and click OK again.
```

6. Modify the padstack parameters and pad layers as appropriate.
7. Choose *File* - *Update to Design* from the Padstack Designer to update the padstacks of all the pins in the design that were specified. DRC executes on all of the changes to the padstacks in the layout. Padstacks in the design refresh unless errors exist in the padstack definition, in which case, a warning message appears, and you can correct the errors in the Padstack Designer.

You can also choose *File* - *Save* from the Padstack Designer to save your padstack data to a library.

Note: A derived padstack name is automatically generated if there are pins referencing the padstack that were not included in the selection set. The derived padstack name is displayed in the new name field. You can edit this name.

Purging Unused Padstacks

You can remove unused padstacks from the list of available padstacks for your design. Purging unused padstacks increases the performance of the editor because the program must load all padstacks, used or unused, load in a design.

1. Run padeditdb.
2. Choose the *Options* tab on the ministatus.
3. Click *Purge* and then choose one of the following options:
   a. Click All.

   If your design contains any unused padstacks, the Unused Padstacks window lists the unused padstacks. A window appears, asking if you want to purge the unused padstacks.
   b. Choose one of the following:

   Yes to purge the unused padstacks from the list of available padstacks and close the window.

   No to leave the unused padstacks in the list of available padstacks and close the window.

To delete all derived padstacks

   a. Click Derived.

   If your design contains any unused derived padstacks, the Unused Padstacks window lists the unused derived padstacks. An Error window appears, asking if you want to purge the unused derived padstacks.
   b. Choose one of the following:

   Yes to purge the unused derived padstacks from the list of available padstacks and close the window.

   No to leave the unused derived padstacks in the list of available padstacks and close the window.

   If your design does not contain any unused padstacks, the Padstack Selection dialog box displays a message with this information.

Recording/Replaying Padstack Scripts

You can automate the process of entering padstack data by creating a script that lets you record the entries that you make on the Padstack Designer dialog box. When you want to define new padstacks that share similar padstack specifications, you can replay the script file and edit the new padstacks as necessary. See script for more information.

Updating Library Padstacks

1. Run refresh padstack.

   The *Refresh Padstac*k dialog box appears.
2. Choose the padstack types that you want to update with the latest library files.

Note: Note: You must enter the name of a list file if you want to update padstacks from a list.

3. Review the *refresh_padstack.log* file.

   This log file is generated in the current working directory each time you update your padstacks.

Viewing Padstack Details

As you work in the Padstack Designer, you have the ability to view information about the padstack that you are currently working on.

1. Choose *Reports* - *Padstack Summary* from the Padstack Designer menu bar.

   A window appears displaying the padstack information.
2. Choose *File* - *Save* to save the summary to disk.

   The file is saved with the default name of *pad_summary.txt*.
3. Click *Close* to close the window.

Viewing Padstack Instances

When you edit layout padstacks, and you start the Padstack Designer from within your tool user interface using `padeditdb`, you can view a report that shows all of the pins and vias that currently use the padstack you are editing.

1. Run `padeditdb` to choose a design padstack.
2. Click *Edit* in the Options tab of the Control Panel.

   The Padstack Designer appears.
3. Choose *Reports* - *Show Instances*.

   A window displays all the pins and vias in your design that currently use the padstack that you are editing.
4. Choose *File* - *Save* to save the listing with the default name of *pin_instances.txt*.
5. Click *Close* to close the window.

[<< package_height]

[pad_designer >>]