PADS® Layout User's Guide and Reference Manual

PADS 9.4

Contractor/manufacturer is:
Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com
SupportNet: supportnet.mentor.com/
Send Feedback on Documentation: supportnet.mentor.com/doc_feedback_form

Table of Contents

What’s New

Want to know what’s new in PADS 9.4?

- See the details of new enhancements, features, and functionality in the Release Highlights on SupportNet:

  English

  http://supportnet.mentor.com/docs/201201028/release_docs/PADS_rh.pdf

  Japanese

  http://supportnet.mentor.com/docs/201201028/release_docs/PADS_rh_ja.pdf

- For a list of defect fixes, see the Release Notes:

  http://supportnet.mentor.com/docs/201201028/release_docs/PADS_rn.pdf

List of Figures

List of Tables

Chapter 1
PADS Layout QuickStart

Welcome to PADS Layout, a powerful yet easy to use layout application.

To get up and running quickly, perform the following steps:

Step 1 - Create a Board Outline

Board outlines are created using the drafting tools. The drafting tools are located in the Drafting toolbox.

To create a board outline:

1. In the Welcome screen, click **Start a new design**.
2. Click the **Drafting Toolbar** button.
3. On the Drafting toolbar, click the **Board Outline and Cut Out** button.
4. Right-click and select your shape preference.
5. Draw an outline using the pointer, making a series of clicks.
6. Click to complete a rectangular or circular outline. Double-click to complete a polygonal outline.

Step 2 - Import a Netlist and Disperse Parts

Typically, net and part design data originate in a schematic capture application and are transferred to a layout application via an ASCII netlist. Use the Import command to import ASCII files into your design.

To import a netlist:

1. On the File menu, click **Import**.

   Tip: Save the design when you are prompted to.
2. In the File Import dialog box, click the file type to import from the Files of type list.
3. Navigate to the file to import, and click **Open** to import the file.
4. Review any errors or warnings, in the *ascii.err* file, that may result from the import process.

After you import nets and parts into your design, disperse the parts around the outside of the board using the Disperse components command.

To disperse parts:

1. On the Tools menu, click **Disperse Components**.
2. Click **Yes** to confirm the dispersion of the parts.

Step 3 - Setup Design Rules

Setup all design rules, including clearance, routing, high-speed and others, within the design rule dialog boxes.

To setup design rules:

1. **Setup menu** > **Design Rules**.
2. Click the **Default** button.

   Result: The Default Rules dialog box appears. Use this dialog box to select the type of default rule to enter. See the table below for more information.
3. Click the **Clearance** button to enter default clearance rules.
4. Click in any one of the boxes in the matrix to change its value.
5. Click **OK** to apply the clearance rule changes.
6. Click any other button to set default rules for the associated type.
7. Click **Close** to close the Default Rules dialog box.
8. Click **Close** to close the Rules dialog box.

Table 1-1 lists and describes the rules types:

Table 1-1. Design Rules Types

Rules Type	Includes rules or preferences for
Clearance	Minimum clearance, minimum same net clearance, trace width, drill-to-drill and part body-to-part body minimum clearances.
Routing	Trace topology, copper sharing, layer bias, via bias, and trace protection level.
High Speed	Parallelism, shielding, and thresholds for trace length, delay capacitance, and impedance.
Fanout	Fanout construction (used in PADS Router only).
Pad Entry	Trace patterns at pad entries and exits (used in PADS Router only).
Conditional	Overriding unique conditions when specific design objects are adjacent to other specific design objects. The rules include minimum clearances and parallelism.
Differential Pairs	Net length and topology of differential pairs.

Step 4 - Set Grids

Several grid settings are available in PADS Layout. Set grids in the Options dialog box or using the grid modeless command.

To set a design grid:

- Type the **g** modeless command followed by the grid values, and press **Enter**.

Example: Type g50 and press Enter to set a .050” design grid.

To set a via grid:

- Type the **gv** modeless command followed by the grid value, and press **Enter**.

Example: Type gv50 and press Enter to set a .050” via grid.

To set a display grid:

- Type the **gd** modeless command followed by the grid value, and press **Enter**.

Example: Type gd50 and press Enter to set a .050” display grid.

Tip: Setting a grid with a single value sets that value for x and y. You can set independent values for x and y by entering them as g <x value> <y value>. For example, type g 25 50 to set a .025” x and .050” y design grid.

Table 1-2 lists and describes the grid types:

Table 1-2. Grid Types

Grid Type	Used for
Design	General placement of all objects
Via	Placement of vias
Display	Visual aid

Step 5 - Place Parts

The placement commands are located on the Design Toolbar.

Result: You enter move component mode.

To place parts:

1. **Design Toolbar button** > **Move button**.
2. Click a part to begin moving it.
3. Right-click and choose other commands to rotate the part 90 degrees, flip it to the opposite side of the PCB or spin the part at .001 degree resolution.
4. Click in the new location to place the part.

Other placement commands exist that help you place parts including Radial Move, Rotate, and Spin.

Step 6 - Route and Unroute Traces

In PADS Layout, you convert unroutes to traces using any one of several routing commands. You can route traces with on-line checking of design rules to eliminate rule violations as you create traces.

To route traces using the basic route editor:

1. **Design Toolbar button** > **Add Route button**.

   Result: You enter basic route mode.
2. Click an unroute or component pin to start routing.

   Result: A new trace is started with the end point attached to the pointer.
3. Move the pointer and click to add new trace corners, or Shift+click to add new vias.
4. Move the pointer over the completion point of the unroute, such as a component pin, and click to complete it.

To route traces using the dynamic route editor:

1. Enable on-line design rule checking by typing **drp** and pressing **Enter**.
2. On the Design toolbar, click the **Dynamic Route** button.
3. Click an unroute or component pin to start routing.

   Result: A new trace is started with the end point attached to the pointer.

4. Move the pointer in the direction of the completion point. New corners are added for you.
5. Move the pointer and click to add additional trace corners, or Shift+click to add new vias.
6. Move the pointer over the completion point of the unroute, such as a component pin, and click to complete it.

To route using the dynamic autorouter:

1. Enable on-line design rule checking by typing **drp** and pressing **Enter**.
2. On the Design toolbar, click the **Auto Route** button.
3. Click an unroute or component pin to start the dynamic autoroute command. A new trace is added.

Tip: The dynamic autorouter is a single layer autorouter. It is limited to completing traces not requiring a via.

You can also reroute and unroute trace segments.

Step 7 - Create Plane Layers

There are two types of plane layers in PADS Layout —CAM and Split/Mixed. CAM plane layers are processed using the CAM commands and are strictly intended for whole layer planes without routing. Split/Mixed are planes containing multiple plane areas, with or without routing.

To create a CAM plane:

1. **Setup menu** > **Layer Definition**.
2. Click a layer in the layer list to set as a CAM plane.
3. In the Plane type area, select **CAM Plane**.
4. Click **Assign Nets**.
5. In the Plane Layer Nets dialog box, select a net and click **Add** to add it to the assigned nets list and assign it as a plane layer for the net.
6. Click **OK** to close the dialog box and complete the layer definition.

To create a split/mixed plane:

1. Click a layer in the layer list to set as a split/mixed plane.
2. In the Plane type area, click **Split/Mixed Plane**.

3. Click **Assign Nets**.
4. In the Plane Layer Nets dialog box, select one or more nets and click **Add** to add it to the assigned nets list and assign it as a plane layer for the selected nets.
5. Click **OK** to close the dialog box and complete the layer definition.

You can also create plane areas on split/mixed plane layers and split a plane area polygon.

Step 8 - Check for Rule Violations

Use Verify Design to check for clearance, connectivity, high speed, plane connection, test point, fabrication, and other rule violations.

To check a design for rule violations:

1. **Tools menu** > **Verify Design**.
2. In the Check area, click the type of check to perform.
3. Click **Setup** to set checking options.
4. Click **Start** to start the checking.
5. Violations are marked with graphical error markers at their location in the design and details are provided in the Verify Design dialog box.

Table 1-3 lists and describes the rule violation checks:

Table 1-3. Rule Violation Checks

This check	Checks
Clearance	For violations of minimum clearance rules with options for checking other design features including same net minimum clearance, drill-to-drill, trace width, and part-to-part minimum clearances.
Connectivity	For trace continuity errors resulting from incomplete routing, pins not connected plane layers, and drill size greater than pad size.
High Speed	Specified nets for threshold violations of capacitance, impedance, delay, parallelism, and track length. It also checks for trace loops and stubs.
Plane	For conditions that prevent a pin or via from connecting to a plane layer. These conditions include:· A pin or via has a pad size on a plane layer greater than the drill size.· A pin or via has plane thermal status turned off and is not connected to the plane by some other means.· A pin is isolated from the plane due to discontinuities in plane shape (opens).

Table 1-3. Rule Violation Checks (cont.)

This check	Checks
Test Points	For violations of test point rules and preferences including minimum test point clearances, preferred side, minimum pad size, test points per net, and others.
Fabrication	For conditions that result in fabrication errors including acid traps, slivers, silkscreen over pad, minimal annular ring, solder bridges, and others.
Latium Design Verification	For violations of rules not verified in PADS Layout, including vias under SMDs and differential pairs, using a PADS-Router-based clearance rules check.
Wire Bonds	All die parts for violations of wire bond length, width, maximum angle, and minimum wire bond to pad rules.

Step 9 - Annotate the Design

You add dimensions to your design using the dimensioning tools.

To dimension automatically:

1. **Dimensioning toolbar button** > **Auto button**.

   Result: You enter Auto mode.

2. Click a line segment, route segment, or round pad. The dimension attaches to the pointer.
3. Click to place the dimension.

To dimension by selection:

1. Click a button associated with a type of dimensioning.
2. Right-click and select any snap and edge preferences you want to use. Snap and edge preferences determine where the dimension text snaps to and what alignment it will use.
3. Click the first object.
4. Click the second object. The dimension is attached to the pointer.
5. Click to place the dimension.

Step 10 - Generate Reports

Reports are generated in two ways within PADS Layout. You can use the Reports command to generate several standardized reports or you can generate custom reports using Basic scripting.

To generate a standardized report:

1. On the File menu, click **Reports**.
2. In the Report dialog box, click the report to run, and click **OK** to generate the report.

To generate a Basic report:

1. On the Tools menu, point to Basic Scripts, and then click **Basic Scripts**.
2. In the Basic Scripts dialog box, click the script to run, and click **Run** to execute the script and generate the report.

Step 11 - Output the Design

All of the typical outputs required for getting a design fabricated and assembled are generated in the CAM tool. Each output is defined as a document in the CAM document manager.

1. **File menu** > **CAM**.
2. Click **Add** to define a document.
3. In the Add Document dialog box, type a document name.

   Example: Photo plot layer 1
4. In the Document Type list, select a document to create.

   Result: The Layer Association dialog box may appear. It will not appear for custom, NC drill, or Verify photo document types.
5. In the Layer list, select a layer and click **OK**.
6. Type an Output File name or accept the default name.
7. In the Customize Document area, click the **Layers** button to select which items to include in the output. Click the **Options** button to set other options including positioning, drill symbols, name suppression, etc.
8. In the Output Device area, click the button that represents the output device to use. The device type should correspond with the document type.
9. Click **OK**.
10. Select the CAM document to process, and click **Run**.

Chapter 2
Managing Licensed Options

Installed Options Dialog Box

Use the Installed Options dialog box to verify and configure your PADS Layout licensing information (that is, the licensing options that are available for your system and your license file or status).

- **Help** menu > **Installed Options**.

To use the two tabs available on the Installed Options dialog box:

- Use the Options tab to display the status of your licensing options. If you are using floating licensing, use this tab to view or control the availability of different options from the floating license server (that is, to check different options and their associated modules in or out). If you are using node-locked licensing, you generally use this tab to verify the list of licensing options tied to your hardware key.
- Use the License File tab to select and then view license file information, either the actual license file (for node-locked licensing) or the feature usage status associated with a server license (for floating licensing).

Checking Licensing Options In and Out

Use the Options tab to configure your licensing information. You generally need to check individual licensing options in or out if you are using floating licenses. The configuration is stored in the powerpcb.ini file.

Tip: For node-locked licensing, all of the licensed options are available and checked out. For floating licensing, the first person logged in controls the available options.

- **Help** menu > **Installed Options** > **Options** tab.

To check all available licensing options in and out:

1. Make sure that the **Check Out All Available Options** check box is checked. If this box is not checked and all options are not checked in the Option list, you can check out all available options by choosing **Select All** or by checking the **Check Out All Available Options** check box.
2. Click **OK**.

To select the individual options associated with individual PADS Layout modules and optionally select a database limit:

1. Clear the **Check Out All Available Options** check box.
2. Optionally, click **Select None** to clear all of the options before selecting individual options. This is helpful if you want to start with a clean slate.
3. Select options by doing one of the following:
   - In the check boxes to the left of the options in the Options list, check each option you want to use.
   - In the Modules area, select the predefined set of options associated with one or more PADS Layout modules.

   Tip: If more than one module sets a particular option, turning off one of the modules does not turn off that shared option. The option is only turned off when all of the modules that use it are turned off. Also, some individual options control other options. For example, clearing the Advanced Packaging Toolkit option clears BGA options.
4. Optionally, select the database limit that is appropriate for your licensing scheme in the Database Limit area. The database limit is associated with the number of connections (pin pairs) you can have: 1500 for the Standard Database and 6250 for the Expanded Database. Select Standard, Expanded, or Unlimited for the database limit and verify the availability of the selected limit, per your license.
5. When you are done checking out your options, click **OK** to apply your selections.

Tips:

- If PADS Layout can check out the options in the modules, the Status is Available. If you can't check out the module, either because it isn't part of your PADS Layout package or because the licenses are already in use, the Status is Not Available.
- When the options associated with a particular module are Not Available, you will not be able to access the options in PADS Layout (that is, the feature will be grayed out). For example, if Assembly Variants has a status of Not Available, the PADS Layout Tools menu will show Assembly Variants as grayed out.

Related Topics

Installed Options Dialog Box

Viewing a License File or License Status

License File Definition

Viewing a License File or License Status

If you are using node-locked licensing, you can view the contents of a license file. If you are using floating licenses, you cannot view the actual license file, but you can view the status of the features associated with a server license.

- **Help** menu > **Installed Options** > **License File** tab.

For node-locked licenses

To view a license file:

1. Select the license file that you want to view from your list of license files.
2. Click **View** and the bottom portion of the screen displays the contents of the selected file.

For floating licenses

To view the status of the features associated with a server license:

1. Select the server license file for which you want to display feature status.
2. Click **Status**. The feature usage information appears in the bottom portion of the screen.

Related Topics

Installed Options Dialog Box

Checking Licensing Options In and Out

License File Definition

Checking Out Suite Licenses

Use the PADS Suite Configuration dialog box to manage which suite licenses you consume.

Restriction

This functionality is applicable only if the following is true:

- You have floating licenses

  AND

- On your license server you have:
  - A mix of different PADS Suite licenses
    Example: PADS LS/padsls_c and PADS ES/padses_c

Or

- One or more PADS Suite license
  Example: PADS ES/padses_c
  AND
  One or more PADS Kit license
  Example: pwrshell and other similar features, as part of PADS Layout 065 Kit

Warning: To use this functionality it is imperative that you remove any FlexNet Options files and/or batch script customization; otherwise it will not work correctly.

Procedure

1. **Help** menu > **Installed Options** > **Suite Configuration** button.
2. The check the status of all listed licenses, click **Check status**.
   Tip: To see which license features are being consumed for each of the product options, see the Installed Options dialog box.
3. Select the **Control suite license checkout and select from the following list** check box.
   Tip: Click to clear the Control suite license checkout and select from the following list check box to have your licensing consumption work as it did in PADS 9.3.1 and earlier versions.
4. Select one or more licenses from the list.

   Tips:

   - If you select a specific Suite license for check out, it does not impact your ability to also check out other option licenses using the Installed Options dialog box. For example, you can still check out pwradvrules for Advanced Rules.
   - To check out Kit licenses only (also known as atomic or unbundled licenses), for example "pwrshell" for PADS Layout, select the Control suite license checkout and select from the following list check box and click Select none.
5. To check to see which Suite licenses are being used each time you start PADS Layout, select the **Show this dialog box on program startup** and **Automatically check license status on dialog box startup** check boxes.
6. To copy what you've done to PADS Logic and/or PADS Router, select the product in which you want to copy to.
   Warning: Copying your selections may make the other PADS products open in demonstration mode.

   Tip: If you make a change in one PADS product and Copy to one or more of the other PADS products, the change is immediate in the PADS product you make the change in, but NOT for the other products(s) that may be open. The change will go into effect for the other PADS product the next time you open it or if you go click Apply on the Suite Configuration dialog box.

7. Click **OK**.

Related Topics

PADS Suite Configuration Dialog Box

License File Definition

This section describes the license file and the options enabled within that file. All installed options appear in the Installed Options Dialog Box. The lines in this example license file include most available options. Because all license files differ based on the options you purchase, you may not have all of the following lines in your license file.

Options

```
INCREMENT pwradvedtrf mgcld \
```

Provides access to the following commands: Add from DXF in the Decal Editor, the Via Patterns tab in Options, Add Via Shield for nets and pin-pairs, Via Stitch and Via Stitch Mode for coppers and hatch outlines, and Create Copper Path and Convert to Copper Paths for pin-pairs.

This feature is listed as Advanced Editing/RF on the Installed Options dialog box.

```
INCREMENT pwradvpkgtool mgcld \
```

Provides access to BGA functions. If you do not have this option installed, the BGA toolbar button is disabled, as are all BGA functions.

This feature is listed as Advanced Packaging Toolkit on the Installed Options dialog box.

```
INCREMENT pwradvrules mgcld \
```

Expands the design rules hierarchy to include access to Group, Pin Pairs, Decal, Component, Conditional Rules, and Differential Pairs buttons in the Design Rules dialog box.

This feature also provides High Speed Checking, or electrodynamic checking, on the Verify Design dialog box and related functionality.

This feature is listed as Advanced Rules on the Installed Options dialog box.

```
INCREMENT pwranalog mgcld \
```

Provides access to analog objects. Allows custom control of teardrop shape, length, and width in the Teardrops Options dialog box. Also enables the Jumpers dialog box to let you add jumpers when adding wires during routing or route editing.

This option is listed as Analog on the Installed Options dialog box.

```
INCREMENT pwrautoconn mgcld \
```

Provides access to the BGA Route Wizard. This option is listed as Auto-connect on the Installed Options dialog box.

```
INCREMENT pwrautoplace mgcld \
```

Provides access to the Cluster Placement dialog box. If you do not have this option, automatic placement functions are disabled.

This option is listed as Auto-placement on the Installed Options dialog box.

```
INCREMENT pwrbgafanout mgcld \
```

Provides access to BGA fanout features. This affects the BGA Fanouts tab and the BGA Fanouts checkbox of the Connections tab in the BGA Route Wizard dialog box.

This option is listed as BGA Fanout on the Installed Options dialog box.

```
INCREMENT pwrbgarouter mgcld \
```

Provides access to BGA routing. If you do not have this option, the Routing tab and the Generate Connections and Route radio button are disabled. This option is listed as BGA Router on the Installed Options dialog box.

```
INCREMENT pwrbndpadfanout mgcld \
```

Provides access to the Substrate Bond Pad Fanout (SBP) checkbox of the Connections tab in the BGA Route Wizard dialog box. If you do not have this option, the Substrate Bond Pad Fanout checkbox of the Connections tab in the BGA Route Wizard dialog is disabled.

This option is listed as Bond Pad Fanout on the Installed Options dialog box.

```
INCREMENT pwrcam mgcld \
```

Provides access to the Define CAM Documents dialog box from the CAM item on the File menu. CAM allows creation and modification of plot definitions and allows creation of CAM output.

This option is listed as CAM on the Installed Options dialog box.

```
INCREMENT pwrcam350 mgcld \
```

Provides access to the CAM350 dialog box and related functionality, as well as to the external CAM350 support executable (for design output) and the CAM350 Link functionality. This affects the CAM350 item on the Tools menu.

This option is listed as CAM350 Link on the Installed Options dialog box.

```
INCREMENT pwrcamplus mgcld \
```

Provides access to the CAM Plus dialog box, which enables automatic assembly and pick and place machinery interface. If you do not have this option, the CAM Plus item on the File menu is disabled.

This option is listed as CAM Plus on the Installed Options dialog box.

```
INCREMENT pwrcluplace mgcld \
```

Provides manual and automatic cluster creation and modification and manual union creation and modification. Also provides the Cluster Manager dialog box and Build Clusters and Place Clusters in the Automatic Cluster Placement dialog box. If you do not have this option, the Cluster Manager item on the Tools menu is disabled.

This option is listed as Cluster Placement on the Installed Options dialog box.

```
INCREMENT pwrcopperflood mgcld \
```

Provides access to the flooding functionality (for example, Flood All and Fast Flood on the Pour Manager dialog box). Also provides Flood in the Drafting toolbar and the Flood command in the shortcut menus.

This option is listed as Copper Flood on the Installed Options dialog box.

```
INCREMENT pwrdieflag mgcld \
```

Secures access to the BGA Die Flag Wizard in the BGA toolbar. This option is listed as Die Flag Wizard on the Installed Options dialog box.

```
INCREMENT pwrdrafting mgcld \
```

Provides access to auto-dimensioning functions and drafting (and related 2D line functionality). If you do not have this option, all auto-dimensioning functions (the Auto-dimensioning toolbar) and all drafting editing functions (the Drafting toolbar) are disabled.

This feature is listed as Drafting Editing on the Installed Options dialog box.

```
INCREMENT pwrdre mgcld \
```

Provides access to dynamic routing. If you do not have this option, Dynamic Route, Quick Route, Bus Route, and Smooth commands are disabled. This feature provides dynamic routing on the shortcut menus and provides Dynamic Route and Sketch Route on the Design toolbar.

This option is listed as Dynamic Route Editing on the Installed Options dialog box.

```
INCREMENT pwrdxf mgcld \
```

Provides importing and exporting of AutoCAD® DXF® format files.

This option is listed as DXF on the Installed Options dialog box.

```
INCREMENT pwreco mgcld \
```

Provides the ECO toolbar and related functionality. If you do not have this option, you cannot run PADS Layout in the ECO mode.

This option is listed as ECO on the Installed Options dialog box.

```
INCREMENT pwrgds2 mgcld \
```

Provides access to the GDSII file import feature. This option is listed as GDS-II/Die Import on the Installed Options dialog box.

```
INCREMENT pwrgenedit mgcld \
```

Provides access to the following:

- Auto Route on the Design toolbar and related functionality as well as autorouting features on the shortcut menus. This option secures access to Dynamic Autorouting. If you do not have this option, the Auto Route button on the Design toolbar is disabled, as are the Auto Route entries on the Pin and Pin Pair menus.
- File New, File Save, File Save As commands.
- On-line Design Rule Checking (DRC) options on the Design Options dialog box.
- Move, rotate, spin, flip, disperse, nudge, and align components. Also provides Move Part, Rotate Part, Spin Part, and Swap Part on the Design toolbar.
- Route modifying commands (Move, Stretch, Split, Route Loop, Add Corner, and Add Via) on the shortcut menus. Also provides Add Corner, Split, and Add Route in the Design toolbar.
- Break Reuse

This feature is listed as General Editing on the Installed Options dialog box. It does not control DRE.

```
INCREMENT pwridf mgcld \
```

Provides importing and exporting of IDF format files. This option is listed as IDF Interface on the Installed Options dialog box.

```
INCREMENT pwrlibedit mgcld \
```

Provides access to the PCB Decal Editor, the Library Manager, and related functionality including Save to Library on the shortcut menus. This option is listed as Library Editor on the Installed Options dialog box.

```
INCREMENT pwrplatingtails mgcld \
```

Provides access to the Plating Tails pass of the BGA route. This option is listed as Plating Tails on the Installed Options dialog box.

```
INCREMENT pwrradialplace mgcld \
```

Provides access to Radial Move, Radial Move Setup, and Create Array in the Layout Editor, and Radial and Polar Step and Repeat and Radial Move Setup in the PCB Decal Editor. This option is listed as Radial Placement on the Installed Options dialog box.

```
INCREMENT pwrreuse mgcld \
```

Provides physical design reuse operations including Create, Add, Make Like, and Properties functions. Specifically, this option affects the following commands and dialog buttons:

- Make Reuse
- Add Reuse
- Move Reuse
- Save To Lib
- Reset Origin
- OK and Apply buttons in the Reuse Properties dialog

This option is listed as Physical Design Reuse on the Installed Options dialog box.

```
INCREMENT pwrshell mgcld \
```

Provides the graphical user interface. If you do not have this option, PADS Layout runs in Demo mode.

```
INCREMENT pwrspecctralink mgcld \
```

Provides the SPECCTRA translator. If you do not have this option, the SPECCTRA selection on the Tools menu is disabled.

This option is listed as SPECCTRA Link on the Installed Options dialog box.

```
INCREMENT pwrsplitplanes mgcld \
```

Provides the Plane Connect command in the Pour Manager, flooding of plane layers by any other means, the C modeless command, DRC on split/mixed plane layers, the automatic update of thermal status and unroute visibility, verify design for planes, ability to define a split/mixed plane area, and access to the Split Planes on the Options dialog box. This option is listed as Split/Mixed Plane on the Installed Options dialog box.

```
INCREMENT pwrdbexp, pwrdbstd, or pwrdbunl mgcld \
```

These options set design connection limits as follows:

- Expanded sets the limit to 6250 connections.
- Standard sets the limit to 1500 connections.
- Unlimited allows the removal of connection limitations from the database.

Without one of these licensing options in place, the connection limit is 100. If you open a design or modify a design to exceed your database limits, various functions throughout the program are disabled. This option is controlled on the Database Limit area of the Installed Options dialog box.

```
INCREMENT pwrvariants mgcld \
```

Provides access to Assembly Variants from the Tools menu and the Assembly Variants supporting features within Reports and CAM (for example, the Assembly Option checkbox in the Report dialog box). This feature is listed as Assembly Variants on the Installed Options dialog box.

```
INCREMENT pwrverify mgcld \
```

Provides access to the Verify Design dialog box (accessible from the Tools menu). This option is listed as Verify Design on the Installed Options dialog box.

PADS Router-only options

```
INCREMENT blzadvrulesstd, blzadvrulesexp, or blzadvrulesunl mgcld \
```

Secures access to the use of (and the design verification of) Extended Rules (Net Classes, Pin Pair Groups, Pin Pair, Component, Decal, Differential Pair rules, Conditional Rules, and all Matched Length rules). These consist of the following:

- The New, Delete, and Rename shortcut menu entries for Net Class, Pin Pair Groups, and Conditional Rule items in the Project Explorer Window
- The OK and Apply buttons on the Net Class, Pin Pair Group and Conditional Rule dialog boxes
- The design verification of all the rules mentioned above.

This option is listed as Extended Rules on the Installed Options dialog box in PADS Router.

```
INCREMENT blzanyangle mgcld \
```

Provides access to any-angle routing. This consists of securing the Any Angle radio button in the Routing tab of the Options dialog.

If your design already has any-angle specified and you do not have this option, you are notified and asked if you want to route with Diagonal instead. If you choose to route with Diagonal, the design is changed to this setting. This option is listed as Any Angle on the Installed Options dialog box in PADS Router.

```
INCREMENT blzautoroute mgcld \
```

Allows access to the following automatic routing functions:

- Miters pass type (add miters sub-type) on the Strategy tab of the Options dialog box.
- Optimize pass type (via minimize and smooth sub-types) on the Strategy tab of the Options dialog box.
- Route pass type on the Strategy tab of the Options dialog box.

This option is listed as Route in the Installed Options dialog box in PADS Router.

```
INCREMENT blzdbstd, blzdbexp, or blzdbunl mgcld \
```

Controls access to reading designs up to the pin pair limit. This limit only applies to reading a design, not saving it. If PADS Router creates new pin pairs (thus exceeding the pin pair limit), you are warned when you save the design. This option is listed as Database on the Installed Options dialog box in PADS Router.

These options set pin-pair limits as follows:

- Expanded sets the limit to 6250.
- Standard sets the limit to 1500.
- Unlimited allows the removal of pin-pair limits from the database.

Without one of these licensing options in place, the pin-pair limit is 100.

```
INCREMENT blzdft mgcld \
```

Controls access to the use of test points. This option is listed as Test Point on the Installed Options dialog box in PADS Router.

Tip: This feature also affects the Run button on PADS Layout's DFT Audit dialog box.

```
INCREMENT blzdrestd, blzdreexp, or blzdreunnl mgcld \
```

Controls access to the Dynamic Routing functionality, such as: follow the pointer while avoiding obstacles and adhering to design rules, automatically keeping good pad entries, plowing, smoothing, Quick Route, and local Complete command. This option is listed as Dynamic Route Editing on the Installed Options dialog box in PADS Router.

```
INCREMENT blzgenedit mgcld \
```

Allows access to the general modifying functions on all menus and toolbars including the Save command, and the Undo, Redo, Move, and Delete commands. This option also controls access to online Design Rule Checking (DRC). When this option is disabled, PADS Router works in viewer mode and online DRC checking is disabled.

This option is listed as General Editing on the Installed Options dialog box in PADS Router.

```
INCREMENT blzhsdauto mgcld \
```

Controls access to the routing to the Advanced Rules, such as: Minimum and maximum length, Net Class and Pin Pair groups, Matched Length Net groups, Matched Length Pin Pair groups, differential pairs, and tune all nets attached to a component. This option is listed as High-speed Routing (auto) on the Installed Options dialog box in PADS Router. This option requires both the advanced rules and high-speed DRE options; if you do not acquire both of these options, you cannot select the interactive High-speed Routing (auto) option.

```
INCREMENT blzhsddre mgcld \
```

Controls access to the interactive routing to the high-speed rules (minimum and maximum length, net class and pin pair groups, matched length net and pin pair groups, differential pairs, tune trace length at completion of interactive routing, tune nets that are interactively pushed and shoved), advanced trace length monitor, matched length monitor, and interactive addition of accordions). This option is associated with the PADS Router advanced rules option. If you do not acquire the advanced rules option, you cannot select the interactive High-speed Routing (manual) option.

```
INCREMENT blzlayers2, blzlayers4, blzlayers8, or blzlayersunl mgcld \
```

Controls the number of routable design layers (2, 4, 8, or unlimited). Each layer limit corresponds to a limit licensing option. Any time your design exceeds the limit (either at File Open or because you changed layer routing enable), you are notified, and all routing functions are disabled. This option is controlled by the Routing Layers limit on the Installed Options dialog box in PADS Router.

```
INCREMENT blzshell mgcld \
```

If you do not have this option, PADS Router runs in Demo mode.

```
INCREMENT blzverify mgcld\
```

Controls access to the use of the basic Design Verification functionality, such as all object clearance, which includes the Placement Outline and Via at SMD checks. This option is listed as Verify Design on the Installed Options dialog box in PADS Router.

Tip: This feature also affects Test Point checking and Latium Design Verification in PADS Layout's Verify Design dialog box.

Chapter 3
User Interface

This section teaches you how to start up PADS Layout and discusses various Start-up options. There is also a comprehensive list of Modeless Commands - shortcut keys for common commands. Some modeless commands are commands that are not found elsewhere. A couple of Modeless Commands are used to activate start-up options.

Learn how to use the Project Explorer window, and learn how to customize the default software settings. Learn how to quickly see the design contents within the window.

Starting PADS Layout

You can start PADS Layout from the Windows Start menu or from shortcut icons.

- **Start** > **Programs** > **Mentor Graphics SDD** > **PADS <latest_release>** > **Design Layout & Routing**

Tip: The installation program automatically creates a shortcut on the Windows desktop to PADS Layout.

Start-up Options

You can use start-up options, known as command line switches, to control the initial PADS Layout configuration. Use command line switches to enable different options, to open a file, start macros, and record a PADS Layout session. You can type multiple command line options.

You add start-up options to the PADS Layout program folder or to the PADS Layout shortcut icon.

Table 3-1 lists and describes the command line options you can start when you start PADS Layout.

Table 3-1. PADS Layout Command Line Options

Option	Description
file name	Opens the specified design file when you start PADS Layout. Type the full folder path and file name. Use quotation marks for folders or file names with spaces. **Example:** "*C:\PADS Projects\Samples\preview.pcb*" **Restriction:** Do not use a forward slash (/) before the file name in the command line.

Table 3-1. PADS Layout Command Line Options (cont.)

Option	Description
/BMW[=initials]	([ ] represents optional text.) Opens the Basic Media Wizard. Use the Basic Media Wizard to start recording a session log or to convert the previous session log to media that can be replayed by Basic Log Test. To create session media files for the current PADS Layout session, use the BMW modeless command. To use the BMW command line switch, type /BMW or /BMW=xx, where xx is your initials, in the command line. Note the capitalization. This option is associated with another modeless command, BLT. BLT is the Log Test; it finds and runs the session media created by BMW to play back a recorded PADS Layout session. **See also:** Typing Modeless Commands
/DoTest	Runs the integrity test on every file open.
/l	Opens the last file you had open when you start PADS Layout.
/mmacro name	Runs the specified macro in the default macro file. For example, to run the macro MyMacro, type /mMyMacro.
/Mmacro file	Specifies the file to use as the default macro file. For example, to run the macro MyMacro contained in the file user1.mcr, type /Muser1.mcr /mMyMacro. Note the required capitalization. **Requirement:** Macros must be in the \PADS Projects folder.
/nc	Starts PADS Layout without displaying the splash screen that includes copyright information.
/NTL	Disables true layer associations. When you use Flip Side, the layer attributes will not move to the new layer with the component. Type /NTL to disable true layer associations. Note the required capitalization. TrueLayer is enabled by default and moves mask layer definitions with a part that is placed on the opposite side of the board. It also correctly plots paste masks of documentation-level pad shapes in CAM. The layer that the definitions move to is set in the Component Layer Associations dialog box. **See also:** Add or Edit a CAM Document**,** Associating Component and Documentation Layers
/sXXX	Starts a Basic script when you start PADS Layout. Use quotation marks for file names with spaces. **Example:** "/sC:\PADS Projects\Samples\Scripts\Layout\Unsupported\Attributes to Excel.bas"

Adding Start-up Options to a New PADS Layout Program Folder Item

The PADS Layout program folder item created during installation is read-only and you cannot add start-up options to it. However you can create a new program folder item and add PADS Layout start-up options to it.

To add start-up options to a new program folder item:

1. Right-click over the Windows Start button and click **Explore All Users**. Windows Explorer opens.
2. Navigate to the Programs\Mentor Graphics SDD\PADS<latest_release>\PCB Layout start menu folder under the All Users login name.

   Example: *C:\Documents and Settings\All Users\Start Menu\Programs\Mentor Graphics SDD\PADS<latest_release>\PCB Layout*.
3. In a blank area on the right side of Windows Explorer, right-click, point to **New**, and then click **Shortcut**.
4. Browse to powerpcb.exe. Do not click Next.

   Example: PADS Layout is typically installed to *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs\powerpcb.exe*.
5. Click in the box, press **End**, press **Spacebar**, and then type the command line switch you want to use. Click **Next**.

   Example: To start PADS Layout with preview.pcb, the command line should read: "*C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs\powerpcb.exe*" "*C:\PADS Projects\Samples\preview.pcb*".

   Requirement: Enclose with double quotes " " each string that contains a space.

   Restriction: When specifying a file to start, do not use a / before the file name.

   Tip: You can specify multiple command line switches.
6. Type in the box the name of the program folder item to create, and then click **Finish**.
7. To start PADS Layout using the start-up options, close PADS Layout if it is running, and then click the new program folder item.

Adding Start-up Options to a New PADS Layout Desktop Shortcut Icon

The PADS Layout desktop shortcut icon created during installation is read-only and you cannot add start-up options to it. However you can create a new shortcut and add PADS Layout start-up options to it.

To add start-up options to a new desktop shortcut:

1. Right-click over an empty area of the desktop, point to **New**, and then click **Shortcut**.
2. Browse to powerpcb.exe. Do not click Next.

   Example: PADS Layout is typically installed to *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs\powerpcb.exe*.

3. Click in the box, press **End**, press **Spacebar**, and then type the command line switch you want to use. Click **Next**.

   Example: To start PADS Layout with preview.pcb, the command line should read: "*C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs\powerpcb.exe*" "*C:\PADS Projects\Samples\preview.pcb*".

   Requirement: Enclose with double quotes " " each string that contains a space.

   Restriction: When specifying a file to start, do not use a / before the file name.

   Tip: You can specify multiple command line switches.

4. Type in the box the name of the shortcut to create and click **Finish**.
5. To start PADS Layout using the start-up options, close PADS Layout if it is running, and then double-click the shortcut.

Checking for PADS Updates

The PADS products automatically check for a new software version when you launch an application. If a new version is detected, a tooltip is displayed in the system tray.

Requirement

An internet connection is required for the check.

Procedure - Downloading the Update

Once the new version is detected, you can download the update.

1. Right-click the icon in the system tray.
2. Click **Open download page**.

3. Follow the instructions on the download page.

Procedure - Disabling the Check for Updates

If you don’t want to automatically check to see if there is a new version available, you can disable the Check for Updates. You can enable the check at any time in the future, or you can manually check for updates at any time.

1. **Help** > **Check for updates**.
2. In the Check for Updates dialog box, select **Disable “Check for Updates” functionality**.

Procedure - Checking for Updates Manually

There is no need to check for updates manually unless you’ve disabled the automatic check.

1. **Help** > **Check for updates**.
2. In the Check for Updates dialog box, click **Check for Updates**.

Switching to PADS Router

To quickly move your design to work in PADS Router, use the Route button on the main toolbar. Depending on your settings, this will either open PADS Router with your design displayed, or enter Synchronization mode.

Tips:

- You must enable Synchronization mode and then restart PADS Layout before Synchronization modes takes effect.
- To turn off the display of licensing warnings when switching between PADS Layout and PADS Router, type the following text in the Blazerouter.ini file:

  ```
  [Security]
  DisplayWarnings=0
  ```

  (You don’t need to restart PADS Router.)

Synchronization Mode

Synchronization Mode links PADS Layout and PADS Router. As you work in PADS Router, you’ll see the changes reflected in PADS Layout and vice versa; if PADS Layout is the active program, changes made there are reflected in PADS Router. When you’re in Synchronization mode, you can instantly switch between the two programs. If an unsupported command is performed or an unsupported option is changed, the inactive program will become out of sync.

Tip: You must enable Synchronization mode and then restart PADS Layout before Synchronization modes takes effect.

Procedure

- Main toolbar > **Route** button

Result

PADS Router opens and your design is loaded. PADS Layout is switched to DRC Off mode and is now the inactive program. As you work in PADS Router, you'll see the changes reflected in PADS Layout. You can quickly switch back forth between the two programs and continue to be in Synchronization mode. If an unsupported command is performed or an unsupported option is changed, the inactive program will become out of sync.

Tips:

- Most menu items and commands are disabled or are read-only for the inactive program.
- Look at the status bar to quickly determine which mode the program is in: Active, Inactive, or Out of sync.
- Automation can still be used; however, most Automation methods will not return any valuable results and will not make any changes in the database.
- To switch back to PADS Layout, use the Layout button on the main toolbar in PADS Router.

Enabling Synchronization Mode

You must enable Synchronization mode to link PADS Layout and PADS Router correctly.

Procedure

1. **Tools** menu > **Options**.
2. In the Options Dialog Box, click the Synchronization page.
3. In the Layout and Router Synchronization area, click **Enable**.
4. To restore the DRC mode when you switch back to PADS Layout to what you had before switching to PADS Router, click **Restore DRC Mode on return**.
   Tips:
   - Switching to PADS Router automatically places your design in PADS Layout in DRC Off mode; the DRC mode in PADS Router is not affected.
   - Depending on the size of your design, restoring the DRC mode may take a few minutes; therefore, it is recommended to not restore your DRC mode upon return.

5. To receive a warning about being placed into DRC Off mode when you switch to Synchronization mode, click **Warn about switching to DRC Off mode**.

6. Click **OK**.

7. Restart PADS Layout.

Unsupported Actions in Synchronization Mode

There are a few commands and options that are not supported in Synchronization mode. If you perform one of these commands, or change one of the options, PADS Layout and PADS Router will become out of sync.

Tip: To re-synchronize the two programs, click the Layout button on the main toolbar in PADS Router if PADS Router is active; or click the Route button on the main toolbar in PADS Layout if PADS Layout is active.

Unsupported commands

- Changing a drafting object: copper, copper pour, text, 2D lines, keepouts, labels, board outlines, dimensioning
  Exception: When drafting objects are attached to a component and the component is moved, it will not cause the programs to become out of sync.
- Changing rules
- Changing padstacks or layer definitions
- Importing ASCII files (or any other import)
- Verifying the design
- Changing reuses and clusters
- ECO or BGA operations
- Updating from the library
- Using the Decal Editor
- Inserting an OLE object
- Any operation that clears the Undo buffer including Autorouting and large group moves

Unsupported option changes

- DFF
- DFT
- Verify Design

- Drafting
- Split/Mixed Plane
- Thermals
- ECO Options
- Via Patterns

Exiting Synchronization Mode

To exit Synchronization mode, close one of the applications. For example, if you want to continue working in PADS Layout, but no longer want to be in Synchronization mode, close PADS Router.

Typing Modeless Commands

You can set or change some settings and functions at any time using a shortcut key for the command called a modeless command. There are also many modeless commands that are not available anywhere else.

Procedure

1. Click in empty space in the design area.
2. Type the modeless command (shortcut key) for the command you want.

   Restriction: Use a space between the command and the argument. For example: N GND or GD 50.
3. Click **Enter**.

Related Topics

Modeless Command Dialog Box

Modeless Commands

Shortcut Keys

Project Explorer

- Click the **Project Explorer** button.

The Project Explorer shows a hierarchical structure for the objects in your design. It provides access to objects and rules. When you update your design, the hierarchical structure is automatically updated to reflect the changes you make.

Tip: The Hierarchical structure is available only when a design is open.

Restriction: The Project Explorer is not available in the PCB Decal Editor.

In this topic:

- Selecting Objects
- Zooming to Selection

Selecting Objects

You can select an object in the Project Explorer and have it automatically selected in the workspace.

To select an item in the workspace:

- Right-click and click **Allow Selection**.

Zooming to Selection

You can zoom to the item you select in the Project Explorer.

To zoom to the selection:

- Right-click and click **Zoom to Selection**.

Output Window

Use the Output window for displaying reports and session logs, macro editing and debugging.

- To display the Output window, click the **Output Window** button.

The Output window is located in the lower left section of the display window. You can dock or float the Output window. You can also display or hide the Output window.

See the following sections for more information:

Working with the Status tab

- Managing Session Logs

Working with the Macro tab

- Creating Macros
- Managing Macros
- Playing Back Macros

- Debugging Macro Scripts
- Using Command Line Switches with Macros

Related Topics

- Output Window in the *Concepts Guide*

Managing Session Logs

The following topics describe managing session logs:

- Session Log
- Navigating Pages in the Status Tab
- Filtering the Status Tab Display
- Searching Text in the Status Tab
- Printing Session Log Messages
- Displaying and Printing Reports
- Saving a Session Log to File
- Clearing the Session Log Display

Session Log

A session log, which appears in the Status tab of the Output window, contains all program output for the current session, including names of open and saved files, integrity test results.

The session log presents different types of information in different colors. Underlined items are links. Text colors representations are shown in the following table:

Table 3-2. Session Log Text Color Representations

Color	Meaning
Red	Errors
Green	Warnings
Black	Messages
Blue	Links to files, Web pages, and database objects

Navigating Pages in the Status Tab

Using the Status tab toolbar buttons in the Output window, you can navigate to the previous page, the next page, and refresh a display of reports and other pages. You can also stop updates to pages, and return to the session log display.

To perform these functions, use the following Status tab toolbar buttons:

Table 3-3. Status Tab Toolbar Buttons

Command	Description
Back	Displays the previous page.
Forward	Displays the next page.
Stop	Stops page updates.
Refresh	Refreshes the display of reports and other pages.
Home	Returns to the session log.

Filtering the Status Tab Display

The session log file messages in the Status tab are color coded by subject. You can choose to view any combination of the color coded messages.

To filter the display in the Status tab:

1. Right-click in the Output window and point to **Filter**. The Filter submenu appears and contains the following commands:

Table 3-4. Filter Submenu Commands

Command	Description
Error	Displays error messages.
Warning	Displays warning messages.
Message	Displays messages.
Show all	Displays all messages (error, warning, and message).

2. Click one of the submenu commands to filter the view. Check marks indicate which messages are turned on. Turn off message displays by clearing the check marks.

Searching Text in the Status Tab

Similar to searching in a document, you can search text in the Status tab. To find text in the Status tab:

1. Right-click and click **Find**.
2. In the Find dialog box, type the text you want to find in the dialog box and complete any other dialog box options.
3. Click **Find Next**. The tab scrolls to the occurrence of the word, highlighting the word.

Printing Session Log Messages

You can print a hard copy of the session log for review purposes. To print the session log:

1. Right-click and click **Print**.
2. In the Windows standard Print dialog box, change any Print dialog box options as needed.
3. Click **OK**.

Displaying and Printing Reports

The session log contains links to reports that you can view and print. The links are in blue and underlined.

To display the report:

- Click the link to view the report. The report appears replacing the session log as the active page in the **Status** tab.

To print the report:

1. Right-click and click **Print**.

   Alternative: On the Status tab toolbar, click the Print button.
2. In the Windows standard Print dialog box change any Print dialog box options as needed.
3. Click **OK**.

Saving a Session Log to File

You can save the session log for future reference.

- Click the Log to File button.

  Result: If a session log file already exists, new information is appended. If a session log file does not exist, a new file is created. The default path (\PADS Projects) for the session log file is set in the .ini file when you install the program.

Clearing the Session Log Display

You can clear the session log display each time you open a file. This prevents you from accidentally viewing information from a previously opened file. It does not delete the log file.

- Right-click and click Clear.

Related Topics

- Session Log in the *Concepts Guide*

Creating Macros

You can create macros to simplify redundant activities. You can record any set of procedural steps for replay as a single action. You can also nest macros.

Tip: Dialog box actions are recorded as results rather than actions, so when you replay, you don't see the dialog boxes in the replay process. Because of this you can't create a macro that stops on an open dialog box; it must follow through to some result or action. For example, you can create a macro that selects Open on the File menu, selects a file, and selects OK. The macro, when played back, opens a file.

The following descriptions are included in this topic:

- Creating a New Macro
- Recording Mouse Movements
- Saving a Macro

Creating a New Macro

To create a new macro:

1. Click the **Output Window** button.
2. On the Macro tab, click the **New** button. New macros are given a name of Macro#, where # is a numeric sequence such as Macro1 or Macro2.
3. You can click the **Compress mouse moves** and/or **Relative mouse moves** buttons. See Recording Mouse Movements for more information.
4. On the Macro tab toolbar, click the **Record** button.
5. Perform the keystrokes, commands, and mouse clicks to include in the macro.
6. On the Macro tab toolbar, click the **Stop** button.

You can also script a macro instead of recording mouse actions.

Recording Mouse Movements

Mouse movements are recorded in macros. You can record compressed or uncompressed mouse movements and relative or absolute movements.

Compress Mouse Mode—Compress mouse mode records only the start point and endpoint of a mouse movement. It does not record any of the intermediate coordinates between the start and end points. Compression is recommended under most circumstances because it significantly reduces the size of your macro file. Recording intermediate mouse movements increases the file size, but documents coordinate information if required for a special application.

Relative Mouse Mode—Relative mouse mode records the start point and endpoint of a movement in incremental coordinates instead of absolute coordinates.

Saving a Macro

1. Click the **Save** button.
2. In the standard Windows Save As dialog box, enter a filename, if desired, and click **Save**.

Related Topics

- Using Command Line Switches with Macros
- Macros in the *Concepts Guide*

Managing Macros

This following descriptions are included in this topic:

- Opening an Existing Macro File
- Viewing Multiple Open Macros
- Editing a Macro
- Saving the Macro

Opening an Existing Macro File

Macros are created in and stored in macro files that have a .mcr extension. To open an existing macro file (.mcr), you can use the menus or the toolbar.

1. Click the **Output Window** button and then click the **Macro** tab.
2. Click the **Open** button.
3. In the Open File dialog box, select the macro file to open and click **Open**.

You can open multiple macros in the macro editor. The macro editor also supports nested macros.

Viewing Multiple Open Macros

You can open multiple macros in the macro editor. To switch between these open macros:

- Click the macro you want to view in the List of Open Macros area of the Macro tab.

The macro script appears in the edit area.

Editing a Macro

You can copy or cut selected text to the Clipboard. You can also paste the selection from the Clipboard into the text window. You can paste text from the Clipboard into other applications. You can also switch between open macros to edit multiple macros.

To copy or cut and paste text in a macro:

1. Select the text you want to copy or cut.
2. On the Macro tab toolbar, click the **Copy** or **Cut** buttons.
3. On the Macro tab toolbar, click the **Paste** button. You will see that your selection has been pasted in the Output window at the insertion point.

   Alternative: Right-click in the Output window and click Copy, Cut, or Paste.

If you chose Notepad as the default text editor, longer macro files may not be loaded because of size constraints in Notepad.

To access large files using Edit, you must install an ASCII text editor with a suitable file size capacity. To change the default text editor:

1. Open the powerpcb.ini file in a text editor.
2. Modify the [GENERAL] section, specifying a new text editor executable name. Include the drive and folder if the new editor is not in your Windows folder.
3. Save the .ini file and close the text editor.
4. Proceed with editing a macro.

Saving the Macro

1. Click the **Save** button.
2. In the standard Windows Save As dialog box, enter a filename, if necessary, and click **Save**.

Related Topics

- The Macro Language in the Reference Guide
- Macros in the *Concepts Guide*

Playing Back Macros

You can play back an existing macro using Run. Run also resumes the playback of a paused macro. When you play a macro, you cannot use the mouse in the workspace.

The following descriptions are included in this topic:

- Playing Back a Macro
- Pausing a Playing Macro
- Stopping a Playing Macro

Playing Back a Macro

To play back a macro:

1. On the Macro tab, click the Open button and open a macro (.mcr) file. Recent macros can be found on the Tools menu > point to Macros > Macros menu.
2. On the Macro tab toolbar, click the **Run** button.

   Alternative: Right-click in the Macro tab and click **Run**.

Pausing a Playing Macro

You can pause a playing macro at any time.

- On the Macro tab toolbar, click the Pause button.

Tip: Click the Play button to play the macro again.

Stopping a Playing Macro

You can stop the playback of a macro at any time. However, you cannot resume the playback of the macro once you have stopped it. When you click Run, the macro starts from the beginning.

- Right-click and click **Stop**.

  Alternative: On the Macro tab toolbar, click the Stop button.

Related Topics

- Macros in the *Concepts Guide*

Debugging Macro Scripts

When playing back a macro, you can run it step-by-step, or to a certain location in the script. To perform these debugging tasks, insert breakpoints in the macro at the points at which you want the macro to stop.

The following descriptions are included in this topic:

- Setting or Removing Breakpoints
- Debugging the Macro Scripts
- Correcting Run-Time Errors

Setting or Removing Breakpoints

The ability to set or remove breakpoints is useful when you debug a macro. If the macro engine encounters a breakpoint when playing back a macro, it pauses the macro.

To set a breakpoint in a macro:

1. Place the cursor on the line in which to add a breakpoint.
2. Right-click in the Macro tab and click **Toggle Break**. This inserts a breakpoint at the current cursor location. A breakpoint marker appears in the gutter area.

   Alternative: On the Macro tab toolbar, click the Toggle Breakpoint button.

When the macro engine encounters a breakpoint while playing back a macro, it pauses the macro. The next line in the macro is marked with the instruction pointer.

Debugging the Macro Scripts

Once breakpoints are inserted, you can debug macros using the following tasks.

To play a single line of the macro:

- Right-click in the Macro tab and click **Step Over**.

  Alternative: On the Macro tab toolbar, click the Step over button.

To perform a subroutine call on the current line:

- Right-click in the Macro tab and click **Step Into**.

  Alternative: On the Macro tab toolbar, click the Step into button.

To return from the subroutine to the point from which it was called:

- Right-click in the Macro tab and click **Step Out**.

  Alternative: On the Macro tab toolbar, click the Step out button.

To play back a macro to a point:

- Right-click in the Macro tab and click **Step to Cursor**.

  Alternative: On the Macro tab toolbar, click the Step to cursor button.

To continue the execution from the current point:

- Right click in the Macro tab and click **Run**.

  Alternative: On the Macro tab toolbar, click the Run button.

Correcting Run-Time Errors

If run-time errors occur, the macro debugger switches to step-by-step mode and displays a detailed message on the status bar. The instruction pointer is set on the line that produced the error. After fixing the error, you can resume playback of the macro.

Related Topics

- Macros in the *Concepts Guide*
- The Macro Language in the Reference Guide

Using Command Line Switches with Macros

Using command line switches, you can record your session, load and run a macro, or record a macro to a specific file and location when you start PADS Layout.

The following descriptions are included in this topic:

- Recording a Session
- Recording Log Files to a Specific File and Location
- Running a Macro When You Start the Program

Recording a Session

1. In the Start menu, navigate to the shortcut for PADS Layout.
2. Right-click the shortcut and click **Properties**.
3. Click the **Shortcut** tab and click in the **Target** box.
4. At the end of the existing shortcut, type: **-log**. Make sure to type a space between "powerpcb.exe" and the "-log" command.
5. Click **OK**. When you run the program, a log is created.

Recording Log Files to a Specific File and Location

1. In the Start menu, navigate to the shortcut for PADS Layout.
2. Right-click the shortcut and click **Properties**.
3. Click the **Shortcut** tab and click in the **Target** box.

4. At the end of the existing shortcut, type: **-log:[path and file name]**. For example: -log:C:\PADS Projects\mylog.txt. Make sure to type a space between "powerpcb.exe" and the "-log" command.

5. Click **OK**. When you run the program, a log is created in the specified location using the specified name.

Running a Macro When You Start the Program

You can run a macro at start-up when you start PADS Layout from a shortcut (on your desktop or from the Start menu).

1. In the Start menu, navigate to the shortcut for PADS Layout.

2. Right-click the shortcut and click **Properties**.

3. Click the **Shortcut** tab and click in the **Target** box.

4. At the end of the existing shortcut, type:

   -run=[path and file name]

   For example:

   -run=C:\PADS Projects\Samples\mymacro.mcr

   Requirement: Make sure to type a space between "powerpcb.exe" and the "-run" command.

   Alternative: You can use a / instead of the hyphen (for example, /run=mymacro.mcr).

5. Click **OK**. When you run the program, the specified macro runs as soon as the program starts.

Related Topics

- Macros in the *Concepts Guide*
- Creating Macros

Opening a File That is Already in Use

The PADS products help you avoid making changes to a file that is already opened by another user.

The first user to open a file in a shared location becomes the owner of the file for the duration the file is open; the file is locked to all other users. If you try to open a file that someone else has already opened, you will get a warning message letting you know the current owner and the name of the computer from where the file is locked. You have the option to view a read-only version of the file but you will not be able to update it while the owner still has it open. You can save the file with another name using Save As.

Migrating User Settings

You can use the PADS User Settings Migration tool to extract your settings from one installation of PADS Logic, Layout, and Router and import them into another installation or version. For information on how to do this, see User Settings Migration in the *PADS User Settings Migration Guide*.

Customizing PADS Layout Default Settings

Several settings are used to define the default ASCII parameters of operation of PADS Layout. You can customize the user interface by modifying these system settings and saving the files.

In this topic:

- PADS Layout Default Settings
- Changing the PADS Layout Default Startup Conditions
- Changing the PADS Layout Default Startup File

PADS Layout Default Settings

The file DEFAULT.ASC contains Layout mode settings, and the file DECALEDT.ASC contains Decal Editor mode settings. In addition, you can create your own startup files and save them with the .stp extension.

Use the Save As Start-up File and Set Start-up File menu options to enable Layout mode and Edit Decal mode to work with startup files. Use the Set Start-up File dialog box to set the options you want to save in the file.

Changing the PADS Layout Default Startup Conditions

To change the default startup conditions for PADS Layout:

1. On the Tools menu, click **Options**.
2. Use the Options dialog box to change the design units, grids, and default font, etc.
3. Click **OK** to save the changes.
4. On the Setup menu, click **Layer Definition**.
5. Use the Layers Setup dialog box to change layer mode, number of electrical layers, layer associations, etc.

6. On the Setup menu, click **Design Rules**.
7. Use the Rules dialog box to change any rules for the design.
8. On the File menu, click **Save as Start-up File.** Then set the folder to *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Settings*.
9. Select the appropriate *.stp file from the list of files, or type a new name in the File name area if it does not exist.
10. Click **Save**. The Save as Start-up File dialog box displays.
11. In the Start-up File Output dialog box, click **Select Al**l or leave already selected options unchanged.
12. Leave other options unchanged and click **OK**.

Tip: You can add comments to the start-up file.

Changing the PADS Layout Default Startup File

To change the default start-up file for PADS Layout:

1. On the File menu, click **Set Start-up File**
2. In the Set Start-up File dialog box, select the appropriate file in the list of Start-up designs, and click **OK**.

To Zoom to Board Extents

To fit the board to the view press **Home** on the numeric keypad.

Customizing the PADS Interface

You can customize the PADS interface to suit your work style and design work. You can determine which toolbars are displayed, add items to toolbars and menus, and create custom toolbars, menus and shortcut keys.

To make customizations, use the Customize dialog box. You can invoke the dialog box in two ways:

- From the PADS interface, select **Tools menu** > **Customize**. All customizations you make are applied to the main view of the PADS tool.
- In a window of the interface (for example, the Output Window), right-click and select **Customize**. You customizations apply only to that window.

Your customizations are saved with your current workspace so that all of the changes you make to toolbars, menus, and shortcut keys are present when you work in that workspace again.

Related Topics:

- Customizing Toolbars
- Creating a Custom Menu
- Customizing Shortcut Keys
- Customizing the Appearance of the Screen

Customizing Toolbars

Use the Toolbars and Menus tab on the Customize dialog box (**Tools menu > Customize** > **Toolbars and Menus tab**) to create custom toolbars and shortcut menus.

- Creating a Custom Toolbar
- Resetting Toolbars to Defaults
- Deleting a Custom Toolbar
- Renaming a Custom Toolbar

Tip: To create a custom main menu, use the Commands tab on the Customize dialog box. See Creating a Custom Menu.

Related Topics

Moving Items on Toolbars and Menus

Creating a Custom Toolbar

To create a custom toolbar, you create a new empty toolbar and add items (commands) to it.

To create a custom toolbar:

1. **Tools menu** > **Customize** > **Tools and Menus tab**.
2. In the Toolbars box, click the **New** button.
3. Type the name for the toolbar and click **OK**.

   Results:

   - The new (empty) toolbar appears on the PADS interface.
   - The Toolbars and Menus tab lists the new toolbar, showing it as selected and enabled for display (the check box to the left of its name is selected).
4. Drag the toolbar to the place on the PADS interface where you want it.
5. To add items (commands) to your new toolbar, click the **Commands tab**.

6. In the Categories list, select a menu or toolbar name to display commands specific to that menu or toolbar. Or select **All Commands**.

   Restriction: If you are working in a special mode in PADS Layout or PADS Logic (for example, the Decal Editor in PADS Layout), some categories of commands are not available for customization.

7. In the Commands list, select the command you want and drag it to the toolbar.
8. When you have finished adding commands, click **Close**.

Related Topics

- Deleting a Custom Toolbar
- Adding Items to Toolbars and Menus
- Removing Items from Toolbars and Menus

Showing or Hiding a Toolbar

To increase space in the PADS interface, you can show the toolbars you need to use and hide others.

To show or hide toolbars:

1. **Tools menu** > **Customize** > **Toolbars and Menus tab**.
2. In the Toolbars list, select the toolbar.
3. To display the toolbar in the interface, select the check box to the left of its name. To hide the toolbar, clear the check box.
4. Click **Close**.

Tip: For information on other ways you can customize the appearance of toolbars and menus, see Customizing the Appearance of the Screen.

Deleting a Custom Toolbar

You can delete a custom toolbar (a toolbar you created).

Restriction: You cannot delete a system toolbar.

To delete a custom toolbar.

1. **Tools menu** > **Customize** > **Toolbars and Menus tab**.
2. In the Toolbars list, select a custom toolbar. Then click the **Delete** button.

Related Topics

- Adding Items to Toolbars and Menus
- Removing Items from Toolbars and Menus
- Showing or Hiding a Toolbar

Renaming a Custom Toolbar

You can rename a custom toolbar (a toolbar you created).

Restriction: You cannot rename a system toolbar.

To rename a custom toolbar:

1. **Tools menu** > **Customize** > **Toolbars and Menus tab**.
2. In the Toolbars list, select a custom toolbar and click the **Edit** button.
3. In the Toolbar Name dialog box, type the new name and click **OK**.

Related Topics

- Showing or Hiding a Toolbar
- Deleting a Custom Toolbar

Resetting Toolbars to Defaults

You can reset one or all system toolbars to their default buttons.

To reset a specific toolbar to its default settings:

1. **Tools menu** > **Customize** > **Toolbars and Menus tab**.
2. In the Toolbars list, select the toolbar.
3. Click **Reset**.

Tip: To reset all system toolbars to defaults, click **Reset All**.

Customizing Commands and Menus

You can customize commands that you can then use as selections on menus or as buttons on toolbars and you can customize Menus.

- Creating a Custom Command
- Defining Properties for a New Command
- Editing a Custom Command

- Deleting a Custom Command
- Creating a Custom Menu

Creating a Custom Command

You can create a custom command from:

- A command that already exists as a menu item or toolbar button. To create this kind of command, you select an existing command on which to base your new command. Then you define the properties of your new command.
- A macro command file. See Creating Commands from Macro Command Files.

To create a custom command from an existing command:

1. **Tools menu** > **Customize** > **Commands tab**.
2. In the Categories list, click a menu or toolbar name to display items (commands) specific to that menu or toolbar. Or click **All Commands**.

   Tip: If you made macro commands (on the Macro Files tab) available as commands, the Categories list includes the Macro category and the Commands list includes the macros. For more information, see Creating Commands from Macro Command Files.
3. In the Commands list, select the command on which you want to base your custom command. Then click the **New** button.
4. In the Add Command dialog box, specify the properties of your new command:
   a. In the Command name box, type the name of the command.
   b. In the Arguments box, type arguments for the command. Use a space to separate arguments. If an argument contains a space, enclose the argument in quotation marks (““).
   c. In the Description box, type a description of the custom command.
   d. If an image was associated with the original command, select **Use Default Image** to use that same image with your custom command. Select **Select User-Defined Image** to use a different image, edit an image, or create a new image.
   e. Click **OK** to close the Add commands dialog box and return to the Customize dialog box.
5. If you are finished with all customizations, click **Close**.

Tip: To add the command to a toolbar or menu, click the command and drag it from the Commands list to the toolbar or menu.

Related Topics

- Adding Items to Toolbars and Menus
- Creating Commands from Macro Command Files
- Resetting Toolbars to Defaults

Defining Properties for a New Command

To define properties for a new command:

1. **Tools menu** > **Customize** > **Commands tab**.
2. In the Commands list, click **New**, then click the new button. The Add command dialog box opens.
3. In the Command name box, type the name of the new command.
4. Optionally, in the Based on box, type the name of the command on which the new command is based.
5. In the Arguments box, type space delimited arguments for the new command.

   Requirement: If an argument contains a space, enclose it in quotation marks.
6. In the Description box, edit the command description, for example, to represent the argument values you added.
7. If an image was associated with the original command, select **Use Default Image**.

   or

   Select **Select User-defined Image**, then select a image, edit an image, or create a new one.
8. Click **OK** to close the Add Command dialog box and return to the Customize dialog box.

Related Topics

Customizing Commands and Menus

Editing a Custom Command

Restriction: You can edit only custom commands (commands you created). You cannot edit system commands.

To edit a custom command:

1. **Tools menu** > **Customize** > **Commands tab**.

2. In the Categories list, click a menu or toolbar name to display items (commands) specific to that menu or toolbar. Or click **All Commands**.
3. In the Commands list, select a command and click the **Edit** button.
4. In the Edit commands dialog box, change the properties of your custom command:
    a. In the Command name box, type the name of the command.
    b. In the Arguments box, type arguments for the command. Use a space to separate arguments. If an argument contains a space, enclose the argument in quotation marks (““).
    c. In the Description box, type a description of the custom command.
    d. If an image was associated with the original command, select **Use Default Image** to use that same image with your custom command. Select **Select a User-Defined Image** to use a different image, edit an image, or create a new image.
    e. Click **OK** to close the Edit commands dialog box and return to the Customize dialog box.
5. When you are finished with all customizations, click **Close**.

Click **OK** to close the Edit commands dialog box and return to the Customize dialog.

Deleting a Custom Command

Restriction: You can delete only custom commands (commands you created). You cannot delete system commands.

To delete a command:

1. **Tools menu** > **Customize** > **Commands tab**.
2. In the Categories list, click a menu or toolbar name to display items (commands) specific to that menu or toolbar. Or click **All Commands**.
3. In the Commands list, select a command and click the **Delete** button.
4. Click **Close**.

Creating a Custom Menu

To create a custom menu, you first create a new empty menu and then add items (commands) to it:

To create the new menu:

1. **Tools menu** > **Customize** > **Commands tab**.

2. In the Categories list, select **New Menu**.
3. In the Commands list, select **New Menu and** drag it to the location you want.
    - To create a top-level menu, drag the new menu to the Menu Bar.
    - To create a submenu, drag it over an existing menu name.
4. Click your new menu to select it. Then right-click and select **Button Appearance**.
5. In the Button text field, type the name for the menu and click **OK**. Leave the Customize dialog box open.
6. To add items (commands) to your new menu, click the **Commands tab**.
7. In the Categories list, select a menu or toolbar name to display commands specific to that menu or toolbar. Or select **All Commands**.

    Restriction: If you are working in a special mode in PADS Layout or PADS Logic (for example, the Decal Editor in PADS Layout), some categories of commands are not available for customization.
8. In the Commands list, select the command you want and drag it to the menu.
9. When you have finished adding commands, click **Close**.

Controlling Toolbar and Menu Content

You can modify the items on Toolbars and Menus.

- Adding Items to Toolbars and Menus
- Moving Items on Toolbars and Menus
- Removing Items from Toolbars and Menus

Adding Items to Toolbars and Menus

To add a button to a toolbar or an item to a menu:

1. **Tools menu** > **Customize** > **Commands tab**.
2. In the Categories list, select a toolbar or menu name to display commands specific to that menu or toolbar. Or select **All Commands**.

    Restriction: If you are working in a special mode in PADS Layout or PADS Logic (for example, the Decal Editor in PADS Layout), some categories of commands are not available for customization.
3. In the Commands list, select the command you want and drag it to the toolbar or menu.

Tip: To remove an item from a toolbar or menu (while the Customize dialog box is open), click the item and drag it outside the toolbar or menu.

4. When you have finished adding commands, click **Close**.

Related Topics

Creating a Custom Command

Creating Commands from Macro Command Files

Removing Items from Toolbars and Menus

Moving Items on Toolbars and Menus

Moving Items on Toolbars and Menus

You can rearrange items on a menu or buttons on a toolbar. You can also move or copy an item from one menu or toolbar to another.

Moving Buttons on Toolbars

The method you use for moving toolbar buttons depends on whether the Customize dialog box is open:

If the Customize dialog box is open:

- Click the toolbar button and drag it to a new place on the same toolbar or to a different toolbar.

  Tip: Instead of moving a button, you can copy it and move the copy. Press and hold the **Ctrl** key while dragging the button.

If the Customize dialog box is closed:

- Press and hold the **Alt** key. Drag the toolbar button to a new place on the same toolbar or to a different toolbar.

Moving Items on Menus

Restriction: To move menu items, the Customize dialog box must be open.

To move a menu item:

1. **Tools menu** > **Customize**.
2. In the main window of the PADS tool, display the menu containing the item you want to move.
3. Click the menu item and drag it to its new location on the same menu or to a different menu.

Tip: Instead of moving a menu item, you can copy it and move the copy. Press and hold the **Ctrl** key while dragging the item.

4. Click **Close**.

Related Topics

- Customizing Commands and Menus
- Adding Items to Toolbars and Menus
- Resetting Toolbars to Defaults

Removing Items from Toolbars and Menus

You can remove a menu item or toolbar button. The method to use depends on whether the Customize dialog box is open.

If the Customize dialog box is open:

- Drag the item outside the toolbar or menu. Then close the Customize dialog box.

If the Customize dialog box is closed:

- Press and hold **Alt**. Then drag the item outside the toolbar or menu.

Tip: You can reset a toolbar or shortcut menu back to its default list of items. See Resetting Toolbars to Defaults.

Related Topics

Adding Items to Toolbars and Menus

Customizing Shortcut Keys

You can create and customize shortcut keys by using the Keyboard and Mouse tab of the Customize dialog box (**Tools menu** > **Customize** > **Keyboard and Mouse tab**).

- To generate a report of available shortcut keys, see Listing Available Shortcut Keys.
- To create a shortcut key or reassign an existing shortcut, see Creating a New Shortcut Key.
- To assign a shortcut key to a macro command, Assigning Shortcut Keys to Macros.
- To delete an existing shortcut key, see Deleting a Shortcut Key.
- To reset all shortcut keys to their default settings, see Resetting Default Shortcut Keys.

Creating a New Shortcut Key

You create shortcuts that apply in any mode. Thus, the same shortcut key may have different functionality depending on the mode in which you are working.

To create a shortcut key:

1. **Tools menu** > **Customize** > **Keyboard and Mouse tab**.
2. In the Mode box, select the mode to which you want to apply the shortcut. The available commands for that mode appear in the Commands box.
3. In the Commands box, select the command for which you want to create a new shortcut. If a shortcut already exists, it appears in the Current shortcuts box.

   Tip: To replace an existing shortcut, click **Delete** to remove the existing shortcut, and create a new shortcut for the command.
4. Above the Current shortcuts box, click the **New** button to open the Assign shortcut dialog box.
5. Select one of the following types of shortcut:
   - To assign shortcut keys, select **Press new shortcut key(s)**, and then press the keys that you want to use. For detailed information about rules and restrictions for creating shortcut keys, see Rules and Restrictions for Key Sequences.

     Tip: As you enter the new shortcut, similar shortcuts appear in the Similar shortcuts assigned to other commands box. This helps you to avoid creating a new shortcut that conflicts with an existing shortcut.
   - To create a mouse action, select **or select a pointer event**, and then select a combination of list box options, mouse button events, and modifier keys.
6. Click **OK** to close the Assign shortcut dialog box.

   Result: The new shortcut appears in the Current shortcuts box on the Customize dialog box.

Rules and Restrictions for Key Sequences

The first character may consist of the following, plus Alt, Ctrl, or Shift modifiers:

- All printable characters including Space and Tab
- All function keys
- Extended keys: Up, Down, Left, Right, Insert, Delete, Home, PageUp, PageDown, End
- Numerical keypad keys (when Num Lock is off): Up, Down, Left, Right, Insert, Home, PageUp, PageDown, Del, End, /, *, +, -

- Mouse pointer events: Click, Double-click, RotateForward, RotateBackward

  Restriction: Mouse pointer events cannot be combined with key sequences, although the Ctrl, Alt, and Shift modifiers are allowed.

Subsequent characters may consist of the following:

- Alphanumeric (a-z0-9)

Exception: Some combinations, like Alt+Tab, are intercepted by Windows and thus are not available.

Related Topics

Customizing Commands and Menus

Resetting Default Shortcut Keys

Listing Available Shortcut Keys

Resetting Default Shortcut Keys

Listing Available Shortcut Keys

You can create a table of commands and the shortcuts assigned to them in an HTML file, letting you share the information over the Web with other members of the design team.

To create an HTML file containing the available shortcut keys:

1. **Tools menu** > **Customize** > **Keyboard and Mouse tab**.
2. Click **Report** and then select or type the HTML file name, then click **Save**.

Result: A hyperlink to the file appears in the Output window, under the Status tab.

Related Topics

Customizing Commands and Menus

Creating a New Shortcut Key

Resetting Default Shortcut Keys

Resetting Default Shortcut Keys

Expressions in Shortcut Keys

You can substitute a regular expression for characters in shortcut key command arguments, see Table 3-5.

Table 3-5. Expressions in Shortcut Keys

Expression	**Use to**
*	Match any number of characters.
?	Match any one character.
[set]	Match any character in the specified set. **Tip:** A set is composed of characters or ranges. A range has the form: Character Hyphen Character, such as A-Z or 0-9. The minimum set of characters supported in a set consists of [0-9a-zA-Z_].
[!set] or [^set]	Match any character not in the specified set.
\	To suppress the special syntactic significance of the characters ` [ ] * ? ! ^ - \ ' within a set, and to match the character exactly.

The following table shows examples of regular expressions used in command arguments using the preview.pcb design, see Table 3-6.

Table 3-6. Shortcut Key Expression Examples

Shortcut key	**Result**
H A*	Highlights all nets starting with A, such as A00, A01, A02.
H +??	Highlights all nets starting with +, having two digits or characters after 0, such as +5V.
H A?0	Highlights all nets starting with A, ending with 0, and with any character in between, such as A00 and A10.
H [C-D]*	Highlights all nets starting with C or D, such as CLKIN, D00.
H [!C-D]*	Highlights all nets not starting with C or D, such as A00, GND.

Deleting a Shortcut Key

Delete shortcuts you no longer want to use, or as the first step to changing an existing shortcut.

To delete a shortcut:

1. **Tools menu** > **Customize** > **Keyboard and Mouse tab**.
2. In the Mode box, select the mode for the shortcut you want to delete. The available commands for that mode appear in the Commands box.
3. In the Commands list, select the command whose shortcut you want to delete.
4. In the Current shortcuts list, select the shortcut you want to delete.
5. Click the **Delete** button.

Related Topics

Creating a New Shortcut Key

Resetting Default Shortcut Keys

Resetting Default Shortcut Keys

You can restore all shortcut keys to the default settings.

1. **Tools menu** > **Customize** > **Keyboard and Mouse tab**.
2. Click **Reset All**.
3. On the confirmation dialog box, click **Yes**.

Assigning Shortcut Keys to Macros

You can create a shortcut key that executes a macro.

Tip: In order to assign a macro to a shortcut key, the macro command file (.mcr) must already exist. You can create a macro by recording it in a PADS tool or scripting it in Macro language. For more information, see Creating Macros.

To assign a shortcut key to a macro:

1. **Tools menu** > **Customize** > **Macro Files tab**.
2. In the Macro Command Files area, click the **New** button.
3. In the Open macro file dialog box, select the macro file you want. Then click **Open**. The PADS tool loads the macro and makes it available for use as a command (the check box to the left of the macro name is selected).

   Tip: To close the macro file or make it unavailable in the Customize dialog box, clear the check box next to the macro name.
4. To assign the macro to a shortcut key, click the **Keyboard and Mouse tab**.

5. In the Mode list, select **All modes**.
6. In the Commands area, double-click **Macros** to display a list of available macros. Then select the macro you want.
7. In the Current Shortcuts area, click the **New** button. The PADS tool displays the Assign shortcut dialog box.
8. Select one of the following types of shortcut:
   - To assign shortcut keys, select **Press new shortcut key(s)**, and then press the keys that you want to use. For detailed information about rules and restrictions for creating shortcut keys, see Rules and Restrictions for Key Sequences.

     Tip: As you enter the new shortcut, similar shortcuts appear in the Similar shortcuts assigned to other commands box. This helps you to avoid creating a new shortcut that conflicts with an existing shortcut.
   - To create a mouse action, select **or select a pointer event**, and then select a combination of list box options, mouse button events, and modifier keys.
9. Click **OK** to close the Assign shortcut dialog box.

   Result: The new shortcut appears in the Current shortcuts box on the Customize dialog box.

Creating Commands from Macro Command Files

You can create commands from macro files and add them to toolbars and menus.

Tip: To create a command from a macro command file, the macro command file (.mcr) must already exist. You can create a macro by recording it in a PADS tool or scripting it in Macro language. For more information, see Creating Macros.

To make a macro file available:

1. **Tools menu** > **Customize** > **Macro Files tab**.
2. Click the **New** button.
3. In the Open macro file dialog box, select the macro file you want to use as a command. Then click **Open**. the PADS tool loads the macro and makes it available for use as a command (the check box to the left of the macro name is selected).

   Tip: To close the macro file, or make it unavailable in the Customize dialog box, clear the check box next to the macro name.
4. You can use the macro as you would any other command, for example, to create a Commands list for the Macros category.)

Related Topics

Adding Items to Toolbars and Menus

Adding a Macro to a Menu

Adding a Macro to a Menu

To add a macro to a menu, you create the macro and add the macro command as an item on a menu or a button on a toolbar.

Tip: To add macro to a menu, the macro command file (.mcr) must already exist. You can create a macro by recording it in a PADS tool or scripting it in Macro language. For more information, see Creating Macros.

To add a macro to a menu:

1. **Tools menu** > **Customize** > **Macro Files tab**.
2. In the Macro Command Files area, click the **New** button.
3. In the Open macro file dialog box, select the macro file you want to use as a command. Then click **Open**. The PADS tool loads the macro and makes it available for use as a command (the check box to the left of the macro name is selected).

   Tip: To close the macro file or make it unavailable in the Customize dialog box, clear the check box next to the macro name.
4. To add the macro to the menu, click the **Commands tab**.
5. From the Categories list, select **Macros**.
6. In the Commands list, select the macro and drag it to the menu.
7. When you finish adding macros, click **Close**.

Tip: You can use the same steps to add macros to shortcut menus and toolbars.

Customizing the Appearance of the Screen

You can customize the PADS interface by changing the appearance of menus and toolbars. Use the Options tab of the Customize dialog box (**Tools menu** > **Customize** > **Options tab**)

To customize display of toolbars:

- To display ToolTips for toolbar buttons, select the **Show ToolTips on toolbars** check box.
- To display shortcut key information in ToolTips, select the **Show shortcuts in ToolTips** check box.

- To display large icons, select the **Large Icons** check box.

Tip: To show or hide a toolbar in the PADS interface, use the Toolbars and Menus tab. See Showing or Hiding a Toolbar.

To customize display of menus:

- To change the way a menu is displayed, select an animation from the **Menu animations** list. For example, Unfold displays part of the menu and an arrow you can click to display the rest.
- To show shadows on menu items, select the **Menu shadows** check box.
- To display your recent menu selections before other menu items, select the **Show recent commands first** check box.
- If menus display recent commands first, you can display the full list of menu items after a delay. Select the **Show full menus after delay** check box.
- If you have a shortcut made up of a number of keys, you can delay its execution until you press the Enter key. Select the **Wait until enter before executing long shortcut** check box.
- To delete the record of commands you've used and restore the default set of commands to the menus and toolbars, click **Reset my usage data**. This option does not undo any explicit customizations you made.

Resizing the Layers List

To change the width of the Layers list on the Standard toolbar:

1. **Tools menu** > **Customize**.
2. On the Standard toolbar of the PADS Interface, select the **Layers list** box.
3. Resize as needed.
4. Click **Close**.

Restriction: You cannot use the Alt key to resize the Layers list.

Organizing Windows

You can customize the way windows appear in your workspace in the following ways:

- Showing Windows
- Hiding Windows
- Detaching Windows from the Current View

- Attaching Windows to the Current View
- Embedding Windows within Other Windows

Showing Windows

When you first start the application, several windows display. You can show, hide, and automatically hide any of the windows in the application.

To open windows:

- On the View menu, click the name of the window you want to show.

Your choices may include Output Window, Project Explorer, and Shortcut Dialog.

Hiding Windows

When the application opens, several windows are open in addition to your workspace. You can close some of these windows or hide them automatically in order to maximize your design space.

Closing Windows

To close a visible window:

1. Move your pointer to the title bar of the window you want to hide.
2. Click the small downward pointing arrow on the right side of the window's title bar.
3. In the resulting menu, click **Hide**.

Result: The window closes.

Hiding Windows Automatically

You can also set a window to hide automatically so that it appears when you hover the pointer near it, and automatically minimizes when you move the pointer away from it.

To hide a window automatically:

1. Move your pointer to the right side of the title bar in the window you want to hide.
2. Click the thumbtack in the window's title bar.

   Result: The thumbtack picture changes to point sideways . A new bar appears on the side of the interface. The side on which the bar appears depends on the location of the window. For example, if the Project Explorer is located on the left side of the user interface, when you click the Auto Hide setting from the menu, the new bar appears on the left side of the interface.

The new bar contains a tab that has the same name as the window.

3. Hover over the tab in the new bar.

   Result: The window reappears, covering the application.

4. Move the pointer away from the window.

   Result: The window minimizes to a tab.

Tip: To turn off the Auto Hide feature, hover over the tab in the new bar so the window reappears. Then repeat steps 1-2 in reverse.

Detaching Windows from the Current View

You can detach a window from the current view. This is called floating. A floating window is not attached to the current view; instead, it hovers, blocking the view to anything below it.

Restriction: You cannot float a window that is currently set to hide automatically. Turn off the Auto Hide feature before floating a window.

To float a window:

- Double-click the window's title bar.

Result: The window detaches and you can move it to any part of the screen.

Tip: To undo the floating, see Attaching Windows to the Current View.

Attaching Windows to the Current View

You can attach a window to the current view. This is called docking. A docked window is attached to the current view, and therefore does not block the view to anything below it. You can dock a window in its last docked location, or dock a window to a different location.

Docking to the Last Location

To dock a window to its last docked location:

- Double-click the window's title bar.

Result: The window reattaches to the interface.

Docking to a New Location

To dock a window to a new location:

1. Using the title bar, drag the window.

Result: When you start dragging the window, additional graphics appear in the user interface. At the edges of the user interface, arrows containing graphics appear, as shown in Figure 3-1:

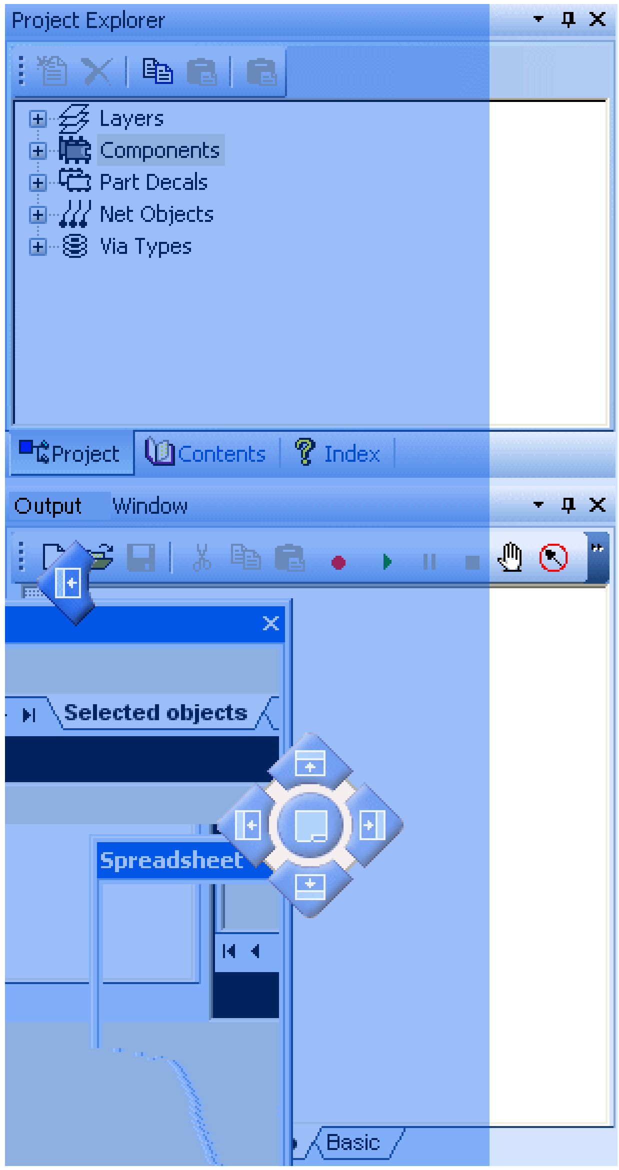
Figure 3-1. Window Dragging Graphic

Tip: A similar group of arrows appears in a group near the center of the screen. Ignore that group of arrows for this procedure.

2. While dragging the window, hover over one of the arrows on the edge of the user interface. For example, hover over the arrow on the left side of the user interface.

   Result: A transparent colored block appears along the side of the user interface to which you are pointing. This block indicates where the window will be docked when you release the mouse button. For example, if you hover over the arrow on the left side of the user interface, a block appears along the left side of the screen, as shown in Figure 3-2.

Figure 3-2. Docking a Window



3. Release the mouse button while hovering over the arrow that indicates where you want to dock the window.

   Result: The window docks to the user interface, and the other windows in the user interface resize.

Embedding Windows within Other Windows

In addition to attaching a window to a side of the user interface, you can embed a window within another window, so that it shares the window space with the original window, or becomes a tab within the original window.

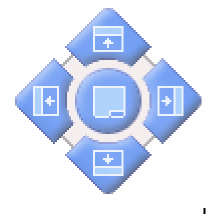
Two Windows Sharing One Window Space

To embed a window within another window's space:

1. Using the title bar, drag a window into another window.

   Result: When you start dragging a window, additional graphics appear in the user interface. A group of arrows containing graphics appears in the center of the window you are dragging, as shown in Figure 3-3. Depending on the window you are dragging, the group of arrows may also have a tab graphic in the center.

   Figure 3-3. Dragging a Window—Arrow Group

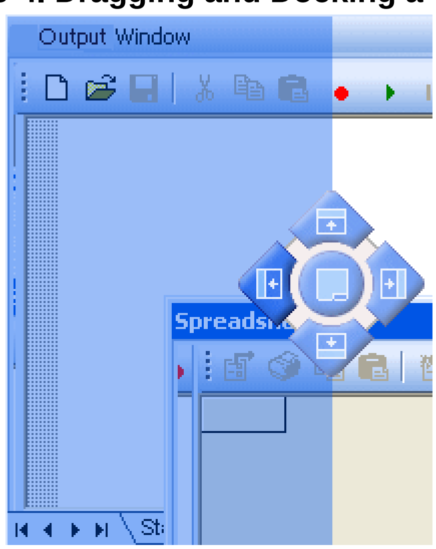
   

   Tip: A similar group of arrows appears at the sides of the user interface. Ignore those arrows for this procedure.

2. While dragging the window, hover over one of the arrows. For example, hover over the left arrow.

   Result: A transparent colored block appears along the side of the window you are dragging, as shown in Figure 3-4. This block indicates where the window will be docked when you release the mouse button. For example, if you hover over the left arrow, a block appears along the left side of the Project Explorer.

   Figure 3-4. Dragging and Docking a Window

3. Release the mouse button while hovering over the arrow that indicates where you want to dock the window.

   Result: The window is embedded within another window, both sharing the space the original window occupied.

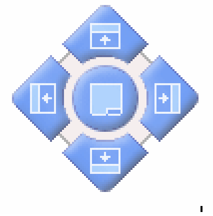
Tip: To maximize your workspace, try setting these embedded windows to hide automatically. Ctrl+click the thumbtack in one of the window's title bars, and all of the windows within the original window frame hide automatically.

Creating Tabs within Windows

1. Using the title bar, drag a window into another window.

   Result: When you start dragging a window, additional graphics appear in the user interface. A group of arrows containing graphics appears in the center of the window you are dragging, as shown in Figure 3-5. Depending on the window you are dragging, the group of arrows may also have a tab graphic in the center.

   Figure 3-5. Dragging and Docking a Window—Arrow Commands

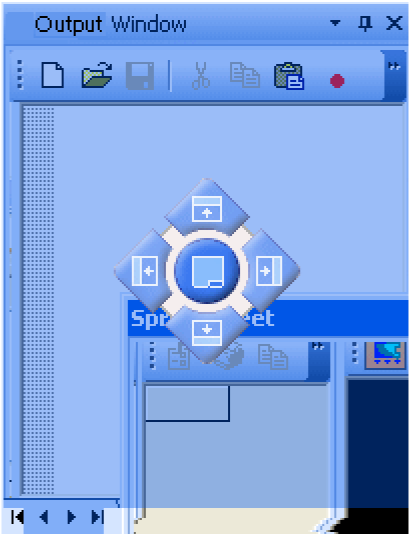
   

   Tip: A similar group of arrows appears at the sides of the user interface. Ignore those arrows for this procedure.

2. While dragging the window, hover over the tab graphic.

   Result: A transparent colored block appears over the window you are dragging, as shown in Figure 3-6. This block indicates where the window will be docked when you release the mouse button. For example, if you hover over the tab in the Project Explorer window, a block appears over the Project Explorer.

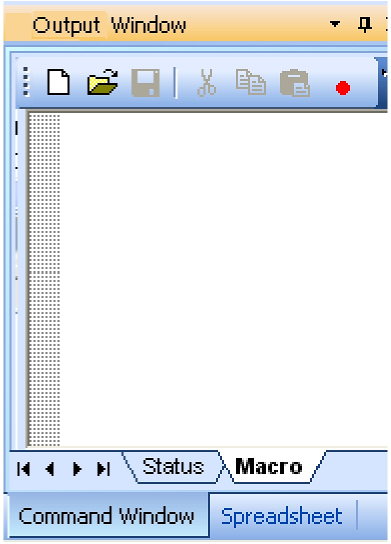
Figure 3-6. Dragging a Window—Transparent Block



3. Release the mouse button while hovering over the tab.

   Result: The window is embedded as a tab within a window, as shown in Figure 3-7. You can click each tab to access each window.

Figure 3-7. Window Embedded as a Tab



Tip: To maximize your workspace, try setting these embedded windows to hide automatically. Ctrl+click the thumbtack in one of the window's title bars, and all of the windows within the original window frame hide automatically.

Managing Window Tabs

Some of the windows in the user interface contain tabs. However, you may decide you do not like the organization or grouping of the tabs. You can create additional tabs by embedding windows within other windows.

See also: Embedding Windows within Other Windows

This topic discusses the following:

- Rearranging Tabs in a Window
- Moving Tabs Between Windows
- Converting Tabs to Windows

Rearranging Tabs in a Window

Restriction: You can only rearrange tabs that you created by embedding a window within one of the docking windows. You cannot rearrange regular tabs, such as those in the Output window.

To change the order of the tabs in a window:

- Drag the tab to a new position within the row of tabs.

Moving Tabs Between Windows

Restriction: You can only move tabs that you have embedded in other windows. In windows that have tabs by default such as the Output Window, you cannot move the tabs. You can only rearrange them. For information on rearranging tabs, see the previous section.

See also: Embedding Windows within Other Windows

To move tabs:

1. Drag the tab to a new window.

   Result: When you start dragging, the tab automatically behaves like a window.

2. Place the tab as you would a window.

See also: Organizing Windows

Converting Tabs to Windows

To create a new window from a tab:

1. Drag the tab.

Result: When you start dragging, the tab automatically behaves like a window.

2. Release the mouse button. Make sure the pointer is not over any arrow graphics.

   Result: You now have a floating window.

3. Place the tab as you would any floating window.

See also: Organizing Windows

Chapter 4
Managing Libraries and Library Data

In this section you learn how to create and modify libraries, set searchability, manage library attributes, import and export libraries, report on contents, and use search parameters.

- Converting Older PADS Libraries to the Current Format
- Creating a Library
- Displaying Items in a Library
- Modifying Library Data
- Setting Library Availability and Search Options
- Managing Library Attributes
- Transferring Library Data
- Importing and Exporting Libraries
- Creating Library Reports
- Wildcards and Expressions

The parts, decals and other items you use to lay out a design in PADS reside in one or more PADS *libraries*. A single PADS library consists of 4 files, each containing items of a specific type identified by the file's extension, as follows:

Table 4-1. PADS Library Files

File Extension	File Contents
.pt	Part Types
.pd	PCB Decals
.ld	CAE Decals
.ln	Line graphics

For information on creating CAE decals, see Creating a New CAE Decal in the *PADS Logic User's Guide*. For information on creating line graphics, see Adding Drafting Items from a Library in this manual.

Converting Older PADS Libraries to the Current Format

For information on how to convert your older PADS libraries to the current PADS format, see Converting Older PADS Libraries in the *PADS Library Converter User's Guide*.

Creating a Library

You create a new library to start a new empty library. You can also add an existing library - for more information, see Adding Libraries to the Library List.

Procedure

1. **File** menu > **Library**.
2. Click **Create New Lib**.
3. In the New Library dialog box, specify the folder and library file name, and then click **Save**.
4. Click **Close**.

Result

Your library is added to the bottom of the Library list which is also the last library in the search order. To move your library up in the library list and search order, see Setting the Library List Order.

Displaying Items in a Library

You can display the items of a library in the Library Manager dialog box.

Procedure

1. **File** menu > **Library**.
2. In the Library Manager dialog box, select a library in the Library list or select *All Libraries*.
3. Click one of these buttons to display categories of items in the library:
    - Decals—PCB decals (component footprints)
    - Parts—Parts
    - Lines—Drafting objects

- Logic—CAE decals (schematic symbols)
  Tip: Use PADS Logic to edit CAE decals

4. To filter the list, type wildcards or expressions in the Filter box, and then click **Apply**.

   Tip: An empty filter box yields no results. If you don't want to restrict the results with a filter, but display all items, type * (asterisk) and click Apply.

Result

Library item names are displayed in the PCB Decals, Part Types, Line Items, or CAE Decals list. (The list name changes based on the Filter you've selected.) The preview window displays a graphic of the library object.

Restriction: Since library Parts have no visual representation, the preview window displays the first PCB decal associated with the part.

Related Topics

Creating a Report of the Parts in a Library

Creating a Report of Decals, Lines or Logic Symbols in a Library

Modifying Library Data

In this section:

- Adding Items to a Library
- Deleting Items from a Library
- Copying a Library Item
- Editing Items in a Library
- Deleting All Items in a Library
- Transferring Library Data

Adding Items to a Library

You can add new items to a library using the Library Manager dialog box.

Procedure

1. **File** menu > **Library**.
2. In the Library Manager dialog box, select a library in the Library list.

   Restriction: If you select (All Libraries), the New button is unavailable.

3. Click one of these buttons to display categories of items in the library:
    - Decals—PCB decals (component footprints)
    - Parts—Parts
    - Lines—Drafting objects
    - Logic—CAE decals (schematic symbols). Unavailable. Use PADS Logic to add schematic decals to a library.
4. Click the **New** button.
    - Decals—Opens the PCB Decal Editor on a new decal.
      See also: Creating a New Decal
    - Parts—Opens the Part Information dialog box on an unnamed part.
      See also: Creating a New Part Type
    - Lines—New and Edit are unavailable. There is no special library lines editor. Use drafting tools to create or edit lines and save them to the library.
      See also: Creating a Drafting Object, Saving a Drafting Item to a Library
    - Logic—New is unavailable. Use PADS Logic to create CAE decals.
      See also: Creating a New CAE Decal

Related Topics

Copying a Library Item

Deleting Items from a Library

You can delete one or more selected items from a library.

Procedure

1. **File** menu > **Library**.
2. In the Library Manager dialog box, select a library in the Library list.

   Restriction: If you select (All Libraries), the Delete button is unavailable.
3. Click the one of these buttons to display categories of items in the library:
    - Decals—PCB decals (component footprints)
    - Parts—Parts
    - Lines—Drafting objects
    - Logic—CAE decals (schematic symbols)
4. To filter the list, type wildcards or expressions in the Filter box, and then click **Apply**.

Tip: An empty filter box yields no results. If you don't want to restrict the results with a filter, but display all items, type * (asterisk) and click Apply.

5. Select one or more items in the PCB Decals, Part Types, Line Items, or CAE Decals list. (The list name changes based on the Filter you've selected.)

   Tip: Use Ctrl+click to select multiple non-sequential items. Use Shift+click or drag the cursor to select a range of items.

6. Click **Delete**.

Related Topics

Deleting All Items in a Library

Copying a Library Item

You can copy a selected item to another name or another library.

Procedure

1. **File** menu > **Library**.
2. In the Library Manager dialog box, select a library in the Library list.

   Restriction: If you select (All Libraries), the Copy button is unavailable.

3. Click one of these buttons to display categories of items in the library:
   - Decals—PCB decals (component footprints)
   - Parts—Parts
   - Lines—Drafting objects
   - Logic—CAE decals (schematic symbols)
4. To filter the list, type wildcards or expressions in the Filter box, and then click **Apply**.

   Tip: An empty filter box yields no results. If you don't want to restrict the results with a filter, but display all items, type * (asterisk) and click Apply.

5. Click **Copy.**
6. In the dialog box that opens, select another library to receive the item, and/or type a new item name, and then click OK.
   - Decals—Opens the Save PCB Decal to Library dialog box
   - Parts—Opens the Save Part Type to Library dialog box
   - Lines—Opens the Save Drafting Item to Library dialog box

- Logic—Opens the Save CAE Decal to Library dialog box

Related Topics

Importing and Exporting Libraries

Editing Items in a Library

You can use the Library Manager to edit items in a library.

Procedure

1. **File** menu > **Library**.
2. In the Library Manager dialog box, select a library in the Library list.

   Restriction: If you select (All Libraries), the Edit button is unavailable.
3. Click the one of these buttons to display categories of items in the library:
   - Decals—PCB decals (component footprints)
   - Parts—Parts
   - Lines—Drafting objects
   - Logic—CAE decals (schematic symbols)
4. To filter the list, type wildcards or expressions in the Filter box, and then click **Apply**.

   Tip: An empty filter box yields no results. If you don't want to restrict the results with a filter, but display all items, type * (asterisk) and click Apply.
5. Click **Edit**.
   - Decals—Opens the PCB Decal Editor with the selected decal.
     See Editing a Library Decal
   - Parts—Opens the Part Information dialog box on a selected part.
     See Creating and Modifying Part Types
   - Lines—Unavailable. There is no special library lines editor. Use drafting tools to create or edit lines and save them to the library.
     See Creating a Drafting Object, Saving a Drafting Item to a Library
   - Logic—Unavailable. Use PADS Logic to create CAE decals.
     See Creating a New CAE Decal

Deleting All Items in a Library

When you delete all items in a library using the following method, you actually replace the existing library with a new empty one of the same name. This ensures that there aren't any leftover items in your library.

Restriction: You cannot delete the contents of a read-only library.

Procedure

1. **File** menu > **Library**.
2. Click **Create New Lib**. The New Library dialog box appears.
3. Select the library file whose contents you want to delete, click **Save**, and then click **Yes** when prompted.
4. Click **Close**.

Related Topics

Deleting Items from a Library

Transferring Library Data

You can transfer library objects from one library to another.

Tip: To copy over single items, see Copying a Library Item.

Procedure - Multiple Items

1. Follow the steps to export library data from the library.
2. Then follow the steps to import the library data to another library.

Setting Library Availability and Search Options

Use the Library List dialog box to specify the libraries available to the design, library search order, and other search-related options. Operations in this dialog box affect the contents of the Library list in the Library Manager dialog box.

In this section:

- Adding Libraries to the Library List
- Removing Libraries from the Library List
- Setting the Library List Order
- Sharing a Library Across a Network

- Controlling Library Search Access
- Protecting Library Files
- Sychronizing with PADS Logic

Adding Libraries to the Library List

You add libraries to the library list in order to make their contents available. The library must be listed in the Library Manager in order to search it and use its parts, decals, or line items in your design.

Procedure

1. **File** menu > **Library** > **Manage Lib. List** button.
2. In the Library List dialog box, click **Add**.
3. In the Add Library dialog box, specify the folder and file name of the library to add, and then click **Open**.

Result

The library is added beneath the currently selected library in the Library list.

Removing Libraries from the Library List

You remove libraries from the library list in order to prevent their contents from being used in the design.

Procedure

1. **File** menu > **Library** > **Manage Lib. List** button.
2. In the Library List dialog box, in the Library list, select one or more libraries, and then click **Remove**.

Tip: The library files are not deleted from the computer.

Setting the Library List Order

You can specify the order in which libraries are searched.

Procedure

1. **File** menu > **Library** > **Manage Lib. List** button.
2. In the Library List dialog box, in the Library list, select the library, and then click **Up** or **Down** as needed.

Result

With each click, the library moves up or down one place in the library list. The libraries are searched top to bottom.

Related Topics

Library Search Order in the *Concepts Guide*

Sharing a Library Across a Network

You can share a library across a networked environment, to allow more than one user to simultaneously access the library.

Procedure

1. **File** menu > **Library** > **Manage Lib. List** button.
2. In the Library List dialog box, in the Library list, select the library.
   Tip: You can select multiple libraries using the Shift and Ctrl keys.
3. Select the **Shared** check box.

Result

More than one user can access the library file at the same time.

Controlling Library Search Access

You can enable or disable the searching of a particular library when performing operations that involve libraries, such as adding parts.

Procedure

1. **File** menu > **Library** > **Manage Lib. List** button.
2. In the Library List dialog box, in the Library list, select the library.
   Tip: You can select multiple libraries using the Shift and Ctrl keys.
3. Select the **Allow Search** check box.

Protecting Library Files

The Read Only check box is only a status indicator. It is always shaded and unavailable. You can set library read-only status only in the Microsoft Windows File Manager. The system administrator who owns the files is the only one who can control this process; this ensures file protection.

Procedure

1. **In Windows Explorer, locate your library files.**

   Tip: By default, libraries installed with the software are located at C:\MentorGraphics\<version>\SDD_HOME\Libraries

2. Select all four library files, right-click and click **Properties**.
3. In the Properties dialog box, on the General tab, select the **Read-only** check box.
4. Click **OK**.

Result

The library Read-Only check box in the Library List dialog box will not update until you reopen the dialog box.

Related Topics

Creating a Library

Sychronizing with PADS Logic

You can enable or disable the synchronizing of library settings between PADS Logic and PADS Layout. When the Synchronize with PADS Logic check box is checked in PADS Layout, all changes made in the libraries within PADS Layout are pushed to PADS Logic.
Tip: To ensure a round-trip synchronization, select the same check box in PADS Logic.

Procedure

1. **File** menu > **Library** > **Manage Lib. List** button.
2. In the Library List dialog box, select the **Synchronize with PADS Logic** check box.

Managing Library Attributes

Use the Manage Library Attributes dialog box to manage attributes on a library-by-library basis. You can add, delete, and rename attributes for all parts or decals in an individual library or in all libraries. You can also display all the attributes in a library, whether the attributes apply to all items or to individual items.

In this section:

- Adding an Attribute to Multiple Library Items
- Deleting Attributes from Library Items
- Renaming Attributes of Library Items

Tip: The Manage Library Attributes dialog box does not manage attributes in the design. Use the Attribute Dictionary to manage attributes in the design. For more information, see Using the Attribute Dictionary

Adding an Attribute to Multiple Library Items

You can add an attribute to all parts and decals, or just to parts or decals individually, in one or all libraries.

Restriction

This process will yield no result or warning if the library you are working with is read-only. Check the status of the library in the Library List dialog box.

Procedure

1. **File** menu > **Library** > Library Manager dialog box > **Attr Manager** button.
2. In the Manage Library Attributes dialog box, in the Select Library list, select an individual library or **(All Libraries)**.
3. In the Item Types list, choose whether to apply the new attribute to All Items, or only either Part Types or PCB Decals.
4. Click **Add Attr**. The Add New Attribute to Library dialog box appears.
5. For the Attribute Name, either:
    - Type an attribute name into the box.

      or
    - Click **Browse Lib. Attr** to search all libraries for an existing attribute name.
6. **Optional:** Type a value in the Attribute Value box.
7. Click **OK**. The Attribute Name appears in the Attributes in Library list.
8. Click **Close**.

Result

Your new attribute is added. Check for the new attribute in the Decal Attributes dialog box (for PCB decals) or on the Attributes tab of the Part Information dialog box (for Part Types).

Related Topics

Setting Library Availability and Search Options

Deleting Attributes from Library Items

You can delete one or more attributes from all parts and decals, or just from parts or decals individually, in one or all libraries.

Restriction

This process will yield no result or warning if the library you are working with is read-only. Check the status of the library in the Library List dialog box.

Procedure

1. **File** menu > **Library** > Library Manager dialog box > **Attr Manager** button.
2. In the Manage Library Attributes dialog box, in the Select Library list, select an individual library or **(All Libraries)**.
3. In the Item Types list, choose whether to delete the attribute(s) from All Items, or from either Part Types or PCB Decals.
4. In the Attributes in Library list, select one or more attributes to delete, and then click **Delete Attrs**.
5. In the prompt that appears, click **OK**.
6. Click **Close**.

Result

Your attribute(s) is deleted. Check for the deleted attribute in the Decal Attributes dialog box (for PCB decals) or on the Attributes tab of the Part Information dialog box (for Part Types).

Renaming Attributes of Library Items

You can rename an attributes of all parts and decals, or just of parts or decals individually, in one or all libraries.

Restriction

This process will yield no result or warning if the library you are working with is read-only. Check the status of the library in the Library List dialog box.

Procedure

1. **File** menu > **Library** > Library Manager dialog box > **Attr Manager** button.
2. In the Manage Library Attributes dialog box, in the Select Library list, select an individual library or **(All Libraries)**.

3. In the Item Types list, choose whether to rename the attribute(s) from All Items, or from either Part Types or PCB Decals.

4. In the Attributes in Library list, select one or more attributes to rename, and then click **Add**.

5. Double-click in the New Name cell, type the name, and then click **Rename Attrs**.

   Tips:

   - To display all existing attributes in the library, click Browse Lib. Attr.
   - You can specify the name of an existing attribute. No error message appears when you do this. The only time this may have an adverse effect is if both attributes are assigned to a single item, in which case the error is reported in the error file and the rename is not performed for those items where there are conflicts.

6. Click **Close**.

Result

Your attribute is renamed. Check for the renamed attribute in the Decal Attributes dialog box (for PCB decals) or on the Attributes tab of the Part Information dialog box (for Part Types).

Importing and Exporting Libraries

Use the Library Manager dialog box to import or export library data in ASCII format.

In this section:

- Importing Library Data
- Exporting Library Data

Importing Library Data

You can import library data from a previously-exported library ASCII file.

Tip: Beginning with PADS 9.0, die parts and flip chips are no longer identified by their family designations (DIE or FLP), but instead by the Special Purpose settings in the General tab of the Part Information dialog box. When you import an ASCII file created by a previous PADS version, these Special Purpose settings are automatically set for parts having the logic family DIE or FLP. The part’s family designation remains the same.

Procedure

1. **File** menu > **Library**.

2. In the Library Manager dialog box, in the Library list, select the library to receive the library data.
3. To import one of the four file types, you must select the matching filter:
   - If the file type is .c, select the Logic filter. This ASCII file contains CAE decals (logic symbols).
   - If the file type is .l, select the Lines filter. This ASCII file contains drafting objects.
   - If the file type is .d, select the Decals filter. This ASCII file contains PCB decals (component footprints).
   - If the file type is .p, select the Parts filter. This ASCII file contains part types.
4. Click **Import**.

   Tip: Import fails if the library to receive imported items is read-only.
5. In the Library Import File dialog box, specify the folder and the file name, and then click **Open**.

Related Topics

Transferring Library Data

Exporting Library Data

You can export library data into an ASCII file for importing into a library.

-
- Processing Decals (Cells) with Library Services
-

Procedure

1. **File** menu > Library.
2. In the Library Manager dialog box, in the Library list, select the library whose data you want to export.
3. Click any of the following:
   - Decals—to export PCB decals (component footprints)
   - Parts—to export Components
   - Lines—to export drafting objects

- Logic—to export CAE decals (schematic symbols)

4. To filter the list, type wildcards or expressions in the Filter box, and then click **Apply**.

   Tip: To display all items in the library, type asterisk * and click Apply.

5. Select one or more items in the list, and then click **Export**.
6. In the Library Export File dialog box, specify the folder, type the file name, and then click **Save**.

Results

- The Special Purpose settings of any die parts and flip chips are cleared.
- Die parts and flip chips having a family designation other than DIE and FLP lose their die part or flip chip special purpose and become normal parts.
- Any normal parts that have the DIE or FLP family designation are treated as die parts or flip chips in the previous PADS version.

Related Topics

Creating a Library

Creating Library Reports

You can create reports from the Library Manager to list any number of library objects. The Parts report can be configured to also list the values of attributes that you choose to include in the report.

In this section:

Creating a Report of the Parts in a Library

From the Library Manager, you can generate a report about the parts in a single library or all libraries. The report (an ASCII file) lists each part and its associated attributes. You can specify the attributes you want reported. To see examples of parts reports, see Figure 4-1and Figure 4-2.

Procedure

1. Select **File** menu > **Library**.
2. In the Library manager dialog box, select a library from the Library list. Or select **All Libraries**.

3. In the Filter area, click **Parts**. A list of parts in the library (or in all libraries) appears in the Part Types area.

   Tip: To refine the list, use the filter field. Type a part name in the field or use wildcards (*) to specify a group of parts. Then click **Apply**.

4. When you have the list of parts you want to report on, click **List to File**.
5. In the Report Manager dialog box, specify the part attributes you want to include in the report. In the Available attributes list, click an attribute to select it. The click **Include>>**. The attributes appear in the Selected attributes list.

   To remove attributes from the Selected attributes list, select them and click **Exclude>>**.

6. Optionally, you can refine the list of parts to report on. In the Part Filter field, type a part name in the field or use wildcards (*) to specify a group of parts. Then click **Apply**.
7. Click **Run**.
8. In the Library List File dialog box, select a folder and file format for the report. You can select either of two formats:
   - Library List format (lst): Information is formatted in columns for viewing or printing. (See Figure 4-1.)
   - Comma-separated values format (csv): format recognized by MS Excel. (See Figure 4-2.)
9. Click **Save**.
10. In the Report Manager dialog box, click **Close**.

Result

The Report Manager generates the report and displays a link to it in the Output window. To view or print the report, click the link. Notepad opens and displays the report.

Figure 4-1. Parts Report in .lst Format

```
      PADS LIBRARY (anlogdev Part Types) DIRECTORY LISTING


Library: anlogdev
---------------------------------------------------------------------------------
|Part Name |Part Number   |Description                               |Manufacturer #
---------------------------------------------------------------------------------
|AD1315    |AD1315KZ      |HIGH SPEED ACTIVE LOAD WITH INHIBIT       |ANALOG DEVICES
|AD1321    |AD1321KZ      |HIGH SPEED PIN DRIVER WITH INHIBIT        |ANALOG DEVICES
|AD1322    |AD1322KZ      |ULTRAHIGH SPEED PIN DRIVER WITH INHIBIT   |ANALOG DEVICES
|AD1376    |AD1376 (J,K)D |HIGH SPEED, 16-BIT A/D CONVERTER          |ANALOG DEVICES
|AD1377    |AD1377 (J,K)D |HIGH SPEED, 16-BIT A/D CONVERTER          |ANALOG DEVICES
|AD1378    |AD1378 (S,T)D |WIDE TEMPERATURE, 16-BIT A/D CONVERTER    |ANALOG DEVICES
|AD1380    |AD1380 (J,K)D |16-BIT SAMPLING ADC                       |ANALOG DEVICES
|AD1382    |AD1382KD      |16-BIT, 500KHZ, SAMPLING ADC              |ANALOG DEVICES
```

Figure 4-2. Parts Report in .csv Format

```
"Library","Part Name","Part Number","Description","Manufacturer #1","PCB Decal 1",
"anlogdev","AD1315","AD1315KZ","HIGH SPEED ACTIVE LOAD WITH INHIBIT","ANALOG DEVICES"
"anlogdev","AD1321","AD1321KZ","HIGH SPEED PIN DRIVER WITH INHIBIT","ANALOG DEVICES",
"anlogdev","AD1322","AD1322KZ","ULTRAHIGH SPEED PIN DRIVER WITH INHIBIT","ANALOG DEVI
"anlogdev","AD1376","AD1376 (J,K)D","HIGH SPEED, 16-BIT A/D CONVERTER","ANALOG DEVICE
"anlogdev","AD1377","AD1377 (J,K)D","HIGH SPEED, 16-BIT A/D CONVERTER","ANALOG DEVICE
"anlogdev","AD1378","AD1378 (S,T)D","WIDE TEMPERATURE, 16-BIT A/D CONVERTER","ANALOG
"anlogdev","AD1380","AD1380 (J,K)D","16-BIT SAMPLING ADC","ANALOG DEVICES","DH-32E",
"anlogdev","AD1382","AD1382KD","16-BIT, 500KHZ, SAMPLING ADC","ANALOG DEVICES","DH-48
```

Related Topics

Report Manager Dialog Box

Creating a Report of Decals, Lines or Logic Symbols in a Library

Managing Libraries and Library Data

Creating a Report of Decals, Lines or Logic Symbols in a Library

From the Library Manager, you can generate a report listing the decals, lines, or logic symbols in a single library. The report is an ASCII file listing each item's name and the date and time the item was modified. For more information, see Creating a Report of the Parts in a Library.

Procedure

1. Select **File** menu > **Library**.
2. In the Library manager dialog box, select a library from the Library list.

   Restriction: The List to File button is unavailable for Decals, Lines, and Logic Symbols if you select All Libraries.
3. In the Filter area, click the filter you want. A list of parts in the library appears in the Part Types area.

   Tip: To select one or more specific item, use the filter field. Type an item name in the field or use wildcards (*) to specify a group of items. Then click **Apply**.
4. When you have the list you want to report on, click **List to File**.
5. In the Library List File dialog box, specify a folder and file name for the report and click **Save**.

Result

Notepad appears, displaying a list of the item names and the date and time when each was last modified. You can print the list from Notepad.

Related Topics

Creating a Report of the Parts in a Library

Managing Libraries and Library Data

Wildcards and Expressions

You can use wildcards and expressions to filter the information that is displayed. The filter supports the wildcards and expressions listed in Table 4-2. Table 4-3 gives examples of wildcard usage.

Table 4-2. Wildcards and Expressions

Expression:	Use to:
*	Match any number of characters.
?	Match any one character.
[set]	Match any character in the specified set. **Tip:** A set is composed of characters or a range of characters; for example, A-Z or 0-9 or a-z.
[!set] or [^set]	Match any character not in the specified set.
\	Match a special syntactic character exactly, suppressing the special character's syntactic significance. **Tip:** The following characters need the \ before them: `[]*?!^-\'

Table 4-3. Usage Examples of Wildcards and Expressions

Expression:	Results in all items that:
74*	start with 74: 7404, 74LS04, 74622.
74??	start with 74 followed by any two characters: 7404, 74T2, 74TP.
74??08	start with 74, followed by any two characters, and end with 08: 74LS08, 74HC08, 744608.
*08	start with any number of characters and end with 08: 2146108, 5408, 54HCT08, 744608.
08	start with any number of characters, followed by 08, and end with any number of characters: 5408, 5408BE, 54HCT08AE, 74ABT08CE2, 941M70839.
[57]*	start with 5 or 7 with any number of characters after: 54HCT244, 5968BAE4, 74ACT44.
[5-7]*	start with 5 or 6 or 7 followed by any number of characters: 54LS08, 6225BE, 69TF77, 74ALS02.

Table 4-3. Usage Examples of Wildcards and Expressions (cont.)

Expression:	Results in all items that:
[57]4HCT??	start with 5 or 7, followed by 4HCT, and end with any two characters: 54HCT04, 54HCT74, 74HCT27, 74HCT84.
74A[CH]*	start with 74A, followed by C or H, and end with any number of characters: 74AC244, 74AHCT27.
74A[!C-H]*	start with 74A, followed by any character except the letters C through H, and end with any number of characters: 74ABT44, 74ALS244, 74ABF365.
[\\]*08	start with the character \, followed by any number of characters, and end with 08: \LS08, \HCT08, \ABT08.

Chapter 5
Creating and Editing PCB Decals

A PCB decal is a software representation of the outline, terminals and attributes of an electrical component, and is used in placement and routing of the component during the design layout process.

You use the PCB Decal Editor to create new decals or to modify existing ones.

In this section:

- Setting Up the PCB Decal Editor
- Setting Colors of Objects in the Decal Editor
- Creating a New Decal
- Editing a Decal
- Decal Editing Tasks
- Working with Solder and Paste Masks
- Updating a Design from the Library

Setting Up the PCB Decal Editor

You can modify the Decal Editor’s layers, colors, rules, editor options and Decal Wizard options.

Procedure

1. Change the layer stackup in the Layers Setup dialog box.
   (**Setup** menu > **Layer Definition)**

   Tip: If you need to add objects to specific layers, etc., do so.

2. Change the colors of objects or layers in the Display Colors Setup dialog box.
   (**Setup** menu > **Display Colors**)

   Tip: As you work on a decal, you may find it useful to apply different colors to objects or layers to visually differentiate them.

   Restriction: You can modify the colors of objects and layers in the decal editor, but when the decal is used by a component in the design, it uses the colors set for the design.

3. Change the options in the Options dialog box.
   (**Tools** menu > **Options**)

4. Change the decal rules in the Decal Rules dialog box.
   (**Setup** menu > **Decal Rules**)

   Restriction: Rules set at this level of the rules hierarchy are used only by PADS Router.

Creating a New Default Decal Editing Environment

There is a default (start-up) decal editing environment that opens when you start the Decal Editor from the Tools menu. This decal editing environment has its own layers, colors, rules, and editor options settings. These settings are very generic in the out-of-box default environment.

You can modify and save the default environment's settings to create a new default that is more appropriate for the decals you typically make. This will make it easier and quicker for you to create new decals.

Procedure

1. **Tools** menu **>** **PCB Decal Editor**. The default decal opens in the Editor.
2. Set options and settings appropriately for the decals you typically make. **See Setting Up the PCB Decal Editor.**
3. **File** menu **> Save as Start-up File.**
4. In the Start-up File Output dialog box, select the appropriate check boxes in the Sections area, and click **OK**.

Result

When you immediately click **File** > **New**, you will see the new default settings.

Setting Colors of Objects in the Decal Editor

As you work on a decal, you can customize display colors to make it easier to see objects as you place them. Using the Display Colors Setup dialog box, you can:

- Set and change the color of objects on a per layer or per object type basis.
- Make objects visible or invisible in the display (also on a per layer or per object type basis).
- Customize the palette of color selections.
- Save your customizations in a configuration file.

Procedure

1. **Decal Editor Setup** menu > **Display Colors**.
2. In the Display Colors Setup Dialog Box, set the color(s) you want:
    a. To set the color for one object, in the **Selected Color** area, click a color tile. Then in the Layers/Object Types table, click the tile for the object type in the correct layer row.
    Tip: To change the palette of colors from which you can select, click **Palette**.
    See also: To Change the Color Palette

    b. **To make all objects on a layer the same color,** click the layer number to select the entire row, and then click the color tile in the Selected Color area.

    c. **To make an object the same color on all layers,** click the object name to select the entire column, and then click the color tile in the Selected Color area.

    d. **To make an object type invisible,** set its tile to the background color. You can make multiple objects invisible (for example, all objects on the same layer).
    See also: Making Objects Invisible in the Display.

    Tip: To make an object type the same color on all layers, click the **Assign All** button and use the Assign Color to All Layers dialog box.
3. Click **Apply**.

Tips:

- You can save display color settings you commonly use as you work on a design. See To Save Color Assignments to a File.
- Color configurations saved in PowerPCB versions 2.1 and lower cannot be used in PowerPCB versions 3.0 and higher.

Related Topics

Creating a New Default Decal Editing Environment

Creating a New Decal

You can create a decal automatically, using the Decal Wizard, or manually, using the buttons on the Drafting toolbar of the PCB Decal Editor. See Creating a Basic Decal Automatically and Creating a Decal Manually for information on how to perform these tasks.

Creating a Basic Decal Automatically

Use the Decal Wizard to automatically create common decal types based on settings you provide in the Decal Wizard dialog box. To display the Decal Wizard dialog box, click

Tools menu > **PCB Decal Editor** > **Drafting Toolbar** button > **Wizard** button

There are two main methods to using the decal wizard. In order to create a decal, you need to begin with the package dimensions and enlarge the decal to the appropriate material condition.

Method 1

You manually calculate the decal dimensions required for the package and enter the dimensions into the decal wizard to create the new decal.

Method 2

You enter the package dimensions into the Decal Calculator area of the dialog box (where it's available) and let the Decal Calculator generate the decal according to preset standards of material conditions.

After you create the basic decal, you can edit it to add other items, such as labels, attributes, keepouts, solder mask relief, height values, and hard breakouts/fanouts. See Decal Editing Tasks.

Creating a Decal Manually

If you can't create a decal using the decal wizard, you can create it manually using the Decal Editor Drafting Toolbar buttons. The general procedure for creating a decal manually is as follows:

1. **Tools** menu > **PCB Decal Editor (Opens the default (start-up) decal.)**

   Alternative: You don't have to start with the default decal when creating a new decal. If the new decal you want to create will be similar to an existing decal, you can edit that decal and save it under a new name to create the new decal.

2. Set the decal origin.
3. Add and place the terminals.
4. Create any copper shapes, such as copper associated with terminal pads.
5. Define the terminal pad stacks.
6. Create the Assembly, Silkscreen and Placement part outlines.
7. Add other items, such as labels, attributes, keepouts, solder mask relief, height values, and hard breakouts/fanouts. See Decal Editing Tasks.

   Tips:

   - You cannot edit the default "Name" and "Type" labels that appear when you create a new decal; these "placeholder" labels are automatically populated with the correct reference designator and part type name, respectively, when a part using the decal is added to a design.

- You can add other “placeholder” labels to a decal. See Creating Placeholder Attribute Labels.

8. Using **File menu > Save As**, name the new decal and save it to a library.

Editing a Decal

You can edit a library decal, or the decal of a component in a design.

Editing a Library Decal

To edit a library decal:

1. **Tools menu > PCB Decal Editor**
2. **Click the Open button.** The Get PCB Decal from Library dialog box opens.
3. From the Library list in the Filter area, select the library that contains the decal.
4. If you want to filter the PCB Decals list, type a wildcard or expression in the Items box and click **Apply**.

   Requirement: The Items box must contain at least the asterisk (*) to get any search results. An asterisk displays all parts in the list.
5. Select the decal from the **PCB Decals** list. The preview area displays a graphic of the decal.
6. Click **OK**.

   Tip: If the decal is assigned to any part types in the PADS libraries, the Part Type List for Decal <name> dialog box opens.
7. Edit the decal. (See Decal Editing Tasks for information on how to edit a PCB decal.)
8. Click **File menu > Save** to save the decal.

Checking for Errors Between Part Types and Assigned PCB Decals

For the decal you have opened, in the Part Type List for Decal dialog box, you can check to ensure that the pins match the pins listed in the Part Type > Pins tab table, or when pin mapping exists - on the Part Type > Pin Mapping tab.

- Alternatively, in the PCB Decal Editor, on the Tools menu, click **Part Types** to open the Part Type List for Decal dialog box.

Sorting the Table

You can sort the table by a given column by clicking the column header.

Fixing Logical Errors

1. Select a line item with a logical error.
2. Click the **Show Errors** button to create and open a report with details of the errors.
3. Click the **Edit Part** button to open the Part Type.
4. In the part type, fix the errors and then click **OK**.
5. The Part Type List for Decal dialog box will update the Error Status.

Fixing Mismatched Pins Errors

1. Select a line item with a mismatched pin error.
2. Click the **Show Errors** button to create and open a report with details of the errors.
3. In the PCB Decal Editor, correct the error.
4. In the Part Type List for Decal dialog box, click **Refresh**, to update the dialog box.

Tip: You can click the Check Decal button to check the decal against all associated part types to show all mismatched pin number errors.

Related Topics

Part Information Dialog Box, Pins Tab

Part Information Dialog Box, Pin Mapping Tab

Editing the Decal of a Component in the Design

You can edit the decal used by one or more components selected in a design, and apply the changes to those components alone, or to all the components in the design using that decal.

You can also save the changes to the existing decal in the library, or create a new decal.

To edit a design component decal from the design editor:

1. Select the component(s). The selected components must all use the same decal.
2. Right-click and select Edit Decal to open the PCB Decal Editor.
3. Edit the decal. (See Decal Editing Tasks for information on how to edit a PCB decal.)
4. If you want to save changes to the library:
    i. **File menu** > **Save Decal As**

ii. In the Save PCB Decal to Library dialog box, set the Library and Name of PCB Decal fields to save the changes to a new decal. If you want to save the changes to the existing decal, leave the fields as they are.

iii. Click OK.

5. **File menu > Exit Decal Editor**
6. In the Apply Decal Changes dialog box, Click Selected to apply the changes to the selected components only, or All to apply the changes to all components using that decal in the design.

Tip: If:

- You have a design open in default layer mode, and
- You open the Decal Editor to edit the decal of a component in the design, and
- You attempt to change the decal to increased layer mode,

the message "You will not be able to apply decal changes to the design. Continue?" appears.

Either click Cancel to return to the Layers Setup Dialog box without changing the decal to increased layer mode, or press OK to proceed to the Increase Maximum Layer Number dialog box, where you can save the decal changes to the library; then exit the PCB Decal Editor without applying changes. Then switch the current design to increased layer mode and update the decal.

Decal Editing Tasks

Editing the Properties of a Decal Item

You can edit the properties of decal items, including terminals, labels, text and 2D items.

Procedure

1. In the Decal Editor, select an item, right-click, and click **Properties**.

   Tip: You can also Double-click on the item, or select the item and click the **Properties** button on the toolbar.

2. In the displayed dialog box, edit the decal name properties.
3. Click **OK**.

Setting the Decal Origin

The location of a decal's origin affects how decals attach to the cursor when you move a component in the Move by Origin mode. It also affects how pick and place machines populate the board with components. Typically, through-hole devices have their origin at Pin 1, and surface-mount devices at the geometric center of the component; other devices may have other origin requirements.

When you're creating a new decal, you can define the origin whenever it's most convenient. For example, if you're creating a through-hole device decal:

- You could create Pin 1 on the default origin, and build the rest of the decal around that.
- You could build the entire decal anywhere, and then move the origin to Pin 1.

From the Decal Editor, to set the origin of a decal:

1. **Setup** menu > **Set Origin**
2. Click where you want the new origin to be.

You can also set the origin of a decal after it has been created.

1. Select the decal.
2. Type the modeless command **SO <x> <y>**.
   Tip: Type SO for relative coordinates, SOA for absolute coordinates.

Working with Terminals

In this section:

- Adding Terminals
- Assigning JEDEC Pinning
- Modifying Terminal Properties
- Modifying Terminal Number Properties
- Moving a Terminal
- Moving a Terminal Number

Adding Terminals

Terminals are the pads or pins associated with a decal.

Procedure

1. **PCB Decal Editor** > **Drafting** button > **Terminal** button.
2. In the Add Terminals dialog box, in the Start pin number area, type values in the Prefix and/or Suffix boxes for the pin numbering. A preview of pin numbers based on your input is displayed below the boxes.

Tips:

- Alphabetic and numeric values can be used in either box. For example, A1 or 1A.
- For a single numeric, use either Prefix or Suffix box, and void the other box.

3. In the Increment options area, choose what to increment by clicking either **Increment prefix** or **Increment suffix**.
4. In the Step value box, type a positive or negative number by which to increase or decrease the pin numbers with consecutive or stepped values.
5. If using alphanumerics, you can select the **Use JEDEC pin numbering** check box to ensure that legal alphanumeric values are used.

   Tip: This option only ensures that valid alpha and numeric combinations are used. To arrange rows and columns according to JEDEC, use the Assign JEDEC Pinning option on the Tools Menu.
6. Click to indicate a position for the new terminal. Repeat as necessary.

Tip: You can also add terminals by replicating an existing terminal using Step and Repeat.

See also: Creating Copper in the Decal, Associating Copper with Terminals

Using Step and Repeat

1. Select an existing object, right click and select Step and Repeat.
2. In the Step and Repeat dialog box, click the tab of the type of array to create: Linear, Polar, or Radial, and set the options for the array.
3. If the object selection contains text or a terminal, you can also automatically increment the text or pin number. See also: Incrementing Texts and Pin Numbering
4. Click **OK** to create the array.

Tips:

- Terminals are replicated with associated coppers.
- Settings in the Step and Repeat tabs are saved when you close the dialog box and when you exit the Decal Editor.

Assigning JEDEC Pinning

You can assign pin numbers to rows and columns of terminals according to the JEDEC standard.

Procedure

1. **PCB Decal Editor** > **Open** button.

2. To filter the decals, in the Filter area, type a wildcard or expression in the Items box and click **Apply**.
3. Click to indicate the decal to which you want to assign JEDEC pinning and click **OK**.
4. Click **Assign JEDEC Array Pinning** from the **Tools** menu. The JEDEC Array Pinning dialog box appears.
5. Click the Decal type: **Component** or **Substrate**.
6. Click **OK**.
7. On the File menu, click **Save**.

Modifying Terminal Properties

Procedure

1. **PCB Decal Editor** > select a terminal > right-click > **Properties**.
2. Modify any of the following information:
   - Terminal x,y coordinates.
   - Terminal pin number. You can change the pin number of the selected terminal.
3. Click **Pad Stacks** to open the Pad Stack Properties dialog box. Use this dialog box to modify one or more selected pad stacks.
4. You can clear the **Associated Copper** check box to unassociate copper from the terminal.
5. Click **Apply** to apply your modifications or **Cancel** to cancel the changes.

If you select another terminal while the dialog box is open, the information is updated for the selected terminal.

Tip: You can use any alphanumeric characters except
brackets { }, asterisks *, commas (,), question marks (?), or spaces.

Related Topics

To Align Objects

Customizing Pad Stacks of Decal Pins

Modifying Terminal Number Properties

1. **PCB Decal Editor** > select a terminal number > right-click > **Properties**.
2. Modify any of the following information:
   - Coordinates—You can also move the terminal number using coordinates.

- Pin Number—You can change the pin number.

If you select another terminal number while the dialog box is open the information is updated for the selected terminal number.

Moving a Terminal

Procedure

1. In the PCB Decal Editor, select the terminal to move. You can select more than one at a time.
2. Drag the terminal. The terminal remains attached to the pointer.
3. Click to complete the move.

   You can also use the arrow keys to control movement: each press of an arrow key moves the selected terminal to the next point on the grid. Using the Terminal Properties dialog box, you can type a new X,Y location.

You can also use Radial Move to manually place decal terminals.

See also: To Use Radial Move

Moving a Terminal Number

Procedure

1. In the PCB Decal Editor, select the terminal number to move. You can select more than one at a time.
2. Drag the terminal number and release the mouse button. The terminal number dynamically attaches to the pointer.
3. Click to indicate a new location to complete the move.

You can also use the arrow keys: each press of the arrow key moves the selected terminal number to the next point on the grid. Using the Terminal Number Properties dialog box, you can type a new X,Y location.

Swapping Terminal Numbers

Procedure

1. In the PCB Decal Editor, select the terminals you want to swap.
2. Right-click and click **Swap**. The terminal numbers switch.

Renumbering a Terminal

You can renumber terminals in the PCB Decal Editor by clicking each pin in the decal, or using the Pin Number dialog box.

In the decal editor, you can renumber a group of terminals in ascending order.

Renumbering by Clicking Terminals

In the decal editor, you can renumber terminals in ascending order by selecting the starting terminal and clicking consecutive terminals.

Procedure

1. **PCB Decal Editor** > select a starting terminal > right-click > **Renumber Terminals**.

   Tip: Renumber Terminals is not available if more than one terminal is selected.

2. In the Start pin number area, type values in the Prefix and/or Suffix boxes. A preview of pin numbers based on your input is displayed below the boxes.

   Tips:

   - Alphabetic and numeric values can be used in either box. For example, A1 or 1A.
   - For a single numeric, use either Prefix or Suffix box, and clear the other box.

3. In the Increment options area, choose what to increment by clicking either **Increment prefix** or **Increment suffix**.

4. In the Step value box, type a positive or negative number by which to increase or decrease the pin numbers with consecutive or stepped values.

5. If using alphanumerics, you can select the **Use JEDEC pin numbering** check box to ensure that legal alphanumeric values are used.

   Tip: This option only ensures that legal alphanumeric combinations are used. To arrange rows and columns according to JEDEC, use the Assign JEDEC Pinning option on the Tools Menu.

6. Click **OK**.

   Result: The terminal is highlighted, and the pointer changes to a small cross. The pointer dynamically attaches to the renumbered terminal. The message "Renumbering… Next New Number: #" also attaches to the pointer.

   Tip: If you have duplicate numbers during the renumbering process, the duplicate numbers appear with a tilde (~) next to them.

7. Select another terminal to assign the next available number.

Tip: If you have renamed all available pins, the message “Renumbering... New Numbers Exhausted!” appears. In other words, the message appears if you renumber all eight terminals on a decal with eight pins.

8. You can back up in the numbering process. Right-click and click **Back** to undo the last assigned number.

9. Right-click and click **Complete** when you finish reassigning terminal numbers.

Renumbering Using the Pin Numbers Dialog Box

You can use the Pin Numbers dialog box to interactively renumber the terminals in the design area. Selecting pin numbers in the dialog box selects the matching pins in the design area and selecting pins in the design area selects the matching pins in the dialog box.

- On the Setup menu, click **Pin Numbers**.

You can modify the pin numbers listed in the dialog box, by editing individual cells, by copying and pasting, or by using the Renumber Pins dialog box.

Tip: Pin number changes on the decal will not be updated until you click Apply or OK.

Editing Individual Pin Numbers

Procedure

1. Double click a Number cell, or select a number cell and click the **Edit** button.
2. Type a new pin number.

Copying and Pasting Pin Numbers

You can copy selected table data from the Pin Numbers dialog or from Microsoft Excel and paste it into the Numbers list. The selected cell in the table is the paste origin. Data is pasted below the paste origin.

Procedure

1. In Excel, select data and use the Excel Copy command in Excel. Or in the Pin Numbers dialog, select data and click the Pin Numbers dialog box **Copy** button.

2. Select the cells into which you want to paste the data.

   Restriction: Data will only paste into selected cells.

3. Click the **Paste** button to paste the data into the table starting at the paste origin.

Editing Pin Numbers Using the Renumber Pins dialog box

You can efficiently edit the pin numbers of numerous pins using the Renumber Pins dialog box.

See also: Renumbering by Clicking Terminals

Deleting a Terminal

Procedure

1. In the PCB Decal Editor, select the terminal to delete. You can select more than one at a time.
2. Press **Delete**.

The remaining terminals are automatically renumbered. The alphanumeric labels remain unchanged.

Moving a Decal Name

Procedure

1. In the PCB Decal Editor, select the decal name to move.
2. Drag the decal and release the mouse button. The decal name dynamically attaches to the pointer.
3. Click to indicate a new location to complete the move.

You can also use the arrow keys to control movement: each press of an arrow key moves the decal to the next point on the grid.

Related Topics

To Rotate an Object

Creating Copper in the Decal

You can create copper in the decal and associate it with a pin to create a custom pad shape or heat sink. You can also create unassociated copper heat sink or shielding shapes that move with the decal in the design.

Requirement

- You must select a layer for placement of the copper. You cannot create a copper shape on all layers (layer number zero). If you need the same copper shape on multiple layers, copy the shape to the other layers.

Procedure

1. Click the **Drafting Toolbar** button.
2. On the Drafting toolbar, select the **Copper** button.

3. Select a layer on which to place the copper.
4. Right-click and click a command to change the values of the drafting object. See also: Setting Values Before Creating a Drafting Object.
5. Create the shape using one of the following:
    - Creating a Circle Drafting Object
    - Creating a Polygon or Path Drafting Object
    - Creating a Rectangle Drafting Object
6. Once you complete the shape, it becomes a filled shape.

Result

Is the resulting shape what you expected?

- If the shape edges are not correct, see Edge Precision of Drafting Shapes.
- If the shape is on the wrong layer or needs to be modified, see Modifying Drafting Object Properties.
- If you want to start over, see Deleting a Drafting Segment or Object.

Related Topics

Associating Copper with Terminals

Creating a Copper Cut Out

Generating Drafting Shapes from Terminals

Creating a Custom Pad Shape

You can create odd shaped (irregular shaped) pads by drawing a copper shape and associating it to the pad while editing the decal. For more information, see Associating Copper with Terminals.

Associating Copper with Terminals

You can associate a drawn copper shape with a terminal/pad. This assigns the copper shape the same net connection as the terminal/pad. You use associated copper to create custom pad shapes.

Restriction: Routing is only completed to the center of the pad and any thermal connections are made to the pad only and not the associated copper shape.

Procedure

1. Draw the copper shape. For more information, see Creating Copper in the Decal.

   Tip: You can create multiple copper shapes.

2. **PCB Decal Editor** > select a terminal > right-click > **Associate**.

3. Select the copper item to associate with the terminal. This electrically connects the copper item with the terminal. You may want to physically connect the terminal and copper item by overlaying part or all of the terminal.

4. If you are using the associated copper to create an odd shaped pad, maximize the pad shape inside the overlapping associated-copper shape. The underlying pad is still the routing target and also the connection point for thermals if this pad will ever be connected to a copper pour or plane area. If associated copper surrounds the terminal pin by a large amount it could prevent thermal spokes from being generated.

   Restriction: When routing to a decal pad with associated copper in the design editor, the trace is flagged as a partial connection if you route only to the associated copper, and not to the center of the decal pad itself. Note, however, that though the connection is *flagged* as partial, it will result in a functional connection on the finished board.

Result

CAM interprets terminals differently when associated with copper. **See** Working with Associated Copper.

Unassociating Copper from Terminals

You can remove the association between a copper item and a terminal.

Procedure

1. In the Decal Editor, select the copper item.

2. Right-click and click **Unassociate**.

   Alternative: In the Terminal Properties, you can clear the **Associated Copper** check box to unassociate copper from the terminal.

Customizing Pad Stacks of Decal Pins

You can customize the pad stacks of the pins in your decal. You can choose to customize one pin, or multiple pins simultaneously.

Restriction

- If you're working in the PCB Decal Editor, you must have a decal open in the editor. The terminals (pins) must exist in your decal before you can modify their default pad stacks.
- If you're working in the Design Editor, the decal you want to edit must exist in the design.

Procedure

1. On the **Setup** menu, click **Pad Stacks**.
2. In the Pad Stack Properties dialog box, ensure that your decal is selected in the Decal name list.
3. Under the *Pin: Plated:* list, select the pin to customize. If you don't want to customize all the pins simultaneously, you can add one or more specific pin numbers to the list.

   See also: Add Pin Dialog Box
4. Under the *Sh: Sz: Layer: list, select the layer of the pad you want to customize.*

   See also: Pad Stack Default Layers, Control of Solder Mask and Paste Mask
5. In the Parameters area, specify the settings for all three **Pad**, **Thermal**, and **Antipad** pad styles if needed. The style of pad used is selected automatically in the design depending on the situation. The thermal and antipad styles are used when the pin is located within a plane.

   See also: Design Rule Versus Pad Stack - Thermals and Antipads, Creating Thermals in the Pad Stacks, Creating Antipads in the Pad Stacks, and Customizing Design Rule Thermals.
6. Below the Parameters area, set the options that apply to the pads on all layers for the selected pin(s).
7. If you need to customize other pins in the decal, return to step 3.
8. Click **OK** to save all the pad stack customizations.

Result

Are your custom thermals not showing up in CAM plane plots? You must select the **Use Custom Thermal Settings** check box in the Plot Options dialog box and ensure that the photo plotter format is RS-274-X. Custom thermals for CAM planes are not supported by RS-274-D.

Related Topics

Assigning Plane Thermal Attributes

Creating Thermals in the Pad Stacks

Thermals are set up wherever pad stacks can be created or modified. Create thermals on split/mixed plane layers and CAM negative planes (for RS-274-X output).

Restrictions:

- Pad stack thermals work only in plane areas; they do not work in copper pours.
- In the PCB Decal Editor, thermals can't be created in the Pad Stack Properties for Pin dialog box. Instead, on the Setup menu click Pad Stacks to use the Pad Stacks Properties dialog box.

Tip: The Sh: Sz: Layer: area of these dialog boxes contains one global listing for inner layers, listed as <Inner Layers>. Setting a thermal definition for this inner layer modifies the photoplot output for CAM planes and split/mixed planes. If you want a unique thermal on a particular layer, add the new layer, and define the specific thermal setting.

Procedure

1. Select a layer.
2. In the Pad Style list, select **Thermal**.
3. Click the appropriate pad shape to enable the size and shape controls for thermals.

   Tip: The pad stack thermal settings use oval and rectangular shapes for slotted holes as well as for oval and rectangular pads.

4. Change the default thermal values in the fields provided.

   Restriction: No pad stack thermal will be used if the Use design rules for thermals and antipads check box is selected in the Split/Mixed Plane options. See also Customizing Design Rule Thermals.

Exception: CAM planes are negative images compared to other layers. When you set thermals for CAM plane layers, you must select the Use Custom Thermal Settings check box in the Plot Options dialog box. Make sure you are also using RS-274-X output format in the Photo Plotter Advanced Setup dialog box.

Related Topics

Customizing Pad Stacks of Decal Pins

Creating Antipads in the Pad Stacks

Antipads are set up wherever pad stacks can be created or modified. Create antipads on split/mixed plane layers and CAM negative planes (for RS-274-X output).

Restrictions:

- Pad stack antipads work only in plane areas; they do not work in copper pours.
- In the PCB Decal Editor, antipads can't be created in the Pad Stack Properties for Pin dialog box. Instead, on the Setup menu click Pad Stacks to use the Pad Stacks Properties dialog box.

Tip: The Sh: Sz: Layer: area of these dialog boxes contains one global listing for inner layers, listed as <Inner Layers>. Setting an antipad definition for this inner layer modifies the photoplot output for CAM planes and split/mixed planes. If you want a unique antipad on a particular layer, add the new layer, and define the specific antipad setting.

Procedure

1. Select an inner layer.

   Tip: Antipads are not found when using the generic Mounted Side and Opposite side layers. If you want a custom antipad for the top and bottom layers, you need to add the specific layer.

2. In the Pad Style list, select **Antipad**.
3. Click the appropriate pad shape to enable the size and shape controls for antipads.

   Tip: The pad stack antipad settings use oval and rectangular shapes for slotted holes as well as for oval and rectangular pads.

4. Change the default antipad value in the field provided.

- **Restriction:** No pad stack antipad will be used if the Use design rules for thermals and antipads check box is selected in the Split/Mixed Plane options. See also Customizing Design Rule Antipads of Copper Pours and Customizing Design Rule Antipads of Plane Areas.

Exception: CAM planes are negative images compared to other layers. When you set antipads for CAM plane layers, you must select the Use Custom Thermal Settings check box in the Plot Options dialog box. Make sure you are also using RS-274-X output format in the Photo Plotter Advanced Setup dialog box.

Related Topics

Customizing Pad Stacks of Decal Pins

Editing Pad Stacks

Pad information for the selected decal is listed by pin number to the right of the Setup/Pad Stack form. A typical decal uses one pad stack description for all its pins. There are common exceptions, for example, pin 1 of through-hole components is usually designated by a square pad.

To edit a pad stack for a decal:

1. **Setup** menu > **Pad Stacks**.
2. Set the pad stack type to **Decal** or **Via**.
3. Click the decal or via name from the **Decal Name** list.
4. Make your changes.
5. To save your changes click **OK**.
6. When you are prompted to save changes to all or the selected parts click the appropriate button:

- **All** to assign the decal name to the part type and for all part types in the design using this decal to assume the new pad stack definition.
- **Selected** to rename the individual component's decal using a suffix letter; for example a DIP16 with a local pad stack edit becomes DIP16A. This decal name becomes the only decal listing, primary or alternate, for this particular reference designator.

See also: Customizing Pad Stacks of Decal Pins

Editing a Pad Stack in the PCB Decal Editor

Edit a pad stack in the PCB Decal Editor to change the pad stack of a single pin or to make pad stack changes and save them to the library before adding parts to the design.

To change pad stack definitions for selected pins:

1. **Tools** menu > **PCB Decal Editor**.
2. Open a decal.
3. Select the pin or pins for which you want to change the pad stack.
4. Right-click and click **Pad Stacks**. The Pad Stack Properties for Pin dialog box appears.
5. From **Pin Name**, click the pin you want to change, and from **Shape/Size/Layer**, click the layer on which you want to change the pad stack information.
6. Make the pad stack modifications.
7. Click **Assign to All Pins** to apply the modifications to all of the selected pins. Selected pins appear in the Pin Name list.
8. Click **OK**. The pad stack changes are applied to every selected pin.

Tips:

- To save the changes, save the decal.
- To work with pad stacks on library parts before they are added to the design enter the PCB Decal Editor through the Library Manager. This lets you include decal names in

the part's alternate decals, ensuring smoother operation if you refresh a part type in the design with a definition from the library.

See also: Part Information Dialog Box, PCB Decals Tab

Saving Pad Stack Changes to the Decal Library

Changing the pad stack definition for a single part or for all parts does not change the decal in the part's library definition. Use the Library Manager to include any new decal names in the part's alternate decals. This ensures a smoother operation if you ever refresh the part type in the design with a new definition from the library.

To save the pad stack changes to the parts library:

1. Select the part.
2. Right-click and click **Save to Library**. The Save Part Types and Decals dialog box shows the selected part type and all of its alternate decals as of the last time it was read or refreshed.

   If you created a new suffixed decal, meaning you changed the pad stack for one component only, the new decal also appears.
3. Highlight either the decal you changed or the new decal and the library where you keep the decals.

Using Slotted Holes

You can create slotted holes using the Pad Stacks Properties dialog box or the Pad Stacks Properties for Pins dialog box.

Slotted holes have orientation and offset properties, but have the same unit and range as the associated pad's orientation and offset. You can use slotted holes with only round, square, oval, or rectangular pad shapes. Therefore, you can only define slotted holes for component pins. All pads in the pad stack should be oval or rectangular; the pad shape is checked on the mounted side. You can also create a thermal or antipad definition for slotted holes.

Tip: Control the line width for slotted holes using Line Width in the Drill Drawing Options dialog box.

Related Topics

Creating Slotted Holes in Decals

Creating Slotted Holes in Pins

Using Slotted Holes in CAM350 in the *Concepts Guide*

Customizing Pad Stacks of Decal Pins

Pad Stack Properties for Pin Dialog Box

Setting Drill Drawing Options

Creating Slotted Holes in Decals

Procedure

1. **Tools** menu > **Decal Editor > Setup** menu > **Pad Stacks**.
2. In the Pad Stacks Properties dialog box, set any pad stack options you want, such as plated, size, shape, and layer.
3. Select the **Slotted** check box to add a slotted hole to the decal.
4. Set the length, orientation, and offset of the slotted hole.
5. Click **OK**.
6. Choose whether to keep existing attributes on the pin (**Keep Attributes**), and then choose whether to apply the pad stack changes to all decal types (**All**) or to just the selected component (**Selected**). You can also to cancel the slotted hole creation (**Cancel**).

Creating Slotted Holes in Pins

Procedure

1. Select a pin > right-click > **Properties**.
2. Click **Pad Stack**.
3. Select the **Slotted** check box to add a slotted hole.
4. Set the length, orientation, and offset of the slotted hole.
5. Click **OK**.
6. Choose whether to keep existing attributes on the pin (**Keep Attributes**), and then choose whether to apply the pad stack changes to all decal types (**All**) or to just the selected component (**Selected**). You can also choose to cancel the slotted hole creation (**Cancel**).

Creating Assembly Drawing Decal Objects

The assembly outline and assembly refes are used on the assembly drawing to define the exact outline and identify the body of the part. It shows where the part is to be placed during the board assembly process.

Procedure

1. **Tools menu > PCB Decal Editor**
2. On the Assembly Drawing Top layer, draw the outline.
   - **Drafting Toolbar** button > **2D line** button. See Creating a Drafting Object for information on drawing 2D shapes.

   Tip: If you flip a part to the other side of the board, the assembly outline flips with it. Even though you create the outline on the Assembly Top layer, it automatically flips to the Assembly Bottom layer when you flip the component from the mounted side to the opposite side.
3. Add a second reference designator label on the Assembly Outline Top layer. The reference designator label should be large and located *inside* the outline. For more comprehensive procedures than what is listed below, see Creating Attribute Labels in the PCB Decal Editor.
   a. Click the **Add New Label** button.
   b. In the Add New Decal Label dialog box, in the Attribute list, select the **Ref.Des.** attribute.
   c. In the Layer list, select the assembly drawing top layer.
   d. Click **OK**. The label appears with the value “Name”.
   e. Right-click and click **Move**.
   f. Position the new label.

   Tip: You can also create assembly layer reference designators globally in the design editor. See Generating a Second Set of Reference Designators for Assembly Drawings.

Result

If a 2D item does not appear after you draw it, the color set for Lines on the layer is probably the same as the background color. Set a different color for Lines in the Display Colors Setup dialog box.

Creating Silkscreen Decal Objects

The silkscreen outline is printed on the board, and remains visible on the board after the part has been placed. This outline, along with its reference designator, pin number and polarity labels, identifies the part and its pins to persons assembling, testing, or servicing the board. The silkscreen also serves as a nudge outline if no actual nudge outline exists on Layer 20.

Procedure

1. **Tools menu > PCB Decal Editor**

2. On the Silkscreen Top layer, draw the outline.

   - **Drafting Toolbar** button > **2D line** button. See Creating a Drafting Object for information on drawing 2D shapes.

   Tip: If you flip a part to the other side of the board, the silkscreen outline flips with it. Even though you create the outline on the Silkscreen Top layer, it automatically flips to the Silkscreen Bottom layer when you flip the component from the mounted side to the opposite side.

3. Move the silkscreen refdes to the Silkscreen Top layer. The silkscreen refdes label is automatically added at the creation of the new decal. But by default it is added to the Top (Mounted) layer.

   a. Double-click the "**Name**" label in the decal.

   b. In the Decal Label Properties dialog box, in the Layer list, select the silkscreen top layer.

   c. Click **OK**.

4. Add pin number, and polarity labels if needed. The labels should be located *outside* the outline, where they can be seen after the part is placed. For more comprehensive procedures than what is listed below, see Creating Attribute Labels in the PCB Decal Editor.

   Tip: Silkscreens for large components with many pins often have miniature dots and pin numbers, in order not to have to count the pins when trying to locate a pin on the board. If you use text in the decal for these, they will not be movable when the component is placed in the design. Use labels instead; labels can be moved or deleted in the design if necessary.

   a. Click the **Add New Label** button.

   b. In the Add New Decal Label dialog box, in the Attribute box, type a name for the new attribute.

      Restriction: The value can only be assigned once the decal is used in the design.

   c. In the Layer list, select the silkscreen top layer.

   d. Click **OK**. The label appears with the attribute name you typed.

   e. Right-click and click **Move**.

   f. Position the new placeholder.

Result

If an item does not appear after you add it, the color set for the item is probably the same as the background color. Set a different color in the Display Colors Setup dialog box.

Exceptions: The refdes (Name label) is controlled by the Pin Num column and the part type (Type label) is controlled by the Type column.

Creating a Placement (Nudge) Decal Outline

The placement (nudge) outline (sometimes called a "courtyard") defines the boundary where nudging of the part is initiated when another part is moved against it. Some parts need additional spacing compared to others to allow for machine-placement on the physical board. You can use a larger outline on layer 20 to ensure that your design placement is correct even though you apply the same body-to-body clearance for all components.

Procedure

1. **Tools menu > PCB Decal Editor**
2. On layer 20 (120 in max layers mode) draw the 2D line item(s) defining the nudge outline.
    - **Drafting Toolbar** button > **2D line** button. See Creating a Drafting Object for information on drawing 2D shapes.

      Tip: The outline doesn't have to be a contiguous shape enclosing the part. It can be made up of multiple 2d items--any object placed on the Placement layer is considered part of the "outline".

Result

The outline is considered by the software as part of the component body. Whenever it is encountered, the body-to-body clearance rule is invoked.

If a 2d item does not appear after you draw it, the color for Lines on layer 20 is probably the same as the background color. Set a different color for Lines in the Display Colors Setup dialog box.

Importing RF Shapes in DXF Format

You can import specialized shapes of DXF format into your decal or into the design using the AutoCAD 2004 DXF format.

Restriction: DXF import only supports the following geometries: POINT, LINE, ARC, CIRCLE, ELLIPS, TRACE, SOLID, 3DFACE, POLYLINE, LWPOLYLINE (AutoCAD R14), and BLOCKS with hierarchy.

1. Click the **Drafting Toolbar** button.
2. On the Drafting Toolbar, click the **Import DXF File** button.

Tip: The DXF import functionality of the Import DXF File button is optimized for importing RF shapes as 2D lines or copper. For more DXF import functionality use the File > Import method.

3. In the File Import dialog box, browse for the DXF file and then click **Open**.
4. In the DXF Import dialog box, in the DXF-File Unit list, select the units used in the DXF file.
5. For each layer you want to include in the import:
   a. Select the **Add** check box.
   b. In the PCB Layer column, double-click a layer box and select the PCB layer to use for the items of that DXF Layer.
   c. In the Type column, double-click a type box and select either **2D Line** or **Copper** for the DXF items of that layer.

      Restriction: A PCB layer set to <All Layers> cannot be imported as copper. You cannot have copper items on <All Layers> in a PCB decal.
6. Click **OK**.

Result

The geometries are added to the design or the decal in the PCB Decal Editor. If you need your imported geometries to be single objects, but they have been imported as multiple line items, see Joining and Closing 2D Lines and Copper Shapes.

Related Topics

Importing DXF Files

DXF Format in the *Concepts Guide*

Creating Attributes in the PCB Decal Editor

Attributes can be added automatically to the Attribute Dictionary when returning from the PCB Decal Editor.

You can use attributes in the PCB Decal Editor, but they differ in concept from attributes in the Layout Editor: The only attributes you can create in the PCB Decal Editor are decal attributes, which are associated with the physical decal.

The Attribute Dictionary doesn't exist in the PCB Decal Editor; therefore, attributes in the PCB Decal Editor are text strings only. They don't have properties, types, hierarchies, and other settings that attributes in the Attribute Dictionary have.

When you use Edit Decal to enter the PCB Decal Editor, upon exit, you are asked whether to apply the changes you made. If you apply the changes when you return to the Layout Editor, any attributes you added in the PCB Decal Editor are added to the Attribute Dictionary. The attributes are added at the Decal level of the attribute hierarchy and assigned to the appropriate objects.

Exception: When you enter PCB Decal Editor using the Tools menu command, any attributes you create in the PCB Decal Editor are not added to the Attribute Dictionary.

If you created an attribute in the PCB Decal Editor that exists in the Attribute Dictionary, the existing attribute in the dictionary is maintained. The attribute is assigned to the part that uses the decal. If a label for the attribute exists, it is associated with the attribute in the Attribute Dictionary.

If you created an attribute in the PCB Decal Editor that does not exist in the Attribute Dictionary, a non ECO-Registered attribute is created with the Free Text type. The attribute is then assigned to the part that uses the decal. If a label for the attribute exists, it is associated with the attribute created in the Attribute Dictionary.

Procedure

1. **PCB Decal Editor** > **Edit** menu > **Attribute Manager**.
2. Click **Add**.
3. Do one of the following:
    - Type an attribute name in the new **Attribute** cell, and press **Enter**. Attribute names are not checked in the PCB Decal Editor.
    - Select the new **Value** cell and click **Edit**.
    - Type an attribute value, and press **Enter**. You can also type units for the value. If you don't specify units, PADS Layout uses the same units as the decal.
4. Click **Close**.

Tip: You can also add attributes using the Browse Lib. Attr. button.

See also: Control of Solder Mask and Paste Mask

Modifying an Attribute

To modify an attribute:

1. **PCB Decal Editor** > **Edit** menu > **Attribute Manager**.
2. Select the **Attribute** cell or the **Value** cell for the attribute.
3. Click **Edit**.

4. Type a new name or value, and press **Enter**.
5. Click **Close**.

Creating Attribute Labels in the PCB Decal Editor

Use attribute labels in the PCB Decal Editor exactly as you use labels in the Layout Editor. Labels in the PCB Decal Editor offer you greater flexibility; you can display either decal attributes or an attribute from an object that uses the decal. For example, you can create a label for the part attribute Cost. Since Cost is not a decal attribute, you create the attribute in the Attribute Dictionary in the Layout Editor, and assign a placeholder label in the PCB Decal Editor.

See also: Creating Placeholder Attribute Labels

When you create labels, they may not be visible. Turn on the visibility of labels using the Display Colors Setup dialog box. Here, you can set the color for reference designators, part type, and attribute labels.

Tip: Pre-version 3.0 labels for part names, reference designators, and terminal numbers are converted to labels for version 3.0 and higher when you open the part in the Decal Editor.

Procedure

1. **Tools** menu > **PCB Decal Editor** > with a decal open > **Drafting** button > **Add New Label** button.
2. In the Add New Decal Label dialog box, in the Attribute list, select the attribute you want.

   Alternative:

   a. In the Attribute list click **Browse Lib. Attr.** to pick from the list of all library attributes.

   b. In the Browse Library Attributes dialog box click the attribute and click **OK**. The dialog box closes and the attribute name appears in the Attribute list in the Add New Decal Label dialog box.

   Restrictions:

   - If you are creating labels for jumpers, Reference Designator is the only available attribute.
   - Hidden attributes do not appear in the Attribute list unless the attribute was selected for the label before it was set as a hidden attribute.
3. The Value box lists the value of the selected attribute. Accept this value, or type a new one. This box is unavailable if you selected Reference Designator or Part Type from the Attribute list, if the attribute is read-only, or if you open the Properties dialog box for

labels of different attribute types. However, if the labels you select belong to attributes of the same type, you can edit this box.

Tips:

- If the attributes have different values, the box is blank. You can type a new value in the box to apply it to all of the selected attribute labels and their parent objects.
- Value is also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.

4. In the Show list, select the value you want to control the visibility of the label. You can choose to turn the label off, to display only the label name, to display only the label value, to display the name and value, or to display the full name and value (when labeling a structured attribute).

   Tip: Labels are invisible regardless of this setting unless you use the Display Colors Setup dialog box to change the color of labels to a color different from that of the background.

5. In the Font list, select the font you want to use.

   Tips:

   - Select stroke font or a system font.
   - For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.

6. In the Layer list, select the layer on which you want the decal label.
7. In the Position and sizes area, select the **Relative to** check box to place the label at the X and Y location relative to the component or the jumper. If you clear this check box, the label is placed at the X and Y location relative to the design origin.
8. In the X,Y location boxes, type new values to place the decal label in a specified location.
9. In the Rotation box, type a rotation angle to specify the rotation of the label.
10. In the Size box, type the size you want.
11. For stroke font, type the line width you want.
12. Select the **Mirrored** check box if you want to flip the label. When Mirrored is checked, text is considered readable from the bottom side of the board.
13. In the Justification area, set the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change.

    Tips:

- For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**.
- Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.

14. The Right reading area controls whether labels are readable (from left to right, or from bottom to top if the label is rotated). Click the **None**, **Orthogonal**, or **Angled** button to indicate the direction of reading you want.
15. Click **OK**.

Creating Placeholder Attribute Labels

A placeholder label is a label for an attribute that doesn't exist. Two practical uses for placeholder labels are:

- To predefine label positions in the decal and use them in the Layout Editor. This method also lets you create labels for non-decal attributes.
- To create a label for an attribute that will be created in the Layout Editor, but doesn't exist yet.

Procedure

1. When you create a label in the PCB Decal Editor, in the Add New Decal Label dialog box, in the Attribute list, click **new user attr**. UserAttribN appears in the Attributes list, where N is 1, 2, 3, or the next available number.
2. Type the name of an attribute for which you want to create a label.
3. Continue to define the label.

Once you create a placeholder label, you can easily associate it with an existing attribute in the Attribute Dictionary when you return to the Layout Editor.

Modifying Decal Label Properties

Use the Decal Label Properties dialog box to modify a decal label or to change the attribute the label displays.

See also: Creating Attribute Labels in the PCB Decal Editor

Tip: If you select multiple labels, settings in this dialog box apply to all selected labels.

Procedure

1. PCB Decal Editor > Select a decal label > right-click > **Properties**.

2. In the Decal Label Properties dialog box, in the Attribute list, select the attribute you want. If you are creating or modifying labels for jumpers, Reference Designator is the only available attribute.

   Tip: Hidden attributes do not appear in the Attribute list unless the attribute was selected for the label before it was set as a hidden attribute.

3. The Value box lists the value of the selected attribute. Accept this value, or type a new one. This box is unavailable if you clicked Reference Designator or Part Type from the Attribute list, if the attribute is read-only, or if you open the Properties dialog box for labels of different attribute types. However, if the labels you select belong to attributes of the same type, you can edit this box.

   Tips:

   - If the attributes have different values, the box is blank. You can type a new value in the box to apply it to all of the selected attribute labels and their parent objects.
   - Value is also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.

4. In the Show list, click the value you want to control the visibility of the label. You can choose to turn the label off, to display only the label name, to display only the label value, to display the name and value, or to display the full name and value (when labeling a structured attribute).

   Tip: Labels are invisible regardless of this setting unless you use the Display Colors Setup dialog box to change the color of labels to a color different from that of the background.

5. In the Font list, select the font you want to use.

   Tips:

   - Select stroke font or a system font.
   - For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.

6. In the Layer list, select the layer on which you want the decal label.

7. In the Position and sizes area, select the **Relative to Component** check box to place the label at the X and Y location relative to the component or the jumper. If you clear this check box, the label is placed at the X and Y location relative to the design origin.

8. In the X,Y location boxes, type new values to move the decal label to a specified location.

9. The Rotation box shows the current rotation angle of the label. Type a new rotation angle if you want to change the rotation of the label.

10. In the Size box, type the size you want.

11. For stroke font, type the line width you want.
12. Select the **Mirrored** check box if you want to flip the label. When Mirrored is checked, text is considered readable from the bottom side of the board.
13. In the Justification area, set the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change.

    Tips:

    - For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**.
    - Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.
14. The Right reading area controls whether labels are readable (from left to right, or from bottom to top if the label is rotated). Click the **None**, **Orthogonal**, or **Angled** button to indicate the direction of reading you want.
15. Click **OK**.

Creating Keepout Areas in the PCB Decal Editor

Create a keepout to define areas where design objects cannot be placed. You can create keepout areas using closed polygons (with or without arcs), circles, or rectangles. The current angle mode and design grid settings determine the placement of the lines.

You can create keepouts in both the Layout Editor and the PCB Decal Editor. User interaction is the same for either; the only difference is the type of objects you can restrict.

Procedure

1. With a decal open in the Decal Editor, **Drafting Toolbar** button > **Keepout** button.
2. Right-click and click a draw mode for the type of shape to create.
3. Create a closed shape to define the keepout area. The Drafting Properties dialog box automatically appears when you close the shape.
4. In the Add Drafting Dialog Box, select restrictions.
5. Click the layer on which to place the keepout.

    Tips:

    - When you choose layer assignments, restrictions not available for that layer are unavailable.

- When defining keepouts in the PCB Decal Editor, you can also assign keepouts to an <Opposite Side> layer. You can't do this in the Layout Editor.

6. Click **OK**. The keepout is created. If you create other keepouts, they use the restrictions you set here as the default.

Modifying Decal-level Keepouts

Once you create a decal-level keepout, you must use the PCB Decal Editor to modify any of the keepout properties. You can change the size of a keepout just as you would any other drafting object: move an edge or corner, or change the diameter of a circle. You can also copy a keepout to another location and change its restrictions or layer assignments.

Procedure

1. Select an edge > **right-click > Select Shape**
2. Right-click and click **Properties**.
3. Turn restrictions on or off and modify the layer settings. See also: Modifying Drafting Object Properties
4. Click **OK**.

Tip: You can't modify a keepout that is part of a physical design reuse. If you try to, the message "Reuse elements cannot be modified. Break the reuse first" appears. Click OK to cancel the operation.

Generating Drafting Shapes from Terminals

You can use the outline of a terminal as the basis to create new drafting shapes.

Procedure

1. Select one or more terminals.
2. Right-click and click **Generate Drafting Shape**.
3. In the Generate Drafting Shape dialog box, select the type of drafting item from the Type list. The valid drafting types are 2D Line, Copper, Copper Cut Out, and Keepout.
4. In the Layer list, select the Layer on which to place the new drafting shape.
5. In the Width box, type a line width for the new shape.
6. In the Oversize/Undersize value box, do one of the following:
    - To create a new drafting shape larger than the terminal outline by the typed value, type a positive number.

- To create a new drafting shape equal in size to the terminal, type 0 (zero).
- To create a new drafting shape smaller than the terminal outline by the typed value, type a negative number.

7. Click **Complete** to close the dialog box and create a new shape or click **Create** to keep the dialog box open for creating more shapes.

Working with Solder and Paste Masks

You can create custom openings in the top and bottom solder mask and paste mask layers for individual pad stacks, for decals, or for components. You do this in the Decal Editor, or in the pad stack.

See:

Creating Solder Mask Openings in the Decal Editor

Creating Solder Mask Openings in the Pad Stack

Creating Paste Mask Openings in the Decal Editor

Creating Paste Mask Openings in the Pad Stack

Related Topics

Control of Solder Mask and Paste Mask

Creating Solder Mask Openings in the Decal Editor

There are two ways to create custom solder mask openings in the Decal Editor. You can:

- Copy a pad shape, over- or undersize it, and save it as copper on the Top or Bottom solder mask layer.
- Draw the mask opening shape in copper on the Top or Bottom solder mask layer.

Procedure1—Copy a Pad Shape

1. Open the decal from the library or the design.
2. With nothing selected, **right-click** > **Select Terminals**.
3. Select the terminal you want to copy, then **right click** > **Generate Drafting Shape**.
4. In the dialog box, select copper from the Type list.
5. Select Solder Mask Top or Solder Mask Bottom from the Layer list.
6. Set Width and Oversize as desired.

7. Select **Create**.
8. Select **Complete**.

Results

To see the new opening shape(s):

1. In the Display Colors Setup Dialog Box, assign a color to the Top or Bottom solder mask layer.
2. Make the layer active by selecting it from the Layer List.

Procedure2—Draw the Opening Shape

Tip: You can also use this procedure to create gang relief of solder mask.

1. Open the decal from the library or the design.
2. Select the Solder Mask Top or Solder Mask Bottom layer.
3. Click the Drafting Toolbar button.
4. Click the Copper button.
5. Create the new opening shape.

Results

To see the new opening shape(s):

1. In the Display Colors Setup Dialog Box, assign a color to the Top or Bottom solder mask layer.
2. Make the layer active by selecting it from the Layer List.

Related Topics

Control of Solder Mask and Paste Mask

Creating Solder Mask Openings in the Pad Stack

You can create custom solder mask openings for individual pins or components in the pad stack.

Tip: Solder Mask openings in the pad stack have priority over all other solder mask opening values. See Control of Solder Mask and Paste Mask.

Procedure

1. With nothing selected, **right-click** > **Select Components**.

2. Select the component whose pin(s) you want to create solder mask openings for.
3. **Right-click** > **Properties**
4. In the Component Properties dialog box, select the **Pad Stack** button.
5. In the Pad Stacks Properties dialog box, *if you want to create an opening for only a single pin,* select the **Add** button under the Pin: Plated: list, and select the pin. If not, go to Step 6.
6. Select the **Add** button under the Sh; Sz: Layer: list, and add the Solder Mask Top or Solder Mask Bottom layer.
7. Select the appropriate pad shape, and set the Width;, Length: and Orientation values for the new solder mask opening.
8. Click **OK**.

Results

To see the new opening shape(s):

1. In the Display Colors Setup Dialog Box, assign a color to the Top or Bottom solder mask layer.
2. Make the layer active by selecting it from the Layer List.

Related Topics

Control of Solder Mask and Paste Mask

Creating Paste Mask Openings in the Decal Editor

There are two ways to create custom paste mask openings in the Decal Editor. You can:

- Copy a pad shape, over- or undersize it, and save it as copper on the Top or Bottom paste mask layer.
- Draw the mask opening shape in copper on the Top or Bottom paste mask layer.

Procedure 1—Copy a Pad Shape

1. Open the decal from the library or the design.
2. With nothing selected, **right-click** > **Select Terminals**.
3. Select the terminal you want to copy, then **right click** > **Generate Drafting Shape**.
4. In the dialog box, select copper from the Type list.
5. Select Paste Mask Top or Paste Mask Bottom from the Layer list.

6. Set Width and Oversize as desired.
7. Select **Create**.
8. Select **Complete**.

Results

To see the new opening shape(s):

1. In the Display Colors Setup Dialog Box, assign a color to the Top or Bottom paste mask layer.
2. Make the layer active by selecting it from the Layer List.

Procedure 2—Draw the Opening Shape

1. Open the decal from the library or the design.
2. Select the Paste Mask Top or Paste Mask Bottom layer.
3. Click the Drafting Toolbar button.
4. Click the Copper button.
5. Create the new opening shape.

Results

To see the new opening shape(s):

1. In the Display Colors Setup Dialog Box, assign a color to the Top or Bottom paste mask layer.
2. Make the layer active by selecting it from the Layer List.

Related Topics

Control of Solder Mask and Paste Mask

Creating Paste Mask Openings in the Pad Stack

You can create custom paste mask openings for individual pins or components in the pad stack.

Tip: Paste Mask openings in the pad stack have priority over all other paste mask opening values. See Control of Solder Mask and Paste Mask.

Procedure

1. With nothing selected, **right-click** > **Select Components**.

2. Select the component whose pin(s) you want to create paste mask openings for.
3. **Right-click** > **Properties**
4. In the Component Properties dialog box, select the **Pad Stack** button.
5. In the Pad Stacks Properties dialog box, *if you want to create an opening for only a single pin,* select the **Add** button under the Pin: Plated: list, and select the pin. If not, go to Step 6.
6. Select the **Add** button under the Sh; Sz: Layer: list, and add the Paste Mask Top or Paste Mask Bottom layer.
7. Select the appropriate pad shape, and set the Width, Length and Orientation values for the new paste mask opening.
8. Click **OK**.

Results

To see the new opening shape(s):

1. In the Display Colors Setup Dialog Box, assign a color to the Top or Bottom paste mask layer.
2. Make the layer active by selecting it from the Layer List.

Related Topics

Control of Solder Mask and Paste Mask

Updating a Design from the Library

In PADS Layout, when you import a netlist into a design, a copy of the library content for the parts is written into the design. Over time, however, as updates are made to the library, the parts in the design can become out of sync with newer versions in the library.

Use Update from Library (UFL) to create a report of any part differences between the current design and the library, and to update the design to reflect the current state of the library. You can compare or update:

- Part types
- PCB decals and decal rules
- Attributes and labels

Caution

Since each item in your design is linked to its library counterpart by name only, unconsidered updating can have unforeseen and unintended consequences. Before you update any design, you should always ascertain how updating will affect it. Either generate a comparison report and determine what differences exist between the design and library (and thus what changes will occur during the update,) or make a copy of the design and check/test the updated copy for unwanted changes or behavior.

Tip: To find differences between the library and design in the comparison report, search for “different”.

Caution

Design changes resulting from updating can cause existing associated nets to be truncated, split, or deleted altogether. It can also cause existing associated nets to be extended, or new associated nets to be created if the refdes prefixes of updated components are specified in the Associated Nets dialog box. For more information, see the Associated Nets chapter in this manual.

Procedure

Tip: UFL searches libraries for part types using the library order shown in the Library List dialog box (**File Menu > Library > Manage Lib. List button). If necessary, verify that the library order is correct before executing this procedure.**

From the design editor:

1. If you want to compare/update only specific components, select them. Otherwise, UFL selects all components in the design by default.
2. **Tools menu > Update from Library**
3. Modify the settings in the Update from Library Dialog Box.
4. Click OK.

Results

When the comparison or update operation is completed, the update report is opened for viewing, and a link to the report file is written to the Output Window.

Undoing an Update

If at least one item is updated during an update operation, the update process creates an undo checkpoint. So you can undo any changes made to the design to the last-saved checkpoint using **Edit menu > Undo**.

Tip: If nothing is updated, UFL does not create a new undo checkpoint and preserves undo data and undo checkpoints created by previous commands.

Preventing Undo Buffer Overruns

The data required to undo an update is stored in the *undo buffer*. To prevent overflows of this buffer when updating very large designs, UFL estimates the amount of undo data being generated, and if there is risk of a buffer overflow, displays this prompt:

Respond with one of the following actions:

- Click OK to clear the undo buffer and *disable undo creation for this update*. The update operation will continue, and you will be unable to undo the updates.
- Click Cancel to cancel the update and return to the UFL dialog. Undo data and undo checkpoints created by previous commands are preserved.

Related Topics

The Compare/Update Process

Update from Library Dialog Box

How to Read the Update Report

The Compare/Update Process

The update process begins with a comparison of the design content with the library content. When design content is found to be different from the library content, the design content is updated according to the options set in the Update From Library dialog box, and the results documented in the report file. The compare process is the same, except that the design is not updated.

General Rules

In comparing and updating, the following general rules apply for all items:

- **Matching items** — For each part type and decal listed in the Design items area of the Update from Library dialog box, all available libraries are searched to find a matching named entry, and the first matching entry found is used. If no match is found, comparison of the item is skipped, and a "not found" entry is made in the report file.

  Tips:

  - Libraries are searched in the order shown in the Library List dialog box(**File** menu > **Library** > **Manage Lib. List button**). You can use this dialog box to change the search order.

    - Libraries that have the Allow Search property turned off are ignored.

- **Timestamps** —Each part type and decal in the design and library has a timestamp. Depending on the options chosen in the UFL dialog box, comparison of the timestamps can determine what is compared/updated. See Update from Library Dialog Box.

  Tip: Designs created with PADS Layout versions prior to 9.1 do not have timestamps. See Older Designs for information on how to handle these designs.

What is Compared

Table 5-1 lists the content compared and reported for each item selected for comparison or update.

Table 5-1. Compared and Reported Part Structures

Item	Item Component	What is Compared & Reported
Part Type		Timestamp, Logic Family, ECO Registered (y n), Special Purpose flag, gates count, gate pins count, signal pins count, connector pins count, attributes count, pin mappings count, decals count, associated decals list
	For each gate	Pin count, gate swap ID
	For each gate pin	Pin number, pin name, pin type, pin swap ID
	For each signal pin	Pin number, name
	For each connector pin	Pin number
	For each attribute	Attribute value
	For each pin mapping	Pin number value
PCB Decal		Timestamp, terminals count, padstacks count, attributes count, body outline pieces count, free coppers count, keepouts count, text items count, labels count, dimensions count, associated coppers count, decal rule set count
	For each terminal	Terminal number, associated padstack, location (x y), associated copper list
	For each padstack	Drill diameter, drill plated (y n), drill slotted(y n), drill length, drill offset, drill orientation, pad count, plus: • *For each normal pad:* Layer, pad shape and size, corner style, offset, orientation • *For each thermal pad*: Layer, thermal shape and size, spoke count, spoke width, spoke angle • *For each antipad*: Layer, antipad shape and size
	For each attribute	Attribute value

Table 5-1. Compared and Reported Part Structures

Item	Item Component	What is Compared & Reported
	For each decal rule	Rule value

	For each body outline piece	Layer, shape (OPEN, CLOSED, or CIRCLE), origin (First Corner location) (x y), line width, plus:
		• *If shape is OPEN or CLOSED*: geometry corner count, geometry arc count, plus:
		  • *For each corner:* location (x y)
		  • *For each arc:* arc center and radius
		• *If shape is CIRCLE*: circle radius

	For each free copper	Layer, shape (OPEN, CLOSED, or CIRCLE), origin (First Corner location) (x y), line width, associated cutout count, plus:
		• *If shape is OPEN or CLOSED*: geometry corner count, geometry arc count, plus:
		  • *For each corner:* location (x y)
		  • *For each arc:* arc center and radius
		• *If shape is CIRCLE*: circle radius
		• *For each associated cutout*: shape (OPEN, CLOSED, or CIRCLE), origin (First Corner position) (x y), line width, plus:
		  • *If shape is OPEN or CLOSED*: geometry corner count, geometry arc count, plus:
		  • *For each corner:* location (x y)
		  • *For each arc:* arc center and radius
		  • *If shape is CIRCLE*: circle radius

	For each keepout	Shape (OPEN, CLOSED, or CIRCLE), origin (First Corner location) (x y), line width, restrictions, plus:
		• *If shape is OPEN or CLOSED*: geometry corner count, geometry arc count, plus:
		  • *For each corner:* location (x y)
		  • *For each arc:* arc center and radius
		• *If shape is CIRCLE*: circle radius

	For each text item	Layer, text string, font, text height, line width, mirror (y n), location (x y), rotation, justification (horizontal vertical).
	For each label	Associated attribute, layer, font, text height, line width, mirror (y n), location (x y), rotation, justification (horizontal vertical), right reading, visibility

Table 5-1. Compared and Reported Part Structures

Item	Item Component	What is Compared & Reported

	For each associated copper	Terminal ID, Layer, shape (OPEN, CLOSED, or CIRCLE), origin (First Corner position) (x y), line width, associated cutout count, plus:
		• *If shape is OPEN or CLOSED*: geometry corner count, geometry arc count, plus:
		  • *For each corner:* location (x y)
		  • *For each arc:* arc center and radius
		• *If shape is CIRCLE*: circle radius
		• *For each associated cutout*: shape (OPEN, CLOSED, or CIRCLE), origin (First Corner location) (x y), line width, plus:
		  • *If shape is OPEN or CLOSED*: geometry corner count, geometry arc count, plus:
		  • *For each corner:* location (x y)
		  • *For each arc:* arc center and radius
		  • *If shape is CIRCLE*: circle radius

	For each dimension	Type, layer, origin (x y), textfont, text, piece count, plus: • *For the text*: Layer, font, text height, line width, mirror, location (x y), rotation, justification (horizontal vertical) • *For each piece*: piece type, geometry corner count, geometry arc count, plus: • *For each arc*: arc center and radius

Older Designs

Designs created with PADS Layout versions prior to 9.1 do not have timestamps. When one of these designs is opened, the timestamps on all part types and decals in the design are set to “Jan 01 00:00:00 1970” (UTC). To find out whether the *content* of the part types and decals in the design is in sync with the library, compare the design and library with the Hide identical results check box selected.

Related Topics

Updating a Design from the Library

Update from Library Dialog Box

How to Read the Update Report

How to Read the Update Report

Every time you update a design from the library or generate a comparison report, the results of the operation are written to a new UpdateFromLibrary_Layout_Report.txt file in C:\PADS

Projects. The Update Report gives you information about the items selected for comparison or update, including:

- Whether the timestamp of an item in the design is older, newer, or the same as the item's timestamp in the library.
- Whether the content of an item in the design is the same as or different from the item's content in the library.
- The differences between the compared items.

Update Report Examples

The Update Report is divided into a header and six sections—Report Options, Results Summary, Part Types Summary, PCB Decals Summary, Part Type Comparison Details, and PCB Decal Comparison Details. Examples of each section are given in the following figures:

- **Report Options section**—Lists the options selected for this operation in the Update from Library dialog box. Example:

```
========================================================================
 REPORT OPTIONS
========================================================================

   Mode
      Update design from library
      Include items with identical timestamps:Yes

   Items to compare / update
      Part types:Yes
      PCB decals:Yes
      Decal rules:Yes
      Attributes and labels:Yes

   Update by timestamps
      Update item in design even if library item is older:Yes
      Update item even if timestamps are identical:Yes

   Update preferences
      Preserve alternative decals not found in library:Yes
      Preserve decal rules not found in library:Yes
      Preserve design attributes not found in library:Yes
      Add new attributes not found in design:Yes
      Update common attributes:Yes

   Preserve design labels
      Visibility:Yes
      Location:Yes
      Label properties:Yes

   Report filtering
      Summary and details
      Hide identical results:No
      Hide graphics comparison details:No
```

- **Results Summary section—**Gives a statistical summary of comparison and update results. Example:

```
==================================================================
 RESULTS SUMMARY
==================================================================

      Comparison Summary:

         Selected    -      32
         Compared    -      30
         Different   -      14
         Warnings    -       0
         Errors      -       0
         Skipped     -       2

      Update Summary:

         Updated     -      30
         Warnings    -       0
         Failed      -       0
         Skipped     -       0
```

- **Part Types Summary section—**Lists all the selected part types and decals, and gives comparison/update information about each. Example:

PART TYPES SUMMARY

Part Type	Found in Library	Design Timestamp	Compared Content	Update Status
+5VREG	preview	older than in library	same	updated
24-576MHZ	preview	older than in library	same	updated
26PINCONN	preview	older than in library	same	updated
87C256	preview	older than in library	different	updated
AM100415	amd	older than in library	different	updated
CAP-ELECTAA	misc	older than in library	different	updated
CAP1206	misc	older than in library	different	updated
CD4001B	national	older than in library	different	updated
CD4069	national	older than in library	different	updated
LEDAK	< not found in library >	not checked		not touched
PAL16R8	amd	older than in library	different	updated
R1/8W	misc	older than in library	different	updated
SHIELD	preview	older than in library	same	updated

- **PCB Decals Summary section—**Lists all the PCB Decals associated with the selected part types and gives comparison/update information about each. Example:

PCB DECALS SUMMARY

PCB Decal	Found in Library	Design Timestamp	Compared Content	Update Status
1206	common	older than in library	same	updated
26PINCONN	common	older than in library	different	updated
6032	common	older than in library	same	updated
DIP14	common	older than in library	same	updated
DIP16	common	older than in library	same	updated
DIP20	common	older than in library	same	updated
ELECTAA	common	older than in library	same	updated
LCC20SQ	common	older than in library	same	updated
LEDR_A	< not found in library >	not checked		not touched
OSC	preview	older than in library	different	updated
R1/8W	common	older than in library	same	updated
R1/8WA	common	older than in library	same	updated
R1/8WB	common	older than in library	same	updated
SHIELD	preview	older than in library	same	updated
SO14	common	older than in library	different	updated
SO16	common	older than in library	different	updated
SO20	common	older than in library	different	updated
SO28	common	newer than in library	different	updated
TO-213AA	common	older than in library	same	updated

- **Part Type Comparison Details section—**Lists compared part types, showing detailed comparison results. Example:

```
==========================================================================
 PART TYPE COMPARISON DETAILS
==========================================================================

 Part Type: CAP1206
 ------------------

 Part Type Summary     Comparison            Design                                Library: misc
 ==============================================================================================================
 Timestamp            |             different | Thu Jan 01 00:00:00 1970 UTC        | Wed Dec 22 14:44:43 2004
 Logic Family         |                  same | CAP                                 | CAP
 ECO Registered       |                  same | Yes                                 | Yes
 Special Purpose      |                  same | none                                | none
 Gates                |                  same | count =      1                      | count =      1
 Gate Pins            |                  same | count =      2                      | count =      2
 Signal Pins          |                  same | count =      0                      | count =      0
 Pin Mapping          |                  same | count =      0                      | count =      0
 Attributes           |             different | count =      6                      | count =      9
 Decals               |                  same | count =      1                      | count =      1
    Decal 1           |                  same | 1206                                | 1206
 ==============================================================================================================

 Part Type Details - Gates          | Comparison            | Design                                | Library: misc
 ==============================================================================================================
 Gate: A                            |                  same |                                       |
    Gate Pin Count                  |                  same | 2                                     | 2
    Gate Swap ID                    |                  same | 0                                     | 0
 ==============================================================================================================

 Part Type Details - Gate Pins      | Comparison            | Design                                | Library: misc
 ==============================================================================================================
 Gate Pin: A-1                      |                  same |                                       |
    Pin Number                      |                  same | 1                                     | 1
    Pin Functional Name             |                  same |                                       |
    Pin Swap ID                     |                  same | 1                                     | 1
    Pin Type                        |                  same | Undefined                             | Undefined
 --------------------------------------------------------------------------------------------------------------
 Gate Pin: A-2                      |                  same |                                       |
    Pin Number                      |                  same | 2                                     | 2
    Pin Functional Name             |                  same |                                       |
    Pin Swap ID                     |                  same | 1                                     | 1
    Pin Type                        |                  same | Undefined                             | Undefined
 ==============================================================================================================

 Part Type Details - Attributes     | Comparison            | Design                                | Library: misc
 ==============================================================================================================
 Sim.Analog.Model                   |             different | < no attribute >                      | < no value >
 Sim.Analog.Order                   |             different | < no attribute >                      | Model$
 Sim.Analog.Prefix                  |             different | < no attribute >                      | < no value >
 Description                        |                  same | SURFACE MOUNT CAPACITOR 0.062 X ...   | SURFACE MOUNT CAPACITOR 0.0
 --------------------------------------------------------------------------------------------------------------
                      Library Value | SURFACE MOUNT CAPACITOR 0.062 X 0.126 INCHES
                       Design Value | SURFACE MOUNT CAPACITOR 0.062 X 0.126 INCHES
 --------------------------------------------------------------------------------------------------------------
 Part Number                        |                  same | < no value >                          | < no value >
 Manufacturer #1                    |                  same | IPC SM-782 STD.                       | IPC SM-782 STD.
 Value                              |                  same | ???                                   | ???
 Tolerance                          |                  same | < no value >                          | < no value >
 Voltage Rating                     |             different | < no attribute >                      | < no value >
 Geometry.Height                    |             different | 60.00 mil                             | < no attribute >
 ==============================================================================================================
```

- **PCB Decal Comparison Details section—**Lists compared decals, showing detailed comparison results. Example:

```
==========================================================================
 PCB DECAL COMPARISON DETAILS
==========================================================================

 PCB Decal: 1206
 ---------------

 PCB Decal Summary     Comparison           Design                                  Library: common
 =============================================================================================================
 Timestamp            /          different / Thu Jan 01 00:00:00 1970 UTC          / Tue Aug 31 11:21:39 1999
 Terminals            /               same / count =     2                         / count =     2
 Padstacks            /               same / count =     1                         / count =     1
 Associated Coppers   /               same / count =     0                         / count =     0
 Attributes           /               same / count =     0                         / count =     0
 Rules                /               same / count =     0                         / count =     0
 Attribute Labels     /               same / count =     2                         / count =     2
 Free Texts           /               same / count =     0                         / count =     0
 Free Coppers         /               same / count =     0                         / count =     0
 Keepouts             /               same / count =     0                         / count =     0
 Body Outlines        /               same / count =     2                         / count =     2
 Dimensions           /               same / count =     0                         / count =     0
 =============================================================================================================

 Decal Details - Terminals          / Comparison          / Design                              / Library: common
 =============================================================================================================
 Terminal: 1                        /                same /                                     /
    Terminal Name                   /                same / 1                                   / 1
    Terminal Location               /                same /      (    -67.50      0.00 ) mils   /      (    -67.50      0
    Padstack                        /                same / Padstack     1                      / Padstack     1
 Associated Coppers                 /                same / count =     0                       / count =     0
 -------------------------------------------------------------------------------------------------------------
 Terminal: 2                        /                same /                                     /
    Terminal Name                   /                same / 2                                   / 2
    Terminal Location               /                same /      (     67.50      0.00 ) mils   /      (     67.50      0
    Padstack                        /                same / Padstack     1                      / Padstack     1
 Associated Coppers                 /                same / count =     0                       / count =     0
 =============================================================================================================

 Decal Details - Padstacks          / Comparison          / Design                              / Library: common
 =============================================================================================================
 Padstack                           /                same / Padstack     1                      / Padstack     1
    Drill Diameter                  /                same /             0.00               mils /             0.00
    Plated                          /                same / Yes                                 / Yes
    Slotted                         /                same / No                                  / No
    Pad Count                       /                same / 3                                   / 3
 .............................................................................................................
 Pad: <Mounted Side>                /                same /                                     /
    Shape                           /                same / Square       60.00              mils / Square       60.00
    Corner Style                    /                same / 90 Degrees                           / 90 Degrees
    Offset                          /                same /             0.00               mils /             0.00
    Orientation                     /                same /             0.000           degrees /             0.000
 .............................................................................................................
 .
 .
 .
 =============================================================================================================

 Decal Details - Attribute Labels   / Comparison          / Design                              / Library: common
 =============================================================================================================
 Attribute Label                    /                same / Ref.Des.                            / Ref.Des.
    Layer No.                       /                same / Primary Component Side              / Primary Component Side
    Location                        /                same /      (    -70.00    150.00 ) mils   /      (    -70.00    150
    Font                            /                same / <Romansim Stroke Font>              / <Romansim Stroke Font>
    Line Width                      /                same /            10.00               mils /            10.00
    Text Height                     /                same /           100.00               mils /           100.00
    Text Orientation                /                same /             0.000           degrees /             0.000
    Mirror                          /                same / No                                  / No
    Right Reading                   /                same / None                                / None
    Justification (hor vert)        /                same / Left Down                           / Left Down
    Visibility                      /                same / Value                               / Value
 -------------------------------------------------------------------------------------------------------------
 .
 .
 .
 =============================================================================================================

 Decal Details - Body Outlines      / Comparison          / Design                              / Library: common
 =============================================================================================================
 Body Outline: 1                    /                same /                                     /
    Layer No.                       /                same / <All Layers>                        / <All Layers>
    Piece Type                      /                same / Open Piece                          / Open Piece
    Location                        /                same /      (    -37.50     50.00 ) mils   /      (    -37.50     50
    Corner Count                    /                same / 4                                   / 4
    Arc Count                       /                same / 0                                   / 0
    Geometry Details                /                same /                                     /
       Point Location               /                same /    1:(    -37.50     50.00 ) mils   /    1:(    -37.50     50
       Point Location               /                same /    2:(   -118.00     50.00 ) mils   /    2:(   -118.00     50
       Point Location               /                same /    3:(   -118.00    -50.00 ) mils   /    3:(   -118.00    -50
       Point Location               /                same /    4:(    -37.50    -50.00 ) mils   /    4:(    -37.50    -50
 -------------------------------------------------------------------------------------------------------------
 .
 .
```

Update Report Messages

Table 5-2 lists and describes the messages returned for compared/updated items in the Update Report.

Table 5-2. Update Report Messages—Part Type and Decals Summaries

Appears in Column:	Message	Means
Part Types Summary and PCB Decals Summary Messages		
Design Timestamp	older than in library	Design version is older than Library version.
	identical	Design and library versions have identical timestamps.
	newer than in library	Design version is newer than Library version.
	not checked	Timestamps were not compared because the item was not found in the library.
Compared Content	same	*Compared* item content in the design and library are the same. (Does not apply to any item content that is not compared because the comparison is turned off in the Update from Library dialog box.)
	different	*Compared* item content in the design and library are different.
	not checked	Content was not compared because the item was not found in the library.
Update Status	updated	Design item has been updated with content from the library.

Table 5-2. Update Report Messages—Part Type and Decals Summaries

Appears in Column:	Message	Means
	not touched	Design item (part type/decal) was not updated because: • UFL was run in Compare mode, or • The item was not found in the library.
	update failed	The item could not be updated. Some possible causes are: • Lost connection to library file • Mismatch of pin counts between design and library items • The library part type has no decal assigned. • Decal not found in library when updating part type *and* decal
	skipped	Items are skipped if: • They don't qualify for Compare/Update (for example, items with identical timestamps), or • The item was not found in the library.
Part Type Comparison Details & PCB Decal Comparison Details Messages		
Comparison	same	Item content in the design and library are the same. (Order can be different.)

Table 5-2. Update Report Messages—Part Type and Decals Summaries

Appears in Column:	Message	Means
	different	Item content in the design and library are different. **Tip:** A design that has been exported to ASCII and reimported may show differences in graphical data resulting from optimizations made during ASCII export.
	not found in library	Item does not exist in the library.
	not found in design	Item does not exist in the design.
Library *Design*	none	Item has no Special Purpose flag.
	< no value >	Attribute exists but has no value assigned.
	< no attribute >	Attribute not found.

Related Topics

Updating a Design from the Library

Update from Library Dialog Box

The Compare/Update Process

Chapter 6
Creating and Modifying Part Types

In this chapter:

Creating a New Part Type

There are five different component types that require specific instructions to create the part type:

- It's an electrical part, and the logical and physical pin numbering is the same--both numeric, or both alphanumeric. See Creating a Part Type with Identical Schematic and Layout Pin Numbering.
- It's an electrical part, and the logical and physical pin numbering is different--the logical is alphanumeric and the physical is numeric. See Creating a Part Type with Different Schematic and Layout Pin Numbering.
- It's a connector. See Creating a Connector Part Type.
- It will not be ECO registered. See Creating a Non-ECO-Registered Part Type.
- It's a non-electrical part—such as a mounting screw or locating post—that is not connected to any net. See Creating a Non-Electrical Part Type.

Creating a Part Type with Identical Schematic and Layout Pin Numbering

Use this procedure to create a part type that uses the same logical and physical pin numbering. The schematic-to-layout pin numbering can be either alphanumeric-to-alphanumeric or numeric-to-numeric.

Requirement

Your PCB decal(s) must exist to use this procedure. See Creating a New Decal.

Procedure

1. **File** menu > **Library**
2. In the Library Manager dialog box, select a library for the new part.
3. Click the **Parts** button, and then click the **New** button.
4. On the General tab of the Part Information dialog box, select the logic family; this sets the refdes prefix.
5. Select the **ECO Registered Part** check box.
6. On the PCB Decals tab, assign one or more decals to the part.
   See also: Assigning PCB Decals to Part Types
7. On the Gates tab, add at least one gate. (Add multiple gates if you want to be able to swap them.)
   See also: Assigning CAE Decals to Gates
8. On the Pins tab, you must fill out the Pin Group and Name columns; the other columns are optional.
   See also: Editing Pins Table Data.
9. On the Attributes tab, add attributes as necessary.
   See also: Defining Default Attributes for New Parts
10. Click **OK**.
    See also: Part Type Error Checking
11. In the Save Part Type to Library dialog box, type a name for the new part, and click **OK**.

Related Topics

Creating a New Part Type

Creating a Part Type with Different Schematic and Layout Pin Numbering

Use this procedure to create a part type that uses alphanumeric numbering for the logical pins and numeric numbering for the physical pins.

Requirement

Your PCB decal(s) must exist to use this procedure. See Creating a New Decal.

Procedure

1. **File** menu > **Library**
2. In the Library Manager dialog box, select a library for the new part.

3. Click the **Parts** button, and then click the **New** button.
4. On the General tab of the Part Information dialog box, select the logic family; this sets the refdes prefix.
5. Select the **Define mapping of Part Type pin numbers to PCB Decal** check box.
6. Select the **ECO Registered Part** check box.
7. On the PCB Decals tab, assign one or more numeric decals to the part.
   See also: Assigning PCB Decals to Part Types
8. On the Gates tab, add at least one gate. (Add multiple gates if you want to be able to swap them.)
   See also: Assigning CAE Decals to Gates
9. On the Pins tab:
   a. Change the pin numbers to alphanumerics to match the schematic alphanumerics.
      See also: Renumbering Pins in the Pins Table.
   b. Fill out the Pin Group and Name columns; the other columns are optional.
      See also: Editing Pins Table Data.
10. On the Attributes tab, add attributes as necessary.
    See also: Defining Default Attributes for New Parts
11. On the Pin Mapping tab, map the unmapped logical pins (alphanumeric) to the physical decal pins (numeric).
    See also: Mapping Alphanumeric Pin Numbers to Numeric Decals.
12. Click **OK**.
    See also: Part Type Error Checking
13. In the Save Part Type to Library dialog box, type a name for the new part, and click **OK**.

Related Topics

Creating a New Part Type

Creating a Non-ECO-Registered Part Type

Use this procedure to create a non-ECO-registered part type. A non-ECO-registered part—such as a mounting-hole, for example—is not transferred during forward or backward annotation.

Requirement

Your PCB decal(s) must exist to use this procedure. See Creating a New Decal.

Procedure

1. **File** menu > **Library**
2. In the Library Manager dialog box, select a library for the new part.
3. Click the **Parts** button, and then click the **New** button.
4. On the General tab of the Part Information dialog box, select the logic family; this sets the refdes prefix.
5. Clear the **ECO Registered Part** check box.
6. On the PCB Decals tab, assign one or more decals to the part.
   See also: Assigning PCB Decals to Part Types
7. On the Gates tab, add at least one gate. (Add multiple gates if you want to be able to swap them.)
   See also: Assigning CAE Decals to Gates
8. On the Pins tab, you must fill out the Pin Group and Name columns; the other columns are optional.
   See also: Editing Pins Table Data.
9. On the Attributes tab, add attributes as necessary.
   See also: Defining Default Attributes for New Parts
10. Click **OK**.
    See also: Part Type Error Checking
11. In the Save Part Type to Library dialog box, type a name for the new part, and click **OK**.

Related Topics

Creating a New Part Type

Creating a Non-Electrical Part Type

Use this procedure to create a non-electrical part type—such as a mounting screw or locating post—for adding to a parts list.

Requirement

Your PCB decal(s) must exist to use this procedure. See Creating a New Decal.

Procedure

1. **File** menu > **Library**
2. In the Library Manager dialog box, select a library for the new part.

3. Click the **Parts** button, and then click the **New** button.
4. On the General tab of the Part Information dialog box, select the logic family; this sets the refdes prefix.
5. Clear the ECO Registered Part check box.
6. On the PCB Decals tab, assign a non-electrical (no pin terminals) decal to the part.
   See also: Assigning PCB Decals to Part Types
7. On the Attributes tab, add attributes as necessary.
   See also: Defining Default Attributes for New Parts
8. Click **OK**.
   See also: Part Type Error Checking
9. In the Save Part Type to Library dialog box, type a name for the new part, and click **OK**.

Related Topics

Creating a New Part Type

Creating a Connector Part Type

Use this procedure to create a connector part type, which enables a feature in PADS Logic that splits the connector into multiple pin decals, making each pin its own gate.

Requirement

Your PCB decal(s) must exist to use this procedure. See Creating a New Decal.

Procedure

1. **File** menu > **Library**
2. In the Library Manager dialog box, select a library for the new part.
3. Click the **Parts** button, and then click the **New** button.
4. On the General tab of the Part Information dialog box, select the logic family; this sets refdes prefix.
5. Select the **ECO Registered Part** check box.
6. Select the **Special Purpose** check box, and click **Connector**.
7. On the PCB Decals tab, assign one or more decals to the part.
   See also: Assigning PCB Decals to Part Types

8. On the Pins tab, you must fill out the Pin Group and Name columns; the other columns are optional.
See also: Editing Pins Table Data.

9. On the Attributes tab, add attributes as necessary.
See also: Defining Default Attributes for New Parts

10. On the Connector tab, add pin decals to be used in the schematic.
See also: Assigning Special Symbols to a Connector.

11. Click **OK**.
See also: Part Type Error Checking

12. In the Save Part Type to Library dialog box, type a name for the new part, and click **OK**.

Related Topics

Creating a New Part Type

Modifying Part Types

You use the Part Information dialog box to modify the properties of an existing part type.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **Edit** button

When you have finished modifying the properties, click **OK** to save the changes.

See the following topics for more information:

- Assigning PCB Decals to Part Types
- Assigning CAE Decals to Gates
- Modifying the Pins Table Information
- Part Type Error Checking
- Adding and Modifying Part Type Attributes
- Assigning Special Symbols to a Connector
- Mapping Alphanumeric Pin Numbers to Numeric Decals

Assigning PCB Decals to Part Types

Filter your library search for decals and assign a decal and then alternates if desired.

Procedure

1. On the PCB Decals tab, to filter the contents of the Unassigned Decals list, use any of the following:

   a. In the Library list, select an individual library (or select **All Libraries)**.

   b. In the Filter box type wildcards.

      Tip: Type asterisk * in the Filter box to display all decals.

   c. Type a number in the Pin Count box, and then click **Apply**.

      Tip: Delete all numbers in the Pin Count box to display all decals. This box is always available as a filter to allow decals of differing pin counts to be assigned.

   d. Click the **Show only Decals with pin numbers matching Part Type** check box to filter out decals that do not have pin numbers matching existing gate and signal pins on the Pins tab, or the physical pin numbers on the Pin Mapping tab.

2. To assign one or more decals, select a decal in the Unassigned Decals list and click **Assign**. Assigned PCB decals can have a different number of pins.
   To assign a decal that does not exist in a library, click **Assign New. In** the Assign New PCB Decal dialog box, type the name and then click **OK**.

   Restrictions:

   - You can only assign up to 16 PCB decals to a part.
   - You must assign a decal with enough pins for all the defined gate pins and signal pins on the Pins tab.
   - Only decals with sequential numerical pin numbers can be used with pin mapping.

3. To change the order of decals in the Assigned Decals list, select the decal, and then click **Up** or **Down**.

   Tip: The decal at the top of the list is the default decal and is used when you add the part to the design.

Related Topics

Creating a New Part Type

Assigning CAE Decals to Gates

Type the name of a CAE Decal or use the Assign Decal to Gate dialog box to assign default and alternative CAE decals to gates.

Procedure

1. Select the cell under CAE Decal 1 and click the Edit button or double-click the cell under CAE Decal 1.
2. Type a decal name in the CAE Decal box or click the "..." Browse button to search for a decal from a library.
3. The Browse button opens the Assign Decal to Gate dialog box. To filter the contents of the Unassigned Decals list, use any of the following:
   a. In the Library list, select an individual library (or select **All Libraries)**.
   b. In the Filter box, type wildcards.

      Tip: Type asterisk * in the Filter box to display all decals.
   c. Type a number in the Pin Count box, and then click **Apply**. This box is read-only after a decal is assigned.

      Tip: Delete all numbers in the Pin Count box to display all decals. This box is always available as a filter to allow decals of differing pin counts to be assigned.
4. To assign one or more decals, select a decal in the Unassigned decals list and click **Assign**.
   To assign a decal that does not exist in a library, click **Assign New. In** the Assign New Gate Decal dialog box, type the name and then click **OK**.

   Restrictions:

   - You can assign up to four CAE decals to a part.
   - Assigned decals must have the same number of pins.
5. To change the order of decals in the Assigned Decals list, select the decal, and then click **Up** or **Down**.

   Tip: The decal at the top of the list is the default decal and is used when you add the part to the schematic.
6. Assign Gate pins on the Pins tab.

Related Topics

Creating a New Part Type

Modifying Gates in Parts in the Library in the *Concepts Guide*

Modifying the Pins Table Information

In this section:

- Adding a Series of Pins to the Pins Table
- Editing Pins Table Data
- Renumbering Pins in the Pins Table

Adding a Series of Pins to the Pins Table

You can use the Add Pins dialog box to add a series of pins the Pins table instead of adding one pin at a time.

Procedure

1. On the Pins tab of the Part Information dialog box, click the **Add Pins** button.
2. In the Add Pins dialog box, type the number of pins to add in the Number of pins box.

   Tip: Total pins for the part can not exceed 32,767 pins.
3. In the Start pin number area, type values in the Prefix and/or Suffix boxes. A preview of pin numbers based on your input is displayed below the boxes.

   Tips:

   - Alphabetic and numeric values can be used in either box. For example, A1 or 1A. If you type alphanumerics and the decal uses numerics, you must use the Pin Mapping tab to map the alphanumerics onto the decal.
   - For a single numeric, use either Prefix or Suffix box, and void the other box.
4. In the Increment options area, choose what to increment by clicking either **Increment prefix** or **Increment suffix**.
5. In the Step value box, type a positive or negative number by which to increase or decrease the pin numbers with consecutive or stepped values.
6. If using alphanumerics, you can select the Use JEDEC pin numbering check box to ensure that legal alphanumeric values are used.

Related Topics

Creating a New Part Type

Editing Pins Table Data

You can click a cell in the row of the pin you are editing to edit the cell contents, or select one or more cells of the same column and click the **Edit** button.

Procedure

1. Click the **Pin Group cell** and in the list, choose from gate, signal pin, or unused pin. Gates listed in the Pin Group cell list are added using the Gates tab. Signal pins require a signal name in the Name cell.
2. Click the **Number cell** and type a pin number for the pin.

   Requirement: The pin number must match the PCB decal. For example, alphanumeric to alphanumeric.

   Tip: Pin numbers can be either numeric or alphanumeric. Prior to PADS2007, alphanumeric pin numbers were not legal on the PCB decal but overlaid the numeric decal numbers, and were stored within the Part Type. You can continue to keep numeric PCB decals and use the Pin Mapping tab to overlay different pin numbers onto the numeric PCB decal pin numbers.
3. Click the **Name cell** and type a pin signal or function name. For example, “Clock” or “CLK.” A pin name is not required. The Name column is not used for unused pins.

   Restriction: Gate pin names can contain up to 40 characters, while signal pin names can contain up to 47. All alphanumeric characters are accepted. You cannot use special characters such as question marks ?, curly braces { }, asterisks *, periods ., commas , , or spaces.
4. Click the **Type cell** and in the list, choose a pin type. The type column is only used with gate pins.
5. Click the **Swap cell** and type a swap number, or use the up/down arrows.

   Tip: You swap pins within gates to uncross connections and facilitate routing. Pins with like numbers can swap within a gate. Type 0 to disable swapping.
6. Click the **Sequence (Seq.) cell** and type the gate sequence number. The sequence number determines the mapping of CAE gate pins to PCB decal pins. The sequence is automatically shared with alternate CAE decals. For example, it shows how pin numbers appear on the CAE gate decal; therefore, in Gate A, sequence number 1 could be pin 1, but in Gate B, sequence number 1 would be pin 4.

Exception: When editing pin data for connectors, only the Pin Group and Number columns are relevant. Data entered in other columns is rejected. Connectors don’t have gates, so the Pin Group column just indicates whether a pin is a connector pin or an unused pin.

Related Topics

Creating a New Part Type

Renumbering Pins in the Pins Table

You can renumber pins in the Pin tab table.

1. Select one or more cells in the Number column.
2. Click the **Renumber** button. In the Renumber Pins dialog box, the Number of pins box displays the number of pins selected for renumbering.
3. In the Start pin number area, type values in the Prefix and/or Suffix boxes. A preview of pin numbers based on your input is displayed below the boxes.

   Tips:

   - Alphabetic and numeric values can be used in either box. For example, A1 or 1A.
   - For a single numeric, use either Prefix or Suffix box, and void the other box.
4. In the Increment options area, choose what to increment by clicking either **Increment prefix** or **Increment suffix**.
5. In the Step value box, type a positive or negative number by which to increase or decrease the pin numbers with consecutive or stepped values.
6. If using alphanumerics, you can select the **Use JEDEC pin numbering** check box to ensure that legal alphanumeric values are used.

Related Topics

Creating a New Part Type

Part Type Error Checking

When you click Check Part, OK, Save As, or when you click a different tab, validation occurs and checks for the following error conditions:

- Empty pin numbers or pin numbers with embedded spaces or illegal characters. Duplicated Pin numbers
- Empty, duplicated, or non-sequential sequence numbers within a single gate
- Non-empty Type cells for signal pins or unused pins, or empty Type cells for gate pins
- Pin names with illegal characters for gate pins, net names with illegal characters for signal pins, non-empty name for unused pins, empty name for signal pins. Blank pin names are permitted for gate pins.
- Empty pin swap for gate pins, non-empty pin swap for signal and unused pins. Pin swap values for gate pins outside of the range 0 to 100.

Related Topics

Creating a New Part Type

Adding and Modifying Part Type Attributes

In this section:

- Adding Attributes to a Part Type
- Adding Height Information to Library Parts
- Defining Default Attributes for New Parts

Adding Attributes to a Part Type

You can add attributes to a part type.

Procedure

1. In the Part Information dialog box, click the **Attributes** tab.
2. Click Add to add a new attribute row.
3. Type the new attribute name and value.

   Alternative: Click the Browse Lib Attr button to select an existing attribute from a list.
4. Repeat Steps 2 and 3 to add additional attributes.

Adding Height Information to Library Parts

You can add height information to part types. Height information is used to prevent components being placed in height-constrained areas. It is also used when a design is exported to a 3-dimensional modeling application.

Procedure

1. In the Part Information dialog box, click the **Attributes** tab.
2. Click Add to add a new attribute row.
3. For the new attribute name type **Geometry.Height**.

   Alternative: Click the Browse Lib Attr button to select **Geometry.Height** from a list.
4. Type the value in current design units.

Defining Default Attributes for New Parts

You can save a default set of attributes to automatically use for each new part.

Restriction: Attribute names but not values are saved in the default attribute list.

Procedure

1. In the Part Information dialog box, click the **Attributes** tab.
2. Once the list of attributes is correct, click **Save As Default**.

Result

The default attribute list is saved to *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Settings\defaultattribute.txt*, which is shared between PADS Logic and PADS Layout.

Related Topics

Managing Library Attributes

Creating a New Part Type

Assigning Special Symbols to a Connector

Use the Browse for Special Symbols dialog box to assign one or more pin decals, or Special Symbols, to a connector. This allows you to use any of the special symbols as alternate pins of the connector. Instead of being displayed as a single symbol, the connector is broken up into individual pin symbols in the schematic. You can place the pins all together, or wherever you like on a single page or even across multiple pages.

Procedure

1. **Part Information for Part** dialog box > **Connector** tab.
2. To filter the Gate Decals list, select a library in the Library list, type wildcards in the Filter box, and then click **Apply**.

   Tip: To display all items, type asterisk * and click Apply.
3. Select the item in the list.
4. Click **OK**.

Related Topics

Creating a Connector Part Type

Creating a New Part Type

Mapping Alphanumeric Pin Numbers to Numeric Decals

There are several ways to complete the pin mapping.

Procedure

1. In the decal list above the preview window, select the assigned decal to which you want to map alphanumeric pins.
2. Map the pins using one of the following methods
   - **Using the Unmapped Pins list**—In the Unmapped Pins list, select one or more alphanumerics. Select one or the starting row in the Part Type column if you have a consecutive list to map. Click **Map**.
   - **Using the Part Type column—**Select a cell in the Part Type column and click the **Edit** button, or simply double-click the cell. Then type a pin number. The pin number is removed from the Unmapped pins list.
   - **Using the Preview Window—**Select an alphanumeric in the Unmapped Pins list and double click the pin in the decal preview window to map the alphanumeric to the pin. The next row in the Unmapped Pins list becomes the next selected alphanumeric for mapping.

     Tip: In the preview window, you can click and drag to define a zoom box, or use Shift+click or Shift+right-click to zoom in or out by a factor of two. You can zoom in up to 16X the original scale. The preview window will only zoom out to fit the decal entirely in the view.
   - **Using a Spreadsheet Application—**Click **Copy Map** to copy both columns of the mapping table. Paste the mapping table into Excel. Make mass edits. Copy the data from Excel and click **Paste Map** on the Pin Mapping tab.

     Restrictions: Copy Map and Paste Map only work with the whole pin mapping table and not selective rows.
3. Repeat as necessary.

Related Topics

Creating a Part Type with Different Schematic and Layout Pin Numbering

Creating a New Part Type

Saving Modified Decals and Parts to Libraries

Use the Save Part Types and Decals to Library dialog box to save modified decals or parts to libraries.

Procedure

1. Modify the decal or part in the PCB Decal Editor, and then return to the Layout Editor.
2. Select the components, right-click, and then click **Save to Library**.

3. Select items in the Part Types and Decals lists.

   Tip: If you created a new suffixed decal, meaning you changed the pad stack for one component only and the decal name has a unique suffix, the new decal also appears in the Decals list.

4. Select libraries to receive the information in the Part Type Library and Decal Library lists.

5. Click **OK**.

Related Topics

Creating a Basic Decal Automatically

Creating a Library

Importing and Exporting Libraries

Chapter 7
Working with the BGA Toolkit

To Add a BGA Pin Label

Use the Add BGA Pin Labels dialog box to label the die part's substrate bond pad. Usually labels match the name of the BGA pin to which they are connected.

Tips:

- You can select the pads individually, by group, or by die part.
- Selecting a die part lists all pins in the Connection multicolumn list.
- The substrate bond pad of the selected die part is highlighted in the **Die Pin#** column. The BGA pin labels are listed in the BGA Pad column.

To add a BGA pin label to a die part's substrate bond pad:

1. **BGA Toolbar** button > **Wire Bond Diagram** button > select a **BGA pin**.
2. Select the **Show only pins connected to multiple BGA pads** check box to display only die pins that are connected to multiple BGA pin pads in the **Connection** multicolumn list.
3. In the **Font** list, select the font you want to use.

   Tips:

   - Select stroke font or a system font.
   - For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.
4. In the **Layer** list, select the layer on which you want the BGA pin label.
5. In the **Size** box, type the size you want to use.
6. For stroke font, type the line width you want to use.
7. Click **OK**.

To Add Wire Bonds

You can add individual wire bonds only in the Wire Bond Editor. See “Using the Wire Bond Editor” for more information. You can add wire bonds to your design at any time, but you cannot place a wire by itself without bond pads. The bond pads must exist before you create a new wire bond.

Restriction: This information applies only to the BGA toolkit.

To add a new wire bond between pads:

1. Select **Add WB** on the shortcut menu when nothing is selected, or when a substrate or component bond pad is selected. The new wire bond dynamically attaches to the cursor.
2. Indicate the location at which to place the new wire by clicking on one substrate bond pad and one component bond pad.

Tip: You can attach more than one substrate bond pad to a component bond pad, and vice versa.

To create wire bonds and substrate bond pads for a new die, use the Wire Bond Wizard.

See also: Wire Bond Wizard Dialog Box

To Add Component Bond Pads

You can add component bond pads only in the Wire Bond Editor.

Restriction: This information applies only to the BGA toolkit.

See also: Using the Wire Bond Editor

To add a component bond pad:

1. With nothing selected, select **Add CBP** on the shortcut menu. The Add Component Bond Pad dialog box appears.
2. Enter the values for the new bond pad and click **Add**. The new bond pad dynamically attaches to the cursor.
3. Indicate the location at which to place the new pad.

To Add a Connection in BGAs

You can add connections manually or automatically. To automatically add connections, see Using the PADS Layout Route Wizard.

Restriction: This information applies only to the BGA toolkit.

To manually add a connection between pin pairs:

1. **BGA Toolbar** button > **Add Connection** button.

2. Select the first pin in the connection.
3. Select the second pin in the connection.

   If this pin is part of a different net or **Derive Net Name from Pin Function** is on, a prompt asks you to provide a new name for the combined nets.

   See also: Predefined Netnames in the *Concepts Guide*
4. Type a new name for the combined nets, if necessary.

To Add Die Parts from LIQ

Restriction: This information applies only to the BGA toolkit.

To add die parts from Library IQ:

1. **BGA Toolbar** button > **Add Die Parts** button

   The Add Die Parts dialog box appears.
2. Type the path to the folder where the die parts are located or use Browse to search for the folder. Select the die part on the **Die Part** list box. The die part appears in the preview area. The filename, modification date, number of chip bond pads, number of substrate bond pads, and number of wire bonds appear to the right of the preview area.
3. Click **Add**. The die part appears in the Die Part Type column.
4. Repeat steps 2 and 3 to add additional die parts.
5. Set the Die Data options as required.
6. Click **OK**.

To remove a die part from the Die Part Type column:

1. Select the die part in the Die Part Type column.
2. Click **Remove**.

For information on exporting die component data to Library IQ, see "To Synchronize Die Parts with LIQ."

To Add a Fanout

Adding a fanout adds a new substrate bond pad to the design and automatically connects it to the component bond pad with a wire bond.

Restriction: This information applies only to the BGA toolkit.

To add a fanout in the Wire Bond Editor:

1. Select a component bond pad > right-click > **Add Fanout**.
2. Enter the values for the new bond pad and click **Add** on the dialog box. The new wire bond and bond pad dynamically attaches to the cursor. (Add Fanout simultaneously creates a new wire bond and a new substrate bond pad.)
3. Indicate the location at which to place the new fanout.

To Add a Part in the BFA Toolkit

Restriction: This information applies only to the BGA toolkit.

To import and add a new component to a design:

1. **BGA Toolbar** button > **Add Component** button.
2. In the Get Part Type From Library dialog box select the part name in the **Part Types** list and click **Add**. The component attaches to the pointer.
3. Click **Close** to close the dialog box.
4. Move the part to the new location and indicate the new position. The new part is automatically annotated with the next available reference designation.

To Add Substrate Bond Pads

You can add substrate bond pads only in the Wire Bond Editor.

Restriction: This information applies only to the BGA toolkit.

See also: Using the Wire Bond Editor

There are two methods that you can use to add a substrate bond pad to your design:

Method 1

1. Select **Add SBP** on the shortcut menu when nothing is selected. The Add Substrate Bond Pad dialog box appears.
2. Enter the values for the new bond pad and click **Add** on the dialog box. The new bond pad dynamically attaches to the cursor.
3. Indicate the location at which to place the new pad.

Method 2

1. Select **Add Fanout** on the shortcut menu when a component bond pad is selected. The Add Substrate Bond Pad dialog box appears.

2. Enter the values for the new bond pad and click **Add** on the dialog box. The new wire bond and bond pad dynamically attaches to the cursor. (Add Fanout simultaneously creates a new wire bond and a new substrate bond pad.)

3. Indicate the location at which to place the new fanout.

Adding Routes in BGAs

Restriction: This information applies only to the BGA toolkit.

Note: When routing while the BGA toolbar is open, the pointer changes to a bull's-eye when it is near any eligible completion point, regardless of the net.

To manually enter routes that add to or change the current netlist:

1. **BGA Toolbar** button > **Add Route** button.

2. Select the pin on which to start the trace.

3. Add corners and vias as described in "Routing Manually."

4. Select the pin on which to end the trace.

   If this pin is part of a different net or **Derive Net Name from Pin Function** is on, a prompt asks you to provide a new name for the combined nets.

   See also: Predefined Netnames in the *Concepts Guide*

   If this pin is not part of the same net or has no netlist, all items are considered obstacles. To connect to the pin, right-click and click **Select Target** and select the pin on which you want to end the trace.

5. Netnames are automatically generated for each newly created connection or route.

   Type a new name for the combined nets, if necessary.

Note: You can also add routes automatically using the BGA Route Wizard.

To Adjust Focus for a Substrate Bond Pad

Adjusting focus realigns the substrate bond pad to the component bond pad after a move or rotation.

Restriction: This information applies only to the BGA toolkit.

To adjust focus for a substrate bond pad in the Wire Bond Editor:

1. Select the SBP.

2. Right-click to display the shortcut menu.

3. Select **Adjust Focus**.

To Assign CBPs to Rings

Restriction: This information applies only to the BGA toolkit.

To assign a component bond pad or pads to a ring or rings:

1. In the Assign CBPs to Rings dialog box, select the component bond pads that you want to assign to rings.
2. In the **Assign to** list, check the ring or rings to which you want to assign the selected component bond pad or pads.
3. Click **Apply.**

To Cancel the Connection Function

Restriction: This information applies only to the BGA toolkit.

To cancel the connection function when using **Add Connection** on the BGA toolbar:

1. Right-click to display the shortcut menu.
2. Click **Cancel**.

Checking Wire Bond Rules

Restriction: This information applies only to the BGA toolkit.

If in DRC On mode, the wire bond rules are checked along with other design rules (pad to pad, pad to trace, etc.). In case of DRC errors, a message appears on the status bar.

Most of the die modification operations, such as moving bond pads or adding wire bonds, check wire bond rules on-the-fly. During an operation, any unsatisfied rule results in a violation, and error marker appears at the cursor's current location.

Additionally, you can run wire bond rule batch checking at any time.

See also: Verify the Design

To Copy Component Bond Pads

You can copy component bond pads only in the Wire Bond Editor.

Restriction: This information applies only to the BGA toolkit.

See also: Using the Wire Bond Editor

To make a copy of an existing bond pad:

1. Select the bond pad to copy.
2. Select **Copy** on the Edit menu, or press **Ctrl+C**.

   Or

   Select **Copy** on the shortcut menu. The copy of the bond pad attaches to the cursor.
3. Indicate the position of the new bond pad.

The new bond pad is automatically annotated with the next available pin name.

Tip: Creating a copy of an existing substrate bond pad is not considered an engineering change and therefore does not require ECO mode.

To Copy Substrate Bond Pads

You can copy substrate bond pads only in the Wire Bond Editor.

Restriction: This information applies only to the BGA toolkit.

See also: Using the Wire Bond Editor

To make a copy of an existing bond pad:

1. Select the bond pad to copy.
2. Select **Copy** on the Edit menu, or press **Ctrl+C**.

   Or

   Select **Copy** on the shortcut menu. The copy of the bond pad attaches to the cursor.
3. Indicate the position of the new bond pad.

The new bond pad is automatically annotated with the next available pin name.

To Copy and Paste a Route in BGAs

You can reproduce repetitive routing patterns by creating a segment pattern, copying it, and pasting it on any similar connections. Layer changes and vias in the route are included in the selection. Memory patterns or SMD fanouts are ideal applications for copied routes.

Restriction: This information applies only to the BGA toolkit.

To copy and paste a route:

1. Select a segment or pin pair > **Edit** menu > **Copy**.

   Result: All included segments and vias are copied and attached to your pointer in Move mode.

2. Use the shortcut menu commands to rotate or flip the copy.

3. To place the copy, position it so that it joins the pins you want to paste to and click. A copy is pasted and another copy remains attached to the pointer so you can continue pasting.

   The pointer automatically snaps to a point the same distance, and in the same direction, as the last placement. This makes it easy to install repetitive route patterns.

4. When you finish pasting, press **Esc**.

See also: To Copy and Paste Trace Patterns

Copying Routes Creates New Netnames

Any time a route pattern is copied and the end points of the copied route terminate on unconnected pins, a pin pair is created between the pins. A new netname is automatically assigned in the format $$$1 or $$$2. You can also assign predefined net names using the gate pin name assigned to the part.

See also: Predefined Netnames in the *Concepts Guide*

To Create a BGA Design

Restriction: This information applies only to the BGA toolkit.

The following steps outline the basic workflow for creating a BGA design:

1. Use the Die Wizard to create a die and add it to the design.

   See also: To Create a New Die

2. Create and add a BGA decal using the PCB Decal Editor and the **BGA/PGA** tab in the Pin Wizards dialog box. Click **Substrate Decal Type**.

   See also: BGA/PGA Tab, Pin Wizard

3. Add the ball grid array to the design. You must add the BGA part to the BGA design.

   See also: To Add a Part in the BFA Toolkit

4. Create a substrate outline by clicking the **Board Outline and Cut Out** button on the Drafting toolbar.

5. Use the Wire Bond Wizard to define rings, establish wire bond rules, assign chip bond pads, and generate a wire bond fanout.

See also: Wire Bond Wizard Dialog Box, To Define Wire Bond Rules, To Assign CBPs to Rings, and To Create a Wire Bond Fanout

6. Reposition any die component's substrate bond pads or edit the die part's objects using the Wire Bond Editor.

   See also: To Move Bond Pads, To Edit Component Bond Pads, and To Edit Substrate Bond Pads

7. Check the wire bonds for conformance to wire bond rules.

   See also: Checking Wire Bond Rules

8. Add the traces between the die part's substrate bond pads and the BGA pin pads using **Add Route** on the BGA toolbar or the BGA Route Wizard.

   See also: Adding Routes in BGAs, Using the BGA Route Wizard

9. Create the die flag and power rings information using the Die Flag Wizard.

   See also: To Create a Die Flag and Rings

10. Add BGA pin labels to a die part's substrate bond pads using **Wire Bond Diagram** on the BGA toolbar.

    See also: To Add a BGA Pin Label

Related Topics

BGA Operations in the *Concepts Guide*

The BGA Tutorials. Click Help menu > Tutorial > PADS Layout Advanced Packaging Tutorial

To Create a Die Flag and Rings

Restriction: This information applies only to the BGA toolkit.

The die flag serves the following conductive and bonding functions for the die:

- Connecting to the back-based die, typically the ground
- Providing a heat sink and a pathway for heat dissipation
- Mounting the die to the substrate

One or more rings that typically provide power connections may surround the die flag. The rings may also have other purposes, for example, serving as ground connections or signal rings.

To create a die flag and rings:

1. Click the **BGA Toolbox** button on the toolbar.
2. Click **Die Flag** on the BGA toolbar.

3. Select a die component.
4. Make the appropriate selections as found in Die Flag Wizard Dialog Box.
5. Click **Create** to create the die flag and exit the Die Flag Wizard.

See also: Die Flag Wizard in the *Concepts Guide*

To Create a New Die

Restriction: This information applies only to the BGA toolkit.

To create a new die:

1. **BGA Toolbar** button > **Die Wizard** button.
2. In the Create Die dialog box, click either **From Text File**, **Parametrically**, or **From GDSII File** to begin creating a die.
3. For the procedure to create:
   - A die by importing a text (.csv) file, see "To Create a Die from a Text File."
   - A die from specifications you enter, see "To Create a Die Parametrically."
   - A die by importing a GDSII file, see "To Create a Die from a GDSII File."

To Create a Die from a Text File

Restriction: This information applies only to the BGA toolkit.

To create a die from a text file:

1. **BGA Toolbar** button > **Die Wizard** button.
2. Click From Text File to display the Die Wizard-Create from Text File dialog box.
3. Click **Browse** and select a text (.csv) file to open.
4. Optionally:
   - In Die Part Type, change the name of the type of die part you want to use to create the die.
   - Select Flip Chip to select the IC die for facedown mounting. Flip chip components contain only pins (SBPs).
   - Select Mirror Shapes (if you clicked Flip Chip) to mirror, or flip, the shapes (die geometrical data) on the die display.
   - Set preview colors.

5. Add or modify data on the five tabs. See:
   - To Define a Die Outline for the Die Data from a Text File.
   - To Modify CBP Shapes from a Text File.
   - To Modify the Numbering of CBPs from a Text File.
   - To Define Functions for Pads from a Text File.
   - To Define Preferences for Die Component Creation.

To Create a Die Parametrically

Restriction: This information applies only to the BGA toolkit.

To create a die parametrically:

1. **BGA Toolbar** button > **Die Wizard** button.
2. Click Parametrically to display the Die Wizard-Create Parametrically dialog box.
3. In Die Part Type, type the name of the type of die part you want to use to create the die.
4. Optionally:
   - Select Flip Chip to select the IC die for facedown mounting. Flip chip components contain only pins (SBPs).
   - Select Mirror Shapes (if you clicked Flip Chip) to mirror, or flip, the shapes (die geometrical data) on the die display.
   - Select the global unit type to use to convert system units into a commonly used set of measurements. All values are expressed on the die display in the units you choose. Available unit types are:

     Mils — Expressed in mils (1mil = 2.54*10^-5 m).

     Metric — Expressed in millimeters (1mm = 1.0*10^-3 m).

     Inches — Expressed in inches (1" = 2.54*10^-2 m).

     Tip: You cannot choose **Microns** to use as system units. Use **Metric** if the values in the imported file are expressed in microns.
   - Set the preview colors.
5. Add or modify data on the five tabs. See:
   - To Define a Die Outline Parametrically.
   - To Define a Set of CBPs Parametrically.
   - To Define the Numbering of CBPs Parametrically.

- To Define Functions for Pads Parametrically.
- To Define Preferences for Die Component Creation.

To Create a Die from a GDSII File

Restriction: This information applies only to the BGA toolkit.

To create a die from a GDSII file:

1. **BGA Toolbar** button > **Die Wizard** button.
2. Click From GDSII File to display the Die Wizard-Create from GDSII File dialog box.
3. Click **Browse** and select a GDSII (.gds) file to open.
4. Optionally:
    - Change the size of GDS units (microns).

      Tip: Because GDSII files come in different "flavors," you may need to adjust the unit settings in the file.
    - In Die Part Type, change the name of the type of die part you want to use to create the die.
    - Select Flip Chip to select the IC die for facedown mounting. Flip chip components contain only pins (SBPs).
    - Select Mirror Shapes (if you clicked Flip Chip) to mirror, or flip, the shapes (die geometrical data) on the die display.
    - Select the global unit type to use to convert system units into a commonly used set of measurements. All values are expressed on the die display in the units you choose. Available unit types are:

      Mils — Expressed in mils (1mil = 2.54*10^-5 m).

      Metric — Expressed in millimeters (1mm = 1.0*10^-3 m)

      Inches — Expressed in inches (1" = 2.54*10^-2 m).

      Tip: You cannot choose **Microns** to use as system units. Use **Metric** if the values in the imported file are expressed in microns.
    - Set your preview colors.
5. Add or modify data on the five tabs. See:
    - To Define a Die Outline from a GDSII File.
    - To Define a Set of CBPs from a GDSII File.

- To Define the Numbering of CBPs from a GDSII File.
- To Define Functions for Pads from a GDSII File.
- To Define Preferences for Die Component Creation.

To Create a Wire Bond Fanout

Use the Wire Bond Wizard to create a wire bond fanout.

Restriction: This information applies only to the BGA toolkit.

See also: Wire Bond Wizard in the *Concepts Guide*

1. Select a die component > **Wire Bond Wizard** button.
2. Use the Wire Bond Wizard dialog box to define settings for creation of the wire bond fanout.

   See also: Wire Bond Wizard Dialog Box

   The settings you define include:

   - Geometry of the SBP Guides.

     See also: SBP Guides in the *Concepts Guide*
   - Assignment of component bond pads to SBP rings
   - Wire bond preferences for the SBP rings, wire bond width, and substrate bond pad offset
   - Preferred spacing for the substrate bond pad and the wire bond to substrate bond pad spacing
3. To preview the wire bond fanout that corresponds to your settings, click **Preview Fanout**.
4. When you are satisfied with the fanout design, click **Create Fanout** to save the fanout in the design.

To Create a Wire Bond Report

You can create a report of connections between a die part's substrate bond pads and the BGA pads using a Basic script provided with PADS Layout.

Restriction: This information applies only to the BGA toolkit.

- Open the **Basic Scripting** dialog box and click **Wire Bond Report**.

See also: Basic Scripts Dialog Box

To Cycle a Bond Pad

Restriction: This information applies only to the BGA toolkit.

To cycle, or select a bond pad on a different layer, in the Wire Bond Editor:

1. Select the bond pad.
2. Right-click to display the shortcut menu.
3. Select **Cycle**.

To Cycle Through Wire Bonds

Restriction: This information applies only to the BGA toolkit.

To cycle through wire bonds in close proximity in the Wire Bond Editor:

1. Select the wire bond.
2. Right-click to display the shortcut menu.
3. Select **Cycle**.

To Define a Die Outline for the Die Data from a Text File

The text file format does not contain the description of the die outline. After the Die Wizard obtains all the data from the text file, the die outline is calculated automatically as an extent box that covers all of the CBPs from the text file. You can modify this automatically-generated die outline using the Die Size tab.

Restriction: This information applies to only the BGA toolkit.

To define a die outline for the die data from a text file:

1. Follow the directions in "To Create a Die from a Text File."
2. Type or change values in the following fields:

   Die Size Length — The length of the die, which corresponds to the X size.

   Die Size Width — The width of the die, which corresponds to the Y size.

   Die Size Height — The height of the die, which defines the thickness of the die.

   Die Outline Base Point — The die point (center, lower left, upper left, upper right, or lower right) for which you want to specify the coordinates for the base point, as expressed in X and Y.

X — The die point along the x-axis you want as the base point for X.

Y — The die point along the y-axis you want as the base point for Y.

3. Add or modify data on the remaining tabs, if necessary. See:
    - To Modify CBP Shapes from a Text File.
    - To Modify the Numbering of CBPs from a Text File.
    - To Define Functions for Pads from a Text File.
    - To Define Preferences for Die Component Creation.
4. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Define a Die Outline from a GDSII File

Restriction: This information applies only to the BGA toolkit.

To define a die outline from a GDSII file:

1. Follow the directions in "To Create a Die from a GDSII File."
2. Select the Die Size tab and either set the die size and die outline position manually, or Select a GDS shape representing the die outline.

   Tip: You can modify the Height control when you click Select from GDS Shape, but you cannot modify the other parameters in the Set Size area.

    - To set the die outline manually, click **Set Size**. Type or select values in the following fields:

      Die Size Length — The length of the die, which corresponds to the X size.

      Die Size Width — The width of the die, which corresponds to the Y size.

      Die Size Height — The height of the die, which defines the thickness of the die.

      Die Outline Base Point — The die point (center, lower left, upper left, upper right, or lower right) for which you want to specify the coordinates for the base point, as expressed in X and Y.

      X — The die point along the x-axis you want as the base point for X.

      Y — The die point along the y-axis you want as the base point for Y.

    - If you click **Select from GDS Shape**, specify the shape from the GDSII file to use as the die outline by selecting values in the following fields:

      GDS Layer — Shows the layers defined in the GDSII file. Use to select the layer on which the die outline shape is located.

GDS Shape — Shows the names of all the GDS shapes available for selection as the die outline shape. Only closed, filled shapes from the GDSII file that are also on the selected layer in the GDS Layer list appear.

The shapes appear in the die display area using the Shapes on Selected Layer color. The die display adjusts dynamically as you choose values, to help you select the proper shape.

3. Add or modify data on the remaining tabs, if necessary. See:
    - To Define a Set of CBPs from a GDSII File.
    - To Define the Numbering of CBPs from a GDSII File.
    - To Define Functions for Pads from a GDSII File.
    - To Define Preferences for Die Component Creation.
4. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Define a Die Outline Parametrically

Restriction: This information applies to only the BGA toolkit.

To define a die outline parametrically:

1. Follow the directions in "To Create a Die Parametrically."
2. Type or change values in the following fields on the Die Size tab:

    Die Size Length — The length of the die, which corresponds to the X size.

    Die Size Width — The width of the die, which corresponds to the Y size.

    Die Size Height — The height of the die, which defines the thickness of the die.

    Die Outline Base Point — The die point (center, lower left, upper left, upper right, or lower right) for which you want to specify the coordinates for the base point, as expressed in X and Y.
3. Add or modify data on the remaining tabs, if necessary. See:
    - To Define a Set of CBPs Parametrically.
    - To Define the Numbering of CBPs Parametrically.
    - To Define Functions for Pads Parametrically.
    - To Define Preferences for Die Component Creation.
4. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Define a Set of CBPs from a GDSII File

Use the **CBP** tab to filter the GDS shapes you want to become the CBP shapes. You can filter shapes by size, by layer, and by position relative to the die outline.

Restriction: This information applies only to the BGA toolkit.

Tip: Only closed, filled shapes from the GDSII file (**BOUNDARY** and **RECT GDS** types) are available.

To define a set of CBPs from a GDSII file:

1. Follow the directions in "To Create a Die from a GDSII File."
2. Type or select data in the following fields:

   Min Size — Sets the minimum size you want to use to filter GDS shapes for CBPs, expressed in the current system units.

   Max Size — Sets the maximum size to use to filter GDS shapes for CBPs.

   GDS Layer — Shows the layers defined in the GDSII file. Select the layer on which the die outline shape is located.
3. Click Shapes Inside Outline to include only the shapes found inside the die rectangle for the filtering.
4. Select the pad shape.

   Depending on the Pad Shape value (rectangular or oval), the circumscribed shape is used to derive the CBP shape from the GDS shape.
5. Add or modify data on the remaining tabs, if necessary. See:
   - To Define a Die Outline from a GDSII File.
   - To Define the Numbering of CBPs from a GDSII File.
   - To Define Functions for Pads from a GDSII File.
   - To Define Preferences for Die Component Creation.
6. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Define a Set of CBPs Parametrically

Restriction: This information applies only to the BGA toolkit.

To define a set of CBPs parametrically:

1. Follow the directions in "To Create a Die Parametrically."
2. Do one of the following on the CBP tab:
   - Specify the total pad count and allocate a fixed percentage of the pads as ground and power pads in GND % and PWR %, as follows:

     Total — Type or select the total pad count for the die. If you modify the total pad count, the pads are evenly distributed along the four sides.

     GND % — Type or select the percentage of the total pin count to allocate as ground pads. If you modify the value, the ground pads for each side are derived from the Total count.

     PWR % — Type or select the percentage of the total pin count to allocate as power pads. If you modify the value, the power pads for each side are derived from the Total count.
   - Enter the total pad count for each side and allocate a number of pads for each side as ground or power pads.

     Total Pads — View or double-click to change the total number of pads for each side of the die. If you modify a value, the values in Total, GND %, and PWR % adjust accordingly.

     GND — View or double-click to change the total number of ground pads for each side of the die. If you modify a value, the values in Total, GND %, and PWR % adjust accordingly.

     PWR — View or double-click to change the total number of power pads for each side of the die. If you modify a value, the values in Total, GND %, and PWR % adjust accordingly.
3. Type or select data in the following fields:

   Pad Pitch — Defines the distance between pads. The distance is measured from the left side of one pad to the left side of the next adjacent pad.

   Row Pitch — Defines the distance between rows to control creation of staggered row patterns. For a single row in a straight line around the sides of the die, enter 0. To create staggered row patterns, enter a specific positive or negative number.

   Distance from Die Edge — Defines the distance between the row of the pads and the die edge.

   Pad Shape — Defines the shape of the pads: rectangle or oval.

   Pad Length — Type or select the value for the pad length.

   Pad Width — Type or select the value for the pad width.
4. Add or modify data on the remaining tabs, if necessary. See:

- To Define a Die Outline Parametrically.
- To Define the Numbering of CBPs Parametrically.
- To Define Functions for Pads Parametrically.
- To Define Preferences for Die Component Creation.

5. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Define Functions for Pads from a GDSII File

You can import pad functions from a text file or from the GDS text on a layer.

Restriction: This information applies only to the BGA toolkit.

To define functions for pads from a GDSII file:

1. Follow the directions in "To Create a Die from a GDSII File."
2. Select either From Text File or From GDS Texts on Layer to indicate which one you want to use to assign functions.

   If you select From Text File, click **Browse** to select the file you want to use to assign pad functions.

   If you select From GDS Texts on Layer, click the GDS layer you want to use to get the GDS text items which specify pad functions.
3. Click **Assign** to assign the pad functions.
4. Add or modify data on the remaining tabs, if necessary. See:
   - To Define a Die Outline from a GDSII File.
   - To Define a Set of CBPs from a GDSII File
   - To Define the Numbering of CBPs from a GDSII File.
   - To Define Preferences for Die Component Creation.
5. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Define Functions for Pads from a Text File

Restriction: This information applies only to the BGA toolkit.

To define the functions for pads from a text file:

1. Follow the directions in "To Create a Die from a Text File."
2. Optionally, select the Pad Functions tab and change function names by double-clicking the one you want to change and typing the new name.
3. Add or modify data on the remaining tabs, if necessary. See:
   - To Define a Die Outline for the Die Data from a Text File.
   - To Modify CBP Shapes from a Text File.
   - To Modify the Numbering of CBPs from a Text File.
   - To Define Preferences for Die Component Creation.
4. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Define Functions for Pads Parametrically

On the Pad Functions tab, the pads will obtain the following functions by default:

- Ground pins: <GND Pad Function>
- Power pins: <PWR Pad Function>
- Signal pins: <Signal Function Prefix><Pad #>

Restriction: This information applies only to the BGA toolkit.

To define functions for pads parametrically:

1. Follow the directions in "To Create a Die Parametrically."
2. Optionally, on the Pad Functions tab, to edit the function of an individual pad, change function names by double-clicking the one you want to change and typing the new name.
3. Optionally, to redefine functions for all pads, assign different function names to Signal Function Prefix, GND Pad Function, or PWR Pad Function using the Assign Functions area of the tab. Assign names as follows:

   Signal Function Prefix — Type or view the function prefix you want to use to derive the signal pad names.

   GND Pad Function — Type or view the function name you want to use for the ground pads.

   PWR Pad Function — Type or view the function name you want to use for the power pads.

   If you change the default signal function prefix or the power or ground function, click Assign to assign the new names to all pads.
4. Add or modify data on the remaining tabs, if necessary. See:
   - To Define a Die Outline Parametrically.
   - To Define a Set of CBPs Parametrically.
   - To Define the Numbering of CBPs Parametrically.
   - To Define Preferences for Die Component Creation.
5. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Define Preferences for Die Component Creation

Restriction: This information applies only to the BGA toolkit.

To define preferences for creating die components from a text file:

1. Follow the directions in either To Create a Die from a Text File, To Create a Die Parametrically, or To Create a Die from a GDSII File.
2. Select the **Die Prefs** tab and select either Add Part to Design or Save to Library.
    - To add the part to the design currently open, select Add Part to Design and type the part name. The part name is the reference designator that is automatically assigned to the new die component in the design. To change the reference designator, click on the part name and enter a new name.
    - To add the part to a library, select Save to Library, then select the library in which you want to save the part.
3. Select the layer for the die outline and pads.
4. Add or modify data on the remaining tabs, if necessary. See:
    - To Define a Die Outline for the Die Data from a Text File.
    - To Modify CBP Shapes from a Text File.
    - To Modify the Numbering of CBPs from a Text File.
    - To Define Functions for Pads from a Text File.

To Define the Numbering of CBPs from a GDSII File

Restriction: This information applies only to the BGA toolkit.

To define the numbering of CBPs from a GDSII file:

1. Follow the directions in "To Create a Die from a GDSII File."

2. Select the Pad # tab and select either circular or JEDEC numbering:

Circular Circular numbering

JEDEC JEDEC numbering
Pin rows are lettered from top to bottom starting with A and pin columns are numbered from left to right starting with 1. The letters I, O, Q, S, X, and Z are not used. For arrays with more than 20 rows, row 21 is designated AA and subsequent rows are designated AB, AC, etc.

If you select **Circular Numbering**, select values for the following:

- Direction::

Clockwise Numbering begins with pad 1 and continues in a clockwise direction.

CCW Numbering begins with pad 1 and continues in a counterclockwise (CCW) direction.

- Pad 1 Side:1

Left Places pad 1 on the left side of the design.

Top Places pad 1 on the top of the design.

Right Places pad 1 on the right side of the design.

Bottom Places pad 1 on the bottom of the design.

- Pad 1 Location:

Center Numbers the center pad of the specified side of the design as pad 1.

Left Numbers the leftmost pad of the specified side of the design as pad 1.

Right Numbers the rightmost pad of the specified side of the design as pad 1.

Specify Numbers the pad you select on the specified side of the design as pad 1. Type or select the pad number you want to use for pad 1.

3. Add or modify data on the remaining tabs, if necessary. See:

- To Define a Die Outline from a GDSII File.
- To Define a Set of CBPs from a GDSII File.
- To Define Functions for Pads from a GDSII File.

- To Define Preferences for Die Component Creation.

4. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Define the Numbering of CBPs Parametrically

Restriction: This information applies only to the BGA toolkit.

To define the numbering of CBPs parametrically:

1. Follow the directions in "To Create a Die Parametrically."
2. Select the Pad # tab, then select either circular or JEDEC numbering:

Circular	Circular numbering
JEDEC	JEDEC numbering Pin rows are lettered from top to bottom starting with A and pin columns are numbered from left to right starting with 1. The letters I, O, Q, S, X, and Z are not used. For arrays with more than 20 rows, row 21 is designated AA and subsequent rows are designated AB, AC, etc.

If you select **Circular Numbering**, select values for the following:

- Direction::

Clockwise	Numbering begins with pad 1 and continues in a clockwise direction.
CCW	Numbering begins with pad 1 and continues in a counterclockwise (CCW) direction.

- Pad 1 Side:1

Left	Places pad 1 on the left side of the design.
Top	Places pad 1 on the top of the design.
Right	Places pad 1 on the right side of the design.
Bottom	Places pad 1 on the bottom of the design.

- Pad 1 Location:

Center	Numbers the center pad of the specified side of the design as pad 1.

Left	Numbers the leftmost pad of the specified side of the design as pad 1.
Right	Numbers the rightmost pad of the specified side of the design as pad 1.
Specify	Numbers the pad you select on the specified side of the design as pad 1. Type or select the pad number you want to use for pad 1.

3. Add or modify data on the remaining tabs, if necessary. See:
   - To Define a Die Outline Parametrically.
   - To Define a Set of CBPs Parametrically.
   - To Define Functions for Pads Parametrically.
   - To Define Preferences for Die Component Creation.
4. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Define Wire Bond Rules

Each die part in a design has its own separate set of wire bond rules. If a rule is not set for the current die, the rule is not checked. Use the Wire Bond Rules Dialog Box to define wire bond rules for a die part.

Restriction: This information applies only to the BGA toolkit.

You can set wire bond rules for a die both in Layout Editor and in the Wire Bond Editor.

To set wire bond rules for the current die part in the Wire Bond Editor:

1. With nothing selected, select **Wire Bond Rules** on the shortcut menu. The Wire Bond Rules dialog box appears.
2. Enter values for the minimum and maximum length, maximum angle, and clearance or import the rule values from a Library IQ preset file.

To set wire bond rules for a die part in Layout Editor:

1. Select a substrate bond pad of a die and select **Wire Bond Rules** on the shortcut menu. The Wire Bond Rules dialog box appears.
2. Enter values for the minimum and maximum length, maximum angle, and clearance or import the rule values from a preset file.

Tip: Wire bond rules are saved with the design file. Also, if you save a die part to the library, the wire bond rules for the die part are saved for use in other designs.

To Delete a Bond Pad

Restriction: This information applies only to the BGA toolkit.

To delete a bond pad in the Wire Bond Editor:

1. Select the bond pad.
2. Right-click to display the shortcut menu.
3. Select **Delete**.

To Delete a Connection in BGAs

Use the **Delete Connections** button to delete a pin pair, disconnect a pin from a net, or split a net into two different nets. This also deletes any test point vias that belong to the deleted connection.

Restriction: This information applies only to the BGA toolkit.

To delete connections in a BGA:

1. **BGA Toolbar** button >**Delete Connection** button.

   Tip: Test point vias that belong to the deleted connection are also deleted.

2. The following operations are available:

Delete Pin Pair	Select the connection or trace between the pin pair to delete it. If the net contains only one pin pair, selecting either pin deletes the pin pair.
Disconnect a Pin	Select the pin to disconnect. If a net contains more than one pin pair, a prompt appears to confirm the disconnection, along with a check box to delete the route segment going to the pin.
Split a Net	Select the connection between the pin pair where the net should be split. If a Color by Net setting exists for a net that is split, the new net will use the same color setting.

Note: The maximum netname length is 47 characters. You can use any alphanumeric characters except brackets {}, asterisks *, spaces, question marks, or commas.

To Delete a Net in BGAs

Restriction: This information applies only to the BGA toolkit.

To delete all pin pairs and remove all pins from a net:

1. **BGA Toolbar** button > **Delete Net** button.
2. Select a pin, unrouted pin pair, trace, or via in the net to delete. A confirmation prompt appears.
3. Click **Yes** to proceed with the delete.

Test point vias that belong to the deleted net are also deleted.

To Delete a Wire Bond

Restriction: This information applies only to the BGA toolkit.

To delete a wire bond in the Wire Bond Editor:

1. Select the wire bond.
2. Right-click to display the shortcut menu.
3. Select **Delete**.

To Derive Net Name from Pin Function

The **Derive Net Name from Pin Function** command on the **Add Connection** shortcut menu is a toggle. When selected, it uses a pin's function as a basis for naming a new net when you are adding connections manually.

Restriction: This information applies only to the BGA toolkit.

To select or clear **Derive Net Name from Pin Function** when using **Add Connection** on the BGA toolbar:

1. Right-click to display the shortcut menu.
2. Select **Derive Net Name from Pin Function** to toggle the setting.

To Display Die Pins

Restriction: This information applies only to the BGA toolkit.

To display die pins that are connected to multiple BGA pin pads in the **Connection** multicolumn list:

1. **BGA Toolbar** button > **Wire Bond Diagram** button.

2. Select the die part's substrate bond pad to which you want to add a BGA pin label. Select the pads individually, by group, or by die part. The Add BGA Pin Labels dialog box appears.
3. Double-click in the box next to the die pin number for which you want to view pad labels. The "…" Browse button appears.
4. Type the new label or click **Browse**. The Pads for Die Pin dialog box appears.
5. In the **BGA Pad** list, select the labels that you want to use.
6. Click **OK**. You return to the Add BGA Pin Labels dialog box.
7. Click **OK**.

To Edit Component Bond Pads

You can edit component bond pads only in the Wire Bond Editor.

Restriction: This information applies only to the BGA toolkit.

See also: Using the Wire Bond Editor

Edit component bond pads using the CBP Properties dialog box or the shortcut menu.

To edit component bond pads:

1. Select the bond pad.
2. Right-click and click **Properties**.
3. Modify the properties of the component bond pad, or Wire Bond rules, as needed.

To Modify a Decal in BGAs

Use Edit Decal to modify the decal of a selected part in the design.

Restriction: This information applies to only the BGA toolkit.

To modify the decal of a selected part:

1. Select a part > right-click > **Edit Decal**.
2. Modify the decal.
3. Click **Exit Decal Editor** from the **File** menu. The message "Do you want to apply the changes to all components with decal TMP or just the selected components?" appears in the dialog box.
4. Click **All** to replace all decals or **Selected** to replace selected decals.

To Edit Substrate Bond Pads

You can edit substrate bond pads using the SBP Properties dialog box or the shortcut menu.

Restriction: This information applies only to the BGA toolkit.

You can use the SBP Properties dialog box in Layout Editor and in the Wire Bond Editor.

To edit substrate bond pads:

1. Select the bond pad.
2. Right-click and click **Properties**.
3. Modify the properties of the substrate bond pad, or the Wire Bond rules, as needed.

To Edit the Die Size

You can edit the size (outline) of a die only in the Wire Bond Editor.

Restriction: This information applies only to the BGA toolkit.

See also: Using the Wire Bond Editor

To edit the size of a die:

1. While in the Wire Bond Editor and with nothing selected, select **Edit Die Size** on the shortcut menu. The Edit Die Size dialog box appears.
2. Adjust the die size by typing or selecting values in **Length**, **Width**, and **Height**.
3. Click **OK** to apply the changes and close the dialog box.

Editing Wire Bonds

You can edit wire bonds only in the Wire Bond Editor.

Restriction: This information applies only to the BGA toolkit.

See also: Using the Wire Bond Editor

To Generate Connections

Restriction: This information applies only to the BGA toolkit.

See also: BGA Route Wizard dialog box

To generate connections only, or generate connections and routes:

1. **BGA Toolbar** button > **Route Wizard** button.
2. To generate connections only, click **Generate Connections** from the **Action** area of the BGA Route Wizard dialog box.

To generate connections and routes, click **Generate Connections and Route** from the **Action** area of the BGA Route Wizard dialog box.

1. If generating connections only, set strategy parameters and preferences on the **Connections** tab.

   If generating connections and routes, set strategy parameters and preferences on the **Routing** tab.
2. On the **Select Pads** tab, set the pads to include in processing.
3. On the **BGA Fanouts** tab set the BGA fanout options.
4. Click **Run** to begin processing.

Result: When processing completes, the BGA Route Wizard report (brw_report.lst) appears.

You can interrupt processing by pressing **Esc**. If you interrupt processing, the message “Do you want to interrupt processing?” appears. You can:

- Click **Undo changes** to undo the changes made up to this point.
- Click to clear **Undo changes** to retain the current changes, but remove temporary single pin nets from the design.
- Click **Open BGA Route Wizard Dialog** to open the BGA Route Wizard when you click **OK**.
- Click **Cancel** to continue processing from the point at which you stopped.

Related Topics

BGA Route Patterns in the *Concepts Guide*

To Import the SBP Functions

Restriction: This information applies only to the BGA toolkit.

Follow these steps to copy the values that you imported from the netlist file into the SBP Function column of the SBP Properties dialog box.

1. Click **Derive from Netlist** on the SBP Properties dialog box.
2. When the Load Netlist ASCII File dialog box appears, specify an input file for the substrate bond pad functions.
3. In the Derive SBP Function from Netlist dialog box, select a component from the Select Component list.
4. Click **OK**.

See also: Exporting an ASCII File

To Import Wire Bond Rules

Restriction: This information applies only to the BGA toolkit.

To import wire bond rules from Wire Bond Wizard:

1. On the Wire Bond Rules dialog box, click Import. This Imports wire bond rules saved in the Wire Bond Wizard dialog box from the Wire Bond Wizard Setup file, and assigns the values as rules for the currently open die part when you click **OK** in the Wire Bond Rules dialog box.
2. Select the rules file (.wbw file) you want to use.

To List All BGA Pin Labels

Restriction: This information applies only to the BGA toolkit.

To list all BGA pin labels in the Add BGA Pin Labels dialog box:

1. **BGA Toolbar** button > **Wire Bond Diagram** button.
2. Select the die part's substrate bond pad to which you want to add a BGA pin label. Select the pads individually, by group, or by die part. The Add BGA Pin Labels dialog box appears.

3. Double-click the **BGA Pad** column of the **Connection list** and click the selection button.
4. In the Pads for Die Pin dialog box, select **Apply to All**.
5. Click **Select All**.

To List Specific BGA Pin Labels

Restriction: This information applies only to the BGA toolkit.

To list specific BGA pin labels in the Add BGA Pin Labels dialog box:

1. **BGA Toolbar** button > **Wire Bond Diagram** button.
2. Select the die part's substrate bond pad to which you want to add a BGA pin label. Select the pads individually, by group, or by die part. The Add BGA Pin Labels dialog box appears.
3. Double-click the **BGA Pad** column of the **Connection list** and click the selection button.
4. In the Pads for Die Pin dialog box, select **Apply to All**.
5. Select the pad positions in BGA Pad.

To Modify CBP Shapes from a Text File

Restriction: This information applies only to the BGA toolkit.

To modify CBP shapes from a text file:

1. Follow the directions in "To Create a Die from a Text File"
2. Either leave the pad shape as it is or change it on the CBP tab.

   To change the pad shape, click Override Pad Shape, then type or change values in any or all of the Pad Shape, Pad Length, or Pad Width fields.

   Tip: You cannot modify the shape of an individual CBP. The pad shape you choose on the CBP tab will apply to all CBPs.
3. Add or modify data on the remaining tabs, if necessary. See:
   - To Define a Die Outline for the Die Data from a Text File.
   - To Modify the Numbering of CBPs from a Text File.
   - To Define Functions for Pads from a Text File.
   - To Define Preferences for Die Component Creation.

4. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Modify the Numbering of CBPs from a Text File

Restriction: This information applies only to the BGA toolkit.

To modify the numbering of CBPs from a text file:

1. Follow the directions in "To Create a Die from a Text File."
2. Either leave the pad numbers as they are or change them on the Pad # tab.

   To change the pad numbers, click Override Pad Numbers.

   Select either circular or JEDEC numbering:

Circular	Circular numbering
JEDEC	JEDEC numbering Pin rows are lettered from top to bottom starting with A and pin columns are numbered from left to right starting with 1. The letters I, O, Q, S, X, and Z are not used. For arrays with more than 20 rows, row 21 is designated AA and subsequent rows are designated AB, AC, etc.

   If you select **Circular Numbering**, select values for the following:

   - Direction::

Clockwise	Numbering begins with pad 1 and continues in a clockwise direction.
CCW	Numbering begins with pad 1 and continues in a counterclockwise (CCW) direction.

   - Pad 1 Side:1

Left	Places pad 1 on the left side of the design.
Top	Places pad 1 on the top of the design.
Right	Places pad 1 on the right side of the design.
Bottom	Places pad 1 on the bottom of the design.

   - Pad 1 Location:

Center	Numbers the center pad of the specified side of the design as pad 1.

Left	Numbers the leftmost pad of the specified side of the design as pad 1.
Right	Numbers the rightmost pad of the specified side of the design as pad 1.
Specify	Numbers the pad you select on the specified side of the design as pad 1. Type or select the pad number you want to use for pad 1.

3. Add or modify data on the remaining tabs, if necessary. See:
   - To Define a Die Outline for the Die Data from a Text File.
   - To Modify CBP Shapes from a Text File.
   - To Define Functions for Pads from a Text File.
   - To Define Preferences for Die Component Creation.
4. Optionally, select colors that help you preview your die design. See "To Set Die Preview Colors."

To Move Bond Pads

You can move substrate bond pads in the Layout Editor and in the Wire Bond Editor.

See also: To Move a Die Component Substrate Bond Pad

You can move component bond pads only in the Wire Bond Editor.

See also: Using the Wire Bond Editor

You can move component or substrate bond pads whether DRC is on or off. However, with DRC on you cannot place a bond pad if the attached wire bonds violate any design rules, including wire bond rules.

Restriction: This information applies only to the BGA toolkit.

To move a bond pad:

1. Select the bond pad to move.
2. Select **Move** on the shortcut menu.

   Tip: Right-click and select a command to edit the selected bond pad, to set the pad pitch, or select **WB Rules** to adjust the wire bond rules.
3. Indicate a new location for the bond pad.

To Move a Die Component Substrate Bond Pad

In the Layout Editor, a substrate bond pad is considered a pin. To select a substrate bond pad, turn on the Pins option in the selection filter.

Restriction: This information applies to only the BGA toolkit.

See also: Using the Selection Filter

To move a substrate bond pad in the Layout Editor:

1. Select a die substrate bond pad and select Move SBP on the shortcut menu. The bond pad dynamically attaches to the cursor.

   Tip: Right-click to edit the selected bond pad, adjust the wire bond rules associated with the die part, or adjust the focus, rotate, or spin the bond pad before placing it.

   If Keep Focus is selected, the rotation angle of the substrate bond pad is automatically adjusted to match the direction of the wire bond during the move operation.

   The following actions will occur if the mentioned selections are chosen (Die Component Tab > Options dialog box):

   - If Snap SBP to Guide is selected, the position of the SBP will automatically snap to the nearest SBP Guide if it is within the distance specified in Snap Threshold. This enables snap-to-guide mode when you are moving substrate bond pads. Select or type, in **Snap Threshold**, the number of units that defines the threshold below which snap to guide operates.

     See also: SBP Guides in the *Concepts Guide*

     The threshold value applies to all SBP guides for all die components in the design.

     If you are moving multiple SBPs, each SBP is snapped to the nearest guide independently.

   - If Show SBP Clearance is selected, the clearance outline is added to the display, showing the clearance area that must exist between two SBPs. This shows the SBP clearance outline around moved substrate bond pads when you are moving, adding, spinning SBPs, or adding fanouts. The clearance value is derived as follows:

   - If the SBP is not part of a design net, or if its net and pin pair do not have specific rules defined, then the SMD to SMD clearance is used from the Default Rules hierarchy.

   - If the SBP layer has a specific clearance rule set under layer rules, then the specific SMD to SMD rule is used.

     See also: The Formatting for Conditional Clearance Rules in the ECO File

- If the SBP is part of a design net, and the related net class, net, group, or pin pair has a specific clearance rule set, then the specific SMD to SMD rule is used.
- If the net class, net, group, or pin pair has a layer-specific clearance rule set, then the clearance assigned to the SBP layer is used.

  The SBP clearance outline appears in both DRC On and DRC Off modes.

  See also: Design Rule Checking in the *Concepts Guide*
- If Show Wire Bond Length and Angle is selected, the length of the wire bond and the size of the wire bond angle appear on the display as WB Length and WB Angle, and change dynamically when you move the SBP. This shows the wire bond length and angle when you are moving, adding, spinning SBPs, or adding fanouts or wire bonds.
- If you are moving multiple SBPs, WB Length and WB Angle appear only for the first selected SBP.

  If the SBP has no attached wire bond, the text does not appear. If the SBP has several wire bonds, the displayed values apply only to the first wire bond in the database.

  WB Length and WB Angle appear in both DRC On and DRC Off modes.

2. Indicate the new bond pad location.

Tip: Move SBP is available only for pins of die parts created by the BGA tools in PADS Layout. For pins of other parts, Move SBP is unavailable.

Related Topics

Design Rule Checking in the *Concepts Guide*

To Remove (Backup) the Last Added Connection

The Backup command on the Add Connection shortcut menu removes the last connection you added. Repeat this command to continue removing connections.

Restriction: This information applies to only the BGA toolkit.

To remove the last added connection when using Add Connection on the BGA toolbar:

1. Right-click to display the shortcut menu.
2. Select **Backup**.

To Rename a Net in BGAs

Restriction: This information applies to only the BGA toolkit.

To change the name of a net:

1. **BGA Toolbar** button > **Rename Net** button.
2. Select a pin, unrouted pin pair, trace, or via in the net to rename.
3. A prompt asks for the new name. Type in the name, and click **OK**.

Tip: The maximum netname length is 47 characters. You can use any alphanumeric characters except brackets {}, asterisks *, spaces, question marks, or commas.

To Rename the Current Net

The Rename Current Net command on the Add Connection shortcut menu renames the net currently selected.

Restriction: This information applies to only the BGA toolkit.

To rename the currently selected net when using Add Connection on the BGA toolbar:

1. Right-click to display the shortcut menu.
2. Select **Rename Current Net** to display the Rename Net dialog box.
3. Enter the new name for the current net.

To Rotate a Substrate Bond Pad 90 Degrees

Restriction: This information applies only to the BGA toolkit.

To rotate a substrate bond pad 90 degrees in the Wire Bond Editor:

1. Select the SBP.
2. Right-click to display the shortcut menu.
3. Select **Rotate 90**.

To Select a Component Bond Pad

Restriction: This information applies only to the BGA toolkit.

To select one or more component bond pads in the Wire Bond Editor:

1. Right-click the die component to display the shortcut menu.
2. Select **Select CBPs**.
3. Select the CBP.

   Tip: Ctrl+click to select more than one CBP.

To Select a Substrate Bond Pad

Restriction: This information applies only to the BGA toolkit.

To select one or more substrate bond pads in the Wire Bond Editor:

1. Right-click the die component to display the shortcut menu.
2. Select **Select SBPs**.
3. Select the SBP.

   Tip: Ctrl+click to select more than one SBP.

To Select a Wire Bond

Restriction: This information applies only to the BGA toolkit.

To select one or more wire bonds in the Wire Bond Editor:

1. Right-click the die component to display the shortcut menu.
2. Select **Select WBs**.
3. Select the wire bond.

   Tip: Ctrl+click to select more than one wire bond.

To Select Any Die Component

Restriction: This information applies only to the BGA toolkit.

To select any die component in the Wire Bond Editor:

1. Right-click the die component to display the shortcut menu.

2. Select **Select Anything**.
3. Select the die component.

Selecting Die Part Items

When you start the Wire Bond Editor, the selection filter for this mode replaces the standard Pads Layout selection filter.

Restriction: This information applies only to the BGA toolkit.

The Wire Bond Editor's selection filter confines selection to component bond pads, substrate bond pads, and wire bonds. Choose whether to select any one or all of these object types (Select Anything) on the shortcut menu.

See also: To Edit Substrate Bond Pads, To Edit Component Bond Pads

Select one or more substrate bond pads, component bond pads, or wire bonds within the die to perform editing on these object types. Only items in the currently selected die part can be selected.

You can also edit the die size.

See also: To Edit the Die Size

To Set Die Preview Colors

Restriction: This information applies only to the BGA toolkit.

To set die preview colors, which help you see the die display more clearly:

1. Click Preview Colors in the Die Wizard to display the Preview Colors dialog box.
2. Set a color by clicking it, then click the tile to the right of the area or item you want to color.

   You can set colors for any or all of the following:

   Background — Sets the background color in the die display area.

   Highlight — Sets the highlight color in the die display area.

   Die Outline — Sets the color of the die outline in the die display area.

   CBP — Sets the color of the CBPs in the die display area.

   CBP # — Sets the color of the CBP numbers in the die display area.

All Shapes — Sets the color of all GDS shapes in the die display area. The GDSII shapes area of the dialog box is active only when you are creating a die from a GDSII file.

Shapes on Selected Layers — Sets the color of the GDS shapes in the GDSII file that appear on the selected GDS Layer, in the die display area. The color appears only when the Die Size tab or the Pad Functions tab is active.

The GDSII shapes area of the dialog box is active only when you are creating a die from a GDSII file.

To Set Pad Pitch for a Substrate Bond Pad

Restriction: This information applies only to the BGA toolkit.

To set the pad pitch for a substrate bond pad in the Wire Bond Editor:

1. Select the SBP.
2. Right-click to display the shortcut menu.
3. Select **Set Pad Pitch** to open the Set Pad Pitch dialog box.
4. Type the distance to use for spacing pads uniformly along their guides. The pad pitch spacing is center-to-center.

To Set SBP Names

Restriction: This information applies only to the BGA toolkit.

1. Click **SBP Names** on the Wire Bond Wizard dialog box.
2. In the SBP Properties Dialog Box make selections for substrate bond pads and their functions.
3. To import the SBP functions from a netlist file, click **Derive** from Netlist.

See also: Exporting an ASCII File

To Spin a Substrate Bond Pad

Restriction: This information applies to only the BGA toolkit.

To spin, or freely rotate, a substrate bond pad in the Wire Bond Editor:

1. Select the SBP.
2. Right-click to display the shortcut menu.

3. Select **Spin**. Cross hairs appear intersecting the center of the bond pad.
4. Indicate the new pad position.

To Start the Wire Bond Editor

Restriction: This information applies only to the BGA toolkit.

1. To start the Wire Bond Editor, click the **Wire Bond Editor** button on the BGA toolbar and select a die component: a substrate bond pad, a component bond pad, or a wire bond. Once you select a die component, the Wire Bond Editor mode box appears.
2. Begin modifying the die part items.

   To modify a die part item, select the item (substrate bond pad, component bond pad, or wire bond) and select an action on the shortcut menu.
3. To exit the Wire Bond Editor and return to the Layout Editor, click **Exit Wire Bond Editor** on the mode box.

Tip: You can undo edits made in the Wire Bond Editor, even if you have exited the mode.

To Step and Repeat a Route

Restriction: This information applies to only the BGA toolkit.

To step and repeat a copied route:

1. Select a segment or pin pair > **Edit** menu > **Copy**.

   Result: All included segments and vias are copied and attached to your pointer in Move mode.
2. Use the shortcut menu commands to rotate or flip the copy.
3. To place the copy, position it so it joins the pins you want to paste to and click. A copy is pasted and another copy remains attached to the pointer so you can continue pasting.
4. Right-click and click **Repeat**.
5. Type a repetition amount and click **OK**.

   The route is copied and pasted the specified number of times. Each pasted route is spaced the same distance, and in the same direction, as the initial copied route.

   Tip: New netnames may be created. See Copying Routes Creates New Netnames.
6. When you finish pasting press **Esc**.

See also: Step and Repeat Dialog Box

To Swap Pins in BGAs

Restriction: This information applies to only the BGA toolkit.

To swap the nets on two pins:

1. **BGA Toolbar** button > **Swap Pin** button.
2. Select a pin. All candidates for swapping are highlighted. Candidates with a nonzero swap ID that matches the selected pin's swap ID are highlighted in the highlight color. Candidates with different or undefined swap IDs are highlighted in a complementary color.
3. Select one of the highlighted pins to swap with. If you select a pin that has a matching, nonzero swap ID, the swap is performed.

Tips:

- You cannot swap pins owned by a pin pair with rules.
- If Stretch Traces During Component Move is selected in the Design page of the Options dialog box, traces attached to the swapped pins are rerouted. Otherwise, trace segments are unrouted before the swap is performed.

To Synchronize Die Parts with LIQ

Restriction: This information applies to only the BGA toolkit.

Use Synchronize Die Part to:

- Update die parts in Pads Layout with die part data from Library IQ.
- Update die parts in Library IQ with die part data from Pads Layout.
- Export LIQ data to an .liq file that Library IQ can read. See "Using the Synchronize Die Part Dialog Box" for more information.

Note: Beginning with PADS 9.0, die parts and flip chips are identified by the Special Purpose settings in the Part Type rather than by the DIE and FLP logic families. With this change, *any* reference designator (logic family) can be assigned to a die part or flip chip. If you export a design to LIQ, and you have die parts or flip chips of a family other than DIE or FLP in your design, remember that *all* parts lose their family designation when exported to LIQ, and are assigned either the DIE or FLP family when imported back into Pads Layout; so the original family designation (and reference designator) of these parts is lost in the export/import process.

To Update Die Parts in BGAs

Restriction: This information applies to only the BGA toolkit.

To update die part data in Pads Layout with die part data from Library IQ:

1. **BGA Toolbar** button > **Synchronize Die Part** button

   The Synchronize Die Part dialog box appears.

2. Select the Library IQ die part to synchronize from the Die Part list box.
3. In the Design list box, click **Part Type** and choose the die part to update with the selected Library IQ die part data, or click **Component** and select the component reference designator to update with the selected Library IQ die part data.
4. Click **Update Design**.

   If you selected Part Type in step 3, the message "Update selected Part Type(s) from LIQ Part Type XXXX?" appears. If you selected Component in step 3, the message "Change Part Type of the selected component(s) to the LIQ Part Type XXXX?" appears.

5. Click **OK** to update the die part. Click **Cancel** to cancel the update.

Tip: Updating Die Parts in Pads Layout is allowable only for DRC Off. If DRC On is set, the message "Updating Die Part in Design is allowable only for DRC Off. Switch to DRC Off?" appears. Click OK to turn DRC off and update the die part in Pads Layout. Click Cancel to cancel the update.

Creating Die Information

For background on creating a die, see Creating Die Information in the *Concepts Guide*.

If you import an ASCII file to create the die part definition, see Die Data ASCII File Format in the *Concepts Guide*.

Restriction: This information applies only to the BGA toolkit.

For information on how to create a die, see:

- To Create a Die from a Text File
- To Create a Die Parametrically
- To Create a Die from a GDSII File

Selecting Component Bond Pads

Restriction: This information applies only to the BGA toolkit.

Use any of the following methods to select a component bond pad to edit:

- With nothing selected, choose **Select Anything** or **Select CBPs** on the shortcut menu, and select the component bond pad.
- Select **Select CBPs** on the shortcut menu and area select to edit multiple bond pads.
- With a substrate bond pad or wire bond selected, click **Select CBP** on the shortcut menu.
- Select the substrate bond pad to which the component bond pad is connected. Right-click and click **Properties**. Click **CBP**.
- Select the wire bond to which the component bond pad is connected. Right-click and click **Properties**. Click **CBP**.

Selecting Substrate Bond Pads

When in the Layout Editor, a substrate bond pad is considered a pin. To select a substrate bond pad, turn on the **Pins** option in the selection filter and select the substrate bond pad.

Restriction: This information applies only to the BGA toolkit.

When in the Wire Bond Editor, use any of the following methods to select a substrate bond pad to edit:

- With nothing selected, click **Select Anything** or **Select SBPs** on the shortcut menu, and select the substrate bond pad.
- Select S**elect SBPs** on the shortcut menu and area select to edit multiple bond pads.
- With a component bond pad or wire bond selected, click **Select SBP** on the shortcut menu.
- Select the component bond pad to which the substrate bond pad is connected. Right-click and click **Properties**. Click **SBP**.
- Select the wire bond to which the substrate bond pad is connected. Right-click and click **Properties**. Click **SBP**.

Using the Define Name of New Net Dialog Box

Use the New Net dialog box to add a new net name to the Net list. The Die Flag Wizard adds the new net to the design after you click **Create**.

Restriction: This information applies only to the BGA toolkit.

Wire Bond Report

Wire bond reports provide information on wire bond rules violations or compliance. When you check wire bond rules, a wire bond report is automatically saved to a file named wbr_report.lst. The default location for the file is C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Settings. The report appears in Notepad (or in the default text editor chosen during installation).

Restriction: This information applies only to the BGA toolkit.

The following is an example of a wire bond report:

Wire Bond Rule Checking Report - previewbgadieflag.pcb - Wed Feb 09 14:01:45 2000

```
Checking Die Part U1 <MWG122160ECG> <MWG122160ECG> ...
Wire Bond Rules
Min Length: 30
Max Length: 175
Max Angle: 45.00
WB to WB Clearance: 1
WB to SBP Clearance: 1
Die Part U1: NO errors found
```

Tip: The value “Not Set” appears for rules that are not set in the Wire Bond Rules dialog box.

To Update Die Parts in Library IQ

Restriction: This information applies to only the BGA toolkit.

To manually update die part data in Library IQ with die part data from Pads Layout:

1. **BGA Toolbar** button > **Synchronize Die Part** button

   The Synchronize Die Part dialog box appears.

2. In the Design list box, click **Part Type** and select the die part to synchronize, or click **Component** and select the reference designator of the die part to synchronize.

3. In the Die Part list box, select the die part to update with the selected die part data.

   Tip: A die part can be updated from only one part in the design. If you choose multiple design parts, the first part selected is used to update the Die Part.

4. Click **Update LIQ**. The Confirm Update LIQ dialog box appears.

5. Click **Save Backup** to save a copy of the die part before the update.

6. Use Browse to navigate to a different folder in which to save the backup file.

7. Click **OK** to update the die part. Click **Cancel** to cancel the update.

Tip: You can also export LIQ data to an .liq file that Library IQ can read.

See also: Synchronize Die Part Dialog Box

Action

Generate Connections	Generates logical connections between die and BGA component pins. Connections are not routes; however, BGA fanouts and SBP fanouts are generated if specified in the Connections tab.
Generate Connections and Route	Generates logical connections between component pins and creates trace patterns to route them. During processing, the following occurs:

- BGA fanouts are generated. This is performed on double-sided designs only.
- Serpentine routes and plating tails are generated.
- SBPs are logically assigned to the ends of serpentine traces and logical connections between die pins, and BGA pins are generated.
- SBP fanouts and any-angle coupling traces are generated.

  The BGA Route Wizard generates route patterns on die and BGA layers only. The BGA layer contains only BGA fanouts. All other parts of route patterns (serpentine routes and plating tails, SBP fanouts and any-angle coupling traces) are created on the die layer.

See also: BGA Route Patterns in the *Concepts Guide*

BGA Reference Designator

Displays the BGA reference designator when you open the BGA Route Wizard dialog box for the first time (per design), based on the logic families in the current design.

The BGA Route Wizard looks for a BGA logic family. If no die with this logic family is found, a component is selected randomly. If more than one die has this logic family, one of the dies in that logic family is randomly selected.

BGA Route Wizard Report

Displays information on the connection made during processing, the number of pins selected for connection per side or quadrant, and new netlist information.

Die Reference Designator

Displays the die reference designator when you open the BGA Route Wizard dialog box for the first time (per design).

The BGA Route Wizard looks for a die part or flip chip. If more than one of these is found, one of them is randomly selected. If neither is found, a component is randomly selected.

Select Sides/Select Quadrants

The available partitioning type depends on whether the **Generate Connections** or **Generate Connections and Route** option is selected in the Action area.

Use the partition check boxes to select partition sets. The available Side partition sets are **Right**, **Left**, **Top**, and **Bottom**. The available Quadrant partition sets are **Top Left**, **Top Right**, **Bottom Left**, and **Bottom Right**.

Only the pins from selected sets appear in the **Substrate Bond Pad** and **BGA Pins** lists.

See also: BGA Operations in the *Concepts Guide*

Undo Last Run

Returns the design to its state prior to route processing. **Undo Last Run** is unavailable until you run the routing process.

Using the BGA Route Wizard

The BGA Route Wizard provides automated features that reduce repetitive, and often tedious, design tasks including creating connections between BGAs pads and SBPs, creating BGA fanouts, routing die-to-BGA connecting traces, and generating plating tails.

When you click the BGA Route Wizard button on the BGA toolbar, the BGA Route Wizard dialog box appears. Use the BGA Route Wizard dialog box to generate connections only, or generate connections and routes.

Restriction: This information applies to only the BGA toolkit.

To Use the Dynamic Route Tool in BGAs

Restriction: This information applies to only the BGA toolkit.

To start the Dynamic Route tool:

1. Set **On-line Design Rules Checking** to **Prevent Errors** in the On-Line DRC area on the Design page of the Options dialog box.
2. Make sure you are on the correct layer for the preferred direction.
3. Select a segment or pin, as you would using the Routing Manually.
4. Right-click and click **Dynamic Route**. The trace attaches to the pointer.
5. Guide the pointer through the items you want to bypass without entering corners. You can enter corners with a right-click if you want to; these act as tacked corners to prevent preceding segments from moving.

   Tip: Avoid unnecessary mouse movements since corners are added at all mouse locations. To remove unwanted corners, slowly move the pointer back over the unwanted traces.

   As you route, the guard band appears whenever the head of the trace meets a clearance obstacle that it cannot shove. The dynamic route tool won't allow you to complete the trace without changing the clearance rules or removing the obstacle. The pointer changes to a bull's-eye when it is close enough to the finishing pad to complete the route.

   Tip: Use Transparent Mode, modeless command T, to view obstacles that may lie under traces on the active layer.

See also: To Reroute with Route or Dynamic Route, Routing Dynamically

Routing in DRC Prevent Mode

When you route a trace between pins that are not part of the same net or that have no netlist, all items not belonging to the current net are considered obstacles. To connect to a pin:

- Right-click and click **Select Target** and select the pin on which you want to end the trace.

Using the Wire Bond Editor

In the Layout Editor, only substrate bond pads are available for selection and modification. The Wire Bond Editor opens (explodes) a selected die part, which allows you to move, add, delete, and edit individual component bond pads and wire bonds in addition to substrate bond pads.

Restriction: This information applies only to the BGA toolkit.

You can edit the die size by selecting **Edit Die Size** on the Wire Bond Editor's shortcut menu.

See also: To Edit the Die Size

The Wire Bond Editor is available only for die parts designed in the BGA Toolkit. You can edit only one die part at a time.

Tip: You cannot edit the die outline by selecting it using the Wire Bond Wizard.

To Use Graphical Selection Mode

Result: The BGA Route Wizard dialog box appears.

As an alternative to using the Substrate Bond Pads and BGA Pins lists to select pins for processing, you can use Graphical Selection mode to select pins with the mouse.

Restriction: This information applies to only the BGA toolkit.

To use Graphical Selection mode:

1. **BGA Toolbar** button > **Route Wizard** button.
2. Click the **Select Pads tab**.
3. Click **Select Graphically**. The BGA Route Wizard temporarily closes and the Select Graphically dialog box appears.
4. Click either **Select Pins to Include** or **Select Pins to Exclude**.
5. Select the SBP pins and BGA pads you want to include or exclude. Use standard selection procedures to select pins.

   While in Graphical Selection mode you can use standard zooming features to zoom and pan the work area. All other functionality is unavailable while you are in this mode.

   Tip: If Existing Nets on the Connections tab or Routing tab is cleared, you can select only pads that are not currently part of a predefined connection.
6. Click **OK** on the **Select Graphically** dialog box to complete your pin selection and return to the BGA Route Wizard. The Substrate Bond Pads and BGA Pins lists on the Select Pads tab reflect the inclusions and exclusions you made.

Wire Bond Fanout Workflow

Restriction: This information applies only to the BGA toolkit.

To create a wire bond fanout for a selected die:

1. Use the Wire Bond Wizard dialog box to define substrate bond pad rings and their properties.

   See also: Wire Bond Wizard Dialog Box
2. Use the Rules dialog box to set design rules and the Wire Bond Rules dialog box to set wire bond rules for creating the wire bond fanouts.

See also: Design Rule Hierarchy in the *Concepts Guide*, Wire Bond Rules Dialog Box

3. Use the Assign CBPs to Rings dialog box to specify which component bond pad to wire bond onto which ring.

   See also: Assign CBPs to Rings Dialog Box

   Tip: For each CBP you assign to a ring, an SBP and a wire bond are created. New SBPs are automatically preplaced on the guide of the specified ring or rings.

4. Preview the fanout that corresponds to the current settings by clicking **Preview Fanout**.

   Based on the previewed fanout, you can modify SBP Guides properties, component bond pad assignments, and other properties until the wire bond fanout pattern satisfies the rules you have set.

5. Create the wire bond fanout by clicking **Create Fanout**, which stores the fanout pattern in the design.

Chapter 8
File Operations

In this chapter:

Creating New Files

You use the New command to create a new design file.

To create a new file:

1. **File** menu > **New**.
2. In the Set Start-up File dialog box, select the start-up file.

   Tip: A start-up file contains global settings such as layer definitions, grids, clearance rules, the attribute dictionary, and so on.
3. If you want to use this file for all new design files, select the **Don't Display Again** check box.
4. Click **OK**.

Related Topics

Creating Start-up Files

Importing a Schematic Design Netlist

The process of laying out a design in PADS Layout typically begins with importing a *netlist file* containing all the schematic information for the design, including a list of all the parts and their decals. During the import process, all the parts are sourced from the library, and their decals are stacked at the origin, ready for placement.

Requirement

For a netlist import to succeed, all of the parts and decals listed in the netlist must exist in your library. If any are missing, the import operation writes an ascii.err file listing the missing parts/decals. To import the netlist successfully, you must add these components to your library, either by adding a library which contains the missing components to your library list, or by creating the missing part types and missing decals.

Procedures

The procedure you use to import a netlist depends on which schematic tool you are using. For product specific instructions, see one of the following topics:

- Creating a New PCB Layout from a DxDesigner Design
- Creating a New PCB Design from a PADS Logic Netlist
- Creating a New PCB Design from an OrCAD Netlist

Creating a New PCB Design from an OrCAD Netlist

When you lay out a new PCB design, you typically start by importing a netlist created in a schematic capture application. This topic explains how to import a netlist created in ORCAD.

Prerequisites

The ORCAD netlist file must exist.

Procedure

1. Change the filename extension of the netlist created by OrCAD from .net to .asc. PADS Layout only imports netlist files with .asc extensions.
2. In a text editor, open the netlist file and examine the header of the file. The header is the first line of text in the file. This can be a problem in the OrCAD netlist depending on the .dll file you use to create the netlist. It must be a compatible PADS header. If necessary, replace the header with a header from a current PADS .asc file. To get a current PADS ascii header:
    a. Open PADS Layout and click **File menu** > **Export**.

b. In the File Export dialog box, type a name for the exported file and note its location, then click Save. In the ASCII Output dialog box, in the Sections area, select the Text checkbox. (You need to select at least one of the checkboxes to output an ascii file.)

c. Click OK.

d. Using a text editor, open the .asc file you created. Copy the header from the sample .asc file you created and paste it into your OrCAD netlist file, replacing the original header. The following is an example of a header from the 2007 series software.

```
!PADS-POWERPCB-V2007.0-BASIC! DESIGN DATABASE ASCII FILE 1.0
```

3. In PADS Layout, click **File menu** > **Import**.
4. In the File Import dialog box, in the Files of type list, click **ASCII Files (*.asc)**. Browse for and select the OrCAD netlist file, and then click **Open**.
5. If an ascii.err file is produced, perform the following steps:

a. Close the design into which you just imported the netlist. Click No when you are prompted to save the design.

b. Add any missing components listed in the ascii.err file to your library, either by adding a library which contains the missing components to your library list, or by creating the missing part types and decals. (See Adding Libraries to the Library List, Creating and Modifying Part Types, and Creating and Editing PCB Decals for information on how to do these tasks.)

c. Resolve any other errors found in the ascii.err file.

d. When all the errors have been resolved, repeat this procedure.

Result

The import process sources all the part types and decals from the library and stacks the decals at the origin. You can then use the Tools menu > Disperse Components command to spread out the components.

Cross-Probing

You can cross-probe with PADS Logic or DxDesigner to simplify importing the netlist and placing the parts.

Procedures

The procedure you use to cross-probe depends on which schematic tool you are using. For product specific instructions, see one of the following topics:

- Cross-probing Between PADS Products in the *PADS Logic User's Guide*

- Cross-Probing with DxDesigner

Creating Start-up Files

You use the Start-up File Output dialog box to create a start-up file that contains global settings such as layer definitions, grids, clearance rules, the attribute dictionary, and so on. You can create different start-up files and specify which one to use when creating a new design file. This capability enables you to save setup time when creating a new design by reusing global settings that you saved in the start-up file.

See also: Specifying the Start-up File

Procedure

1. In the current design, specify the options and preferences you want to include in the start-up file.

2. On the File menu, click **Save As Start-up File**.

3. In the Save As dialog box, specify the start-up file name and click **Save**. The Start-up File Output dialog box appears.

4. In the Sections area, select the check box for any of the following settings you want to include in the start-up file:

    - **PCB Parameters**—Global information, such as colors, layer definitions, and grids

    - **Vias**—Via information, such as default via type, jumpers, padstack definitions and locations

    - **Layer Data**—Layer information specified in the Layers Setup dialog box, such as number of layers, layer names, routing direction for the layer, electrical type, and associations

    - **Rules**—Rules information, such as clearance, routing, and high-speed

    - **CAM**—CAM information related to the plot file configurations

    - **Attributes**—Attribute information, such as the attribute dictionary, all attributes assigned to objects in the design, and attribute status (read only, system, ECO registered, or hidden). Values in the attribute hierarchy are not saved.

      See also: Modifying Default Attributes

    Tip: If you want to select all the check boxes, click the Select All button.

5. In the Units list, select the units you want to use in the start-up file.

    Tip: Current units provide more information than Basic units, such as grid positions.

6. In the Start-up File Description box, type a brief description of the global settings you are saving. The description appears when you select the start-up file in the Set Start-up File dialog box, and should help remind you of the settings in the start-up file.

7. Click **OK**.

Result: The start-up file is written to the *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Settings* folder. Start-up files have a .stp file extension.

Related Topics

Specifying the Start-up File

Start-up Files in the *Concepts Guide*

Specifying the Start-up File

You use the Set Start-up File dialog box to select the start-up file to use for new design files. A start-up file contains global settings such as layer definitions, grids, clearance rules, the attribute dictionary, and so on.

The start-up file affects only new designs and specifying a new start-up file does not affect existing designs.

Procedure

1. **File** menu > **Set Start-up File**.

2. In the Set Start-up File dialog box, select the start-up file and click **OK**.

Tip: If you want to use the selected start-up file for all new design files, select the Don't Display Again check box.

Related Topics

Creating Start-up Files

Start-up Files in the *Concepts Guide*

Opening Files

PADS Layout can open the following file types:

Native Design Files, *.pcb	PADS Layout binary format
Reuse Files, *.reu	A physical design reuse type, known as the reuse definition

Old Binary Files, *.job	PADS Perform V6 and PADS Work V7 format .job files

When you open a file, PADS Layout may convert the data to current formats.

Restriction: Files opened by another user are locked to any edits.

Tip: Improvements in thermal relief definition in PADS 9.2 may cause changes in thermal pad appearance, flooding results and CAM output (for CAM Plane layers) when a design from a previous version is opened in PADS 9.2. If incompatibilities are found when such a file is opened, a warning prompt is displayed and the incompatibilities are written to the Powerpcb.Rep file.

Procedure

1. **File** menu > **Open**.

   Result: The File Open dialog box shows files in the default \PADS Projects folder. The list of files includes those created in PADS Layout.

2. On the File Open dialog box, select the file and click **Open**.

Related Topics

File Open Conversions in the *Concepts Guide*

Replacing Fonts

You can set up text strings and labels in your designs to use stroke font or the system fonts that are installed on your system.

When you open a design created with fonts that are not installed on your system, the Font Replacement dialog box opens automatically.

Tip: If the design uses fonts or character sets that are not installed on your system, empty boxes will appear where you expect to find text or symbols. Once the font replacement process completes, the symbols display properly.

There are three types of font replacement:

- Automatic Font Replacement
- Manual Font Replacement
- Skipping Font Replacement

Tips:

- You can select some fonts for automatic replacement, select others for manual replacement, and choose that other font replacements be skipped entirely.
- You can have a combination of stroke font and system fonts within the same design.
- You must set up fonts for each text string and/or label you create in your design. Once you set up fonts for a text string or label, you can then use the Properties dialog to apply a font and font characteristics to all objects that you select for modification with the Properties dialog box.

Restrictions:

- If the design uses fonts or character sets that are not installed on your system, a font substitution process begins automatically when the file is loaded. During this process, you are asked to choose fonts to substitute for those that are missing from your system.
- System font text is supported in RS274X Gerber format when Fill mode is on. System font text is output to Gerber format as a set of filled polygons.
- System fonts are not supported in the RS-274D CAM output format. If you attempt to use this format with system fonts, the program displays a warning message. If you proceed, system fonts will not be output. Instead, you should use the 274X format with system fonts.
- Type 1 fonts are not supported.

Automatic Font Replacement

To replace fonts automatically:

1. In the Font Replacement dialog box, select **Automatic** to use standard Windows font substitution.

   Tip: The Missing font column lists the fonts used in the design that are not installed on your system.
2. In the Replace with font column, select the fonts you want to use as replacements for those fonts you identified for replacement.
3. Click **OK**.

Manual Font Replacement

To replace fonts manually:

1. In the Font Replacement dialog box, select **Manual** to select font substitutions manually.
2. In the Missing font column, select the name of the font you want to replace.

3. In the Replace with font column, select the fonts you want to use as replacements for those fonts you identified for replacement.
4. Click **OK**.
5. Click **OK** again to confirm that the font replacement process should begin.

Skipping Font Replacement

- In the Font Replacement dialog box, select **Skip** in the Mode column to preserve the original font settings in the design.

Tip: When you skip font replacement, clearances for text strings are preserved in the design; these objects display as empty text boxes.

Related Topics

Finding Fonts

Saving Files

To save changes you made to a design information file:

- Click **Save** from the **File** menu. You can create native format, binary design files (*.pcb files).

Restriction: Files opened by another user are locked to any edits.

You can also save by clicking the **Save** button from the standard toolbar.

You cannot save .job format files with PADS Layout. Also, flooding and hatching are not saved with files. You must flood or hatch again the next time you open the file.

Related Topics

To Save As

To Create a Test Point ASCII File

To Save As

To write design information to a different file name or folder:

1. Click **Save As** from the **File** menu.
2. You can change the file name or folder.
3. To change the file name, type the new file name.

4. To select a different folder to which to save the file, use the options at the top of the dialog box.

Related Topics

Saving Files

To Create a Test Point ASCII File

Archiving Your Design

You can create a folder, a PDF, and/or a zip file that contains all of your design files and supporting files. This includes the design itself, a schematic file, libraries, and any additional files or folders you want. You choose what to archive; all fields are optional.

Tip: As an alternative to the PADS Layout Archiver, you can use the Archive Navigator to archive your projects. .

Procedure

1. **Open the design you want to archive.**
2. **File** menu > **Archive.**
3. Select the files and folders you want in the Archiver dialog box.
4. Click **OK**.

Results

An archive folder that contains the design and/or schematic files, the libraries, and any additional files and folders you've indicated is created.
Exception: If you chose to compress the files, the .zip file is the only file in this folder.

If you chose to create a PDF, the file is created using the design name and placed in the archive folder.

If you chose to compress the files using the zip format, a zip file is created having a filename of the following format:

```
<project_name>YYYYMMDDHHMMSS.zip
```

Where YYYY is the year, MM is the month, DD is the day, HH is the hour - in military time, MM is the minute, and SS is the second of the exact time you created the file. The file contains the same folder structure as the archive folder.

Chapter 9 Archive Navigator

Use the Archive Navigator to create, organize and manage your PCB project archives. You can:

- Organize archives into three levels of storage—vaults, folders and projects.
- Store successive numbered archives of a project with identifying data, including a text description.
- Restore any project archive from the vault to a working directory, make changes, and store the modified project as a new archive.
- Create archive templates to simplify customized archiving.

Tip: As an alternative to the Archive Navigator, you can use the PADS Layout Archiver to archive your projects. See Archiving Your Design.

This chapter contains the following topics:

Setup and Vault Management

You can organize your Archive Navigator storage to suit your situation. If you have only a few projects, you can store them at the top level of the Archive Navigator vault. If you have a larger number of projects, you may want to create folders within your vault to organize them. If you have very many projects, you may want to create multiple vaults, each containing multiple folders containing multiple projects.

To begin archiving a PCB project you must:

- Create a vault.

- Create an empty Project container in the vault, and associate it with a working folder containing the PCB project files.

In this section:

Creating a Vault

To begin using the Archive Navigator, the first thing you must do is create a vault.

Tip: You can create more vaults later if you need them.

Procedure

1. Vault view > **Select Vault** button

   Tip: Click Tools menu > Archive Navigator to open the Archive Navigator.

2. In the Select Vault dialog box, click **Create New Vault**.
3. In the Browse for Folder dialog box, select an empty folder in which to place the new vault, and click **OK**.

   Tip: If you haven't already created an empty folder, click **Make New Folder** to create one.

4. Click **OK** in the Select Vault dialog box.

Result

You can now add an empty project to the vault and link it to your working folder.

Adding a Project Container to the Vault

To begin archiving a PCB project in the Archive Navigator, you must create an empty Project container for it in the vault, and associate the Project container with a working folder containing the PCB project.

Tip: If you have a pcb file open in Layout when you create a new Project container, the working folder is automatically set to the location of the open file.

Procedure

1. In the Vault view, right-click a vault or a folder, and click **Create Empty Project.**
2. In the Create Empty Project dialog box, type a name and description for the new project, and click **OK**.
3. Right-click on the new project and click **Set Working Folder**.
4. In the Browse for Folder dialog box, select the folder containing your PCB project.

   Tip: If you don't yet have the PCB project files, you can click **Make New Folder** to create a new folder to put your PCB project in later.
5. Click **OK**.

Result

The new Project container is created and associated with a working folder. Now you can add an archive to the vault**.**

Creating a Vault Folder

If you work with multiple PCB projects, you may want to organize project archive storage into separate folders. Use the following procedure to create a new folder in the vault.

Procedure

1. In the **Vault** tree, right-click the vault or a folder, and click **Create Folder**.
2. In the Create folder dialog box, type a name for the new folder, and click **OK**.

Result

The new folder is added to the vault, or to the folder you selected.

Changing Vaults

Use the following procedure to close the current vault and open another one.

Procedure

1. Vault view > **Select Vault** button
2. In the Select Vault dialog box, click the "…" Browse button to browse for the vault you want to open, or type in the pathname.

3. Click **OK**.

Result

The current vault is closed, and the selected vault is opened.

Deleting Items from the Vault

You can delete folders, projects and archives from the vault.

Procedure

1. In the Vault view, right-click the item and click **Delete**.

Reorganizing the Vault

After you have been using a vault for awhile, you may want to reorganize it. To do this, you can move and copy folders and project containers to new locations in the vault.

Procedure

1. In the Vault view, right-click the item and select **Copy to** or **Move to.**
2. In the **Copy to** or **Move to** window, select the new location for the item, and click **OK.**

Viewing and Editing Properties of a Vault Item

You can view and edit the properties of projects, folders and archives.

Procedure

1. In the Vault view, right-click the item and click **Properties**.
2. (Optional) Edit the properties as described in the Project and Folder Properties Dialog Box topic or the Archive Properties Dialog Box topic.

Related Topics

Finding Projects or Archives in the Vault

Setting Display Options in the Vault View

Use the following procedure to set filtering, display format and display order options that control how archives are displayed in the Vault view.

Procedure

1. In the Vault view, Click the **Options** button.
2. In the Options Dialog Box, set the Archive display format, Archive filters, and Archive order controls.

Working with Archives

In this section:

- Adding an Archive to the Vault
- Creating an Archive Template
- Restoring an Archive to the Working Folder
- Viewing the Contents of an Archive
- Finding Projects or Archives in the Vault

Adding an Archive to the Vault

When you add an archive to the vault, it is an archive of the current contents of the working folder.

Tip: In the vault view, the archive currently restored to the working folder (the base archive) is shown in green. The archive you are adding will be identified as a sub-archive of this archive. For instance, if the archive currently restored to the working folder is Archive 9, the ID of the new archive will be Archive 9.1.

Procedure

1. In the Working Folder view, click the **Add Archive to Vault** button.

   Restriction: The Add archive to Vault button is available only when a project or archive is selected in the Vault tree.

2. In the Add Archive toVault dialog box:

   a. In the Schematic project file box, browse for or type the name of the DxDesigner project (.prj) file.

   b. In the Layout project files box, click the **New (Insert)** button to browse for or type the names of the .pcb file(s).

      Tip: You can also use the keyboard Insert and Delete keys to add and delete files from the list.

c. In the Additional files and folders box, click the **New (Insert)** button to type or browse for the names of any additional files to include in the archive.

Tip: You can also use the keyboard Insert and Delete keys to add and delete files from the list.

d. In the Name box, type a name for the archive.

Tip: Create a name you can search for with the Find in Vault tool.

e. In the Description box, type a description of the archive.

Tip: Create a description you can search for with the Find in Vault tool.

Tip: As an alternative, you can set all these fields using a template. To do this, click Select from the Templates drop-down menu, select a template in the Templates Dialog Box Templates list, and click Select. (See Creating an Archive Template for information on creating templates.)

a. Click **OK**.

Related Topics

Restoring an Archive to the Working Folder

Creating an Archive Template

You can create customized archive templates to make it easier to add new archives.

Tip: Templates are created and saved for a specific project.

Procedure

1. In the Working Folder view, click the **Add Archive to Vault** button.
2. In thethe Add Archive to Vault dialog box, select **New** from the Templates drop-down list.
3. In the Templates Dialog Box, set the Schematic project file, Layout project files, Additional files and folders, Name, and Description fields by browsing for or typing the appropriate content.

   Tip: You can enter the variable “<projectname>” in any of these fields to have the template enter the project name in the matching field of the Add Archive to Vault dialog box. For example, if you enter “<projectname>.pcb” in the Layout project files field of the Templates dialog box, when you use this template to archive the Processor project, “Processor.pcb” will appear in the Add Archive to Vault dialog box; and when you use it to archive the Timer project, “Timer.pcb” will appear.
4. Click **Save**.

Result

The new template is created and stored in the vault project.

Restoring an Archive to the Working Folder

You can select any archive in the vault and restore it to the working folder.

Warning: Archive files will overwrite files of the same name in the working folder.

Procedure

1. In the Vault view, right-click the archive and click **Restore**.
2. When you are prompted to confirm the restore operation, click **Yes**.

Result

The archive is restored to the working folder, and the archive's icon turns green to indicate that it is the currently restored archive.

Viewing the Contents of an Archive

Use the following procedure to view a listing of an archive's content.

Procedure

1. In the Vault view, right-click on the Archive you want to view, and click **View Content**.

Result

The listing is displayed in your default text editor.

Finding Projects or Archives in the Vault

Use the following procedure to search the entire vault for projects or archives.

Procedure

1. In the Vault view, Click the **Find in Vault** button.
2. In the Find in Vault Dialog Box, from the **Items** list select **All**, **Projects**, **Archive**s or **Folders** to specify what you want to search for.
3. Specify the search criteria, as follows:
    a. In any or all of the Name, User, and Description fields, provide a character string to search for.

Tips:

- Searches are not word-based; a search for the string ic will find Iceland, picnicking, and alphanumeric.
- Wild cards are not supported.
- Searches are not case-sensitive.
- A null string in a field returns items with any value.
- Items must match all fields to satisfy the search criteria.
- Both the Name and ID attributes of items are searched for the search string specified in the Name field.

b. Set the **Created from** and **to** checkboxes and date/time fields as follows:

i. Select the **Created from** checkbox to return all items created after the date/time specified in the field.

ii. Select the **to** checkbox to return all items created up to the date/time specified in the field.

iii. Check both checkboxes to return all items created between the dates specified in the two fields.

4. Click **Find**.

Result

The items found are listed in the Find results list box. Double-click any item in the list to highlight it in the Vault tree.

Chapter 10
Working with DxDesigner

In this chapter:

Creating a New PCB Layout from a DxDesigner Design

There are two ways to create a new PCB design from a DxDesigner design.

Restriction

Importing a netlist without rules information does not turn on the plane thermals for CAM or Split/Mixed Planes.

Choosing a Method

Method 1—Use if you have both DxDesigner and PADS Layout on your computer. This automated method is the simplest. You need to open the PCB configuration file in a text editor to make changes.

Method 2—Use if you have both DxDesigner and PADS Layout on your computer. This automated method provides the most control of the process. You can use the the tabbed screens in DxDesigner's PCB Interface to make basic changes to the PCB configuration file and control the constraints.

Method 3—Use if you don't have DxDesigner on your computer and the netlist files are sent to you from the schematic engineer.

Method 1—Automated New PCB Design Using the DxDesigner Link

Use the DxDesigner Link to connect DxDesigner to PADS Layout and bring forward the whole design from the schematic to the layout.

Requirement

You must have both DxDesigner and PADS Layout on the same computer.

Procedure

1. In PADS Layout with the default startup design or a template design open, on the **Tools** menu, click **DxDesigner**.

   Result: In the DxDesigner Link, on the Documents tab, the pathname of your design appears in the PADS Layout Design box and the Connect button has changed to a Disconnect button since you are connected to the PADS Layout design.

2. In the DxDesigner Project File box, accept the file shown or browse for the corresponding schematic. Once your DxDesigner project pathname is in the box, click **Connect.**

3. If there are multiple designs available in the DxDesigner project, select the design you want from the **Design Name** list.

4. In the Forward/Backward configuration file area, click **Browse** to locate the pads<*latest_version*>.cfg file to use for forward or backward annotation between the two applications.

   Tip: Create a copy of the default configuration file and put it in your DxDesigner project directory. Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.

5. If you need to modify the config file settings, click **Edit** to modify the file in your default text editor.

   See also: PCB Interface Configuration Files chapter in the DxDesigner PCB Interfaces User Guide.

6. On the Library tab, select an import mode for Library Parts and choose a library.

   Tip: You can click New Library to create a new PADS library. By default, libraries are located at C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Libraries. The new library is added at the bottom of the library list and search order.

7. On the Documents tab, click **Forward to PCB**.

8. In the Forward Annotation dialog box, select **Create PCB**.

9. In the Data to pass to PCB area, select the schematic data to send to the PADS Layout design.

   Requirement: To include design rules in the forward annotation operation, select the Design Rules option, even if you already selected **Compare Design Rules** on the Preferences tab.

   Optional: Select the **Pause before Updating PCB Design** check box. Selecting this option gives you the chance to check any library import errors before importing the .asc file into PADS Layout. Then you can click Continue to proceed with the process.

10. Click **OK.**

11. In the Process Indicator dialog box, after the forward process if finished, click **Close**.

Method 2—Automated New PCB Design Using DxDesigner's PCB Interface Script

Use the DxDesigner PCB Interface to set basic options in the configuration file and use the command line script to automate the import of the DxDesigner design into PADS Layout.

Requirement

You must have both DxDesigner and PADS Layout on the same computer.

Procedure

1. **DxDesigner Tools** menu > **PCB Interface.**

2. In the PCB Interface dialog box, on the Basic tab, click **Create Netlist for Layout**.

3. Create a copy of the default configuration file and put it in your project directory.

   Tip: Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.

4. Make necessary settings on the Advanced and Constraints tabs of the dialog box according to the requirements of your design.

5. If other modifications to the configuration file are needed, open the file in a text editor and make them.

6. Select the **Run command line after processing** check box. For more infomation on the script, see Using the PCB Interface PADS layout Script.
7. Click **OK**.

   Result: The PCB Interface creates the netlisting files and the DxDesigner To PADS Layout dialog box appears.
8. In the DxDesigner To PADS Layout dialog box:
   a. Click **Create New PCB design**.
   b. If you want to use a template select the Start with Template PCB Design and in the in the Template PCB Design Filename (*.pcb) box, type the pathname or browse for your PCB design.
   c. Click **Next**.
9. In the Library Import Options dialog box if you want to transfer part types from DxDesigner to PADS Layout, select the **Update Part Types in Library** check box. If not, clear the check box and click Next.
   a. Browse for the Part Type ASCII File if it's not already listed. This .p file, along with the rest of the netlist files, is created in your DxDesigner Project folder by the PCB Interface prior to running this script.
   b. In the Import to Library box, select the PADS Layout library to which you want to add the DxDesigner parts.

      Tip: You can click New Library to create a new PADS library. By default, libraries are located at C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Libraries. The new library is added at the bottom of the library list and search order.
   c. Set the Import Mode for Library Parts.
10. Click **Finish**.

Method 3—Manual Create a New PCB Design

Use the DxDesigner PCB Interface to set basic options in the configuration file and output the netlist files. Import the part file (.p) into the library, and import the .asc file into the design.

Procedure

1. **DxDesigner Tools** menu > **PCB Interface.**
2. In the PCB Interface dialog box, on the Basic tab, click **Create Netlist for Layout**.
3. Create a copy of the default configuration file and put it in your project directory.

   Tip: Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.

4. Make necessary settings on the Advanced and Constraints tabs of the dialog box.
5. If other modifications to the configuration file are needed, open the file in a text editor and make them.
6. Clear the **Run command line after processing** check box.
7. Click **OK**.

   Result: The PCB Interface creates the following netlisting files: *.asc, *.fdc, *.ndc, *.p, *.db.
8. **PADS Layout File** menu > **Library**.
9. In the Library Manager dialog box, select the library into which you want to import the DxDesigner parts.
10. Click the **Parts** button.
11. Click **Import**.
12. In the Library Import File dialog box, browse for the .p file and then click **Open**.
13. Close the Library Manager.
14. **File** menu > **Import**.
15. In the File Import dialog box, browse for the .asc file and then click **Open**.

Result

All component decals (footprints) appear in PADS Layout stacked on the origin.

If there are errors during the .asc file import, an ascii.err file will open in your default text editor. All errors must be fixed, and you must repeat the process until there are no errors. No errors file is generated if no errors are found. The following are reported in the errors report file:

- Library issues
- Single or zero pin nets
- Totally floating connections or subnets
- Unnamed dangling connections (one end floating)
- Power and Ground symbols used on nets whose name is different from the default name on the symbol

- Multiple subnet nets where one or more subnets is missing an off-page symbol
- Single subnet nets with an off-page symbol (lonely subnet warning)
- User named subnets that have no visible net name label

Related Topics

Passing Attributes in the *Concepts Guide*.

Cross-Probing with DxDesigner

Use the DxDesigner Link to connect DxDesigner to PADS Layout and cross-probe.

Requirements

- You must have both DxDesigner and PADS Layout on the same computer.
- Ensure that cross-probing is turned on in DxDesigner; click Setup > Cross Probing. If there is a check mark to the left of Cross Probing, this option is already turned on.

Procedure

1. In PADS Layout with your layout design open, on the **Tools** menu, click **DxDesigner**.

   Result: In the DxDesigner Link, on the Documents tab, the pathname of your design appears in the PADS Layout Design box and the Connect button has changed to a Disconnect button since you are connected to the PADS Layout design.

2. In the DxDesigner Project File box, accept the file shown or browse for the corresponding schematic. Once your DxDesigner project pathname is in the box, click **Connect.**

3. If there are multiple designs available in the DxDesigner project, select the design you want from the **Design Name** list.

4. If necessary, resize the windows so you can see the DxDesigner Link dialog box and both applications.

5. On the Selection tab in the Selection Passing area, select both the **DxDesigner to PADS Layout** and **PADS Layout to DxDesigner** check boxes to enable bi-directional selection.

6. Besides selecting components in PADS Layout, you can also zoom to selections or keep your own zoom level and pan to selections. In the PADS Layout list, choose from **None**, **Zoom to Selected**, or **Pan to Selected**.

7. If you are using cross-probing to place parts on the PCB, you can identify which parts are placed and which parts are unplaced, using the Placement tab.

Tip: Parts are considered placed when they are located within the board outline.

Result

Selections can now be passed back and forth. Cross-probing can also be used for placing components on the PCB based on schematic sheet selections.

Related Topics

Managing the Selection List in the *Concepts Guide*

Comparing the PCB Layout Against the DxDesigner Design

There are three ways to compare a DxDesigner schematic design against a PADS Layout design to obtain a differences report.

Choosing a Method

Method 1—Use if you have both DxDesigner and PADS Layout on your computer. This automated method is the simplest and is controlled from PADS Layout.

Method 2—Use if you have both DxDesigner and PADS Layout on your computer. This automated method requires more steps and is controlled from DxDesigner.

Method 3—Use if you don't have DxDesigner on your computer and the netlist files are sent to you from the schematic engineer.

Method 1—Using the DxDesigner Link

Use the DxDesigner Link to connect DxDesigner to PADS Layout and generate a differences report.

Requirement

You must have both DxDesigner and PADS Layout on the same computer.

Procedure

1. In PADS Layout with your layout design open, on the **Tools** menu, click **DxDesigner**.

   Result: In the DxDesigner Link, on the Documents tab, the pathname of your design appears in the PADS Layout Design box and the Connect button has changed to a Disconnect button since you are connected to the PADS Layout design.

2. In the DxDesigner Project File box, accept the file shown or browse for the corresponding schematic. Once your DxDesigner project pathname is in the box, click **Connect.**

3. If there are multiple designs available in the DxDesigner project, select the design you want from the **Design Name** list.

4. If necessary, resize the windows so you can see the DxDesigner Link dialog box and both applications.

5. On the Preferences tab, select items to be compared.

6. On the Documents tab, click **Compare Designs**.

Result

DxDesigner Link generates and displays three files:

- A differences file (.dif) that reports the differences between the schematic and layout design files.
- An error report (.err).
- An Engineering Change Order file (.eco). The forward and backward annotation operations use this file to synchronize the layout design and the schematic files.

Method 2—Using DxDesigner's PCB Interface Script

Use the script of the DxDesigner PCB Interface to generate a differences report.

Requirement

You must have both DxDesigner and PADS Layout on the same computer.

Procedure

1. **DxDesigner Tools** menu > **PCB Interface.**

2. In the PCB Interface dialog box, on the Basic tab, click **Create Netlist for Layout**.

3. Create a copy of the default configuration file and put it in your project directory.

   Tip: Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.

4. Make necessary settings on the Advanced and Constraints tabs of the dialog box according to the requirements of your design.
5. If other modifications to the configuration file are needed, open the file in a text editor and make them.
6. On the Basic tab, select the **Run command line after processing** check box. For more infomation on the script, see Using the PCB Interface PADS layout Script.
7. Click **OK**.

   Result: The PCB Interface creates the netlisting files and the DxDesigner To PADS Layout dialog box appears.
8. In the DxDesigner To PADS Layout dialog box:
   a. Click **Compare Schematic with existing PCB design**.
   b. If your design pathname doesn't appear in the PCB Design Filename (*.pcb) box, type the pathname or browse for your PCB design.
   c. Select the **Generate Differences Report** check box and clear the Update check boxes.
   d. Click **Next**.
9. In the Library Import Options dialog box clear the **Update Part Types in Library** check box.
10. Click **Next**.
11. In the ECO Compare Options dialog box, set the comparison options you need.
12. Click **Finish**.

Result

The following files are created and located in the DxDesigner project folder:

- A differences file (powerpcb.rep) that reports the differences between the schematic and layout design files.
- An error report (ecogen.err).
- An Engineering Change Order file (*.eco). The forward and backward annotation operations use this file to synchronize the layout design and the schematic files.

Method 3—Manual Comparison

Use the DxDesigner PCB Interface to set basic options in the configuration file and output the netlist files. Use the Compare/ECO Tools to compare the designs and create a differences report.

Procedure

1. **DxDesigner Tools** menu > **PCB Interface.**
2. In the PCB Interface dialog box, on the Basic tab, click **Create Netlist for Layout**.
3. Create a copy of the default configuration file and put it in your project directory.

   Tip: Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.
4. Make necessary settings on the Advanced and Constraints tabs of the dialog box.
5. If other modifications to the configuration file are needed, open a text editor and make them.
6. Clear the **Run command line after processing** check box.
7. Click **OK**.

   Result: The PCB Interface creates the following netlisting files: *.asc, *.fdc, *.ndc, *.p, *.db.
8. In PADS Layout, on the Tools menu, click **Compare/ECO**.
9. In the Compare/ECO Tools dialog box, on the Documents tab, if you know which design is older and which is newer, choose the appropriate design for both the *Original Design to Compare and Update*, and *New Design with Changes* areas. If you don't know which design is older and which is newer then do the following:
   - In the Original Design to Compare and Update area, browse for the .asc netlist file in your DxDesigner project directory.

     Tip: Don't forget to change the Files of type from PADS Layout Files (*.pcb) to ASCII Files (*.asc).
10. In the Output Options area, select the **Generate Differences Report** check box. Clear the other Output Options check boxes.
11. On the Comparison tab, set the comparison options.
12. On the Update tab, clear all check boxes.
13. Click **Run**.
14. In the Process Status dialog box, click **Show Report**.

Result

The following files are created and located in the PADS Layout working folder:

- A differences file (layout.rep) that reports the differences between the schematic and layout design files.
- A log file (layout.log).

Forward Annotating from DxDesigner to PADS Layout

When laying out a design, there may be changes in the schematic that you want to import into the design. This process is called forward annotation. There are three ways to forward annotate design changes from DxDesigner to PADS Layout.

Tip: To avoid unexpected changes during forward annotation, consider comparing data before you forward-annotate. For more information, see Comparing the PCB Layout Against the DxDesigner Design.

Choosing a Method

Method 1—Use if you have both DxDesigner and PADS Layout on your computer. This automated method is the simplest. This method requires you to open the PCB configuration file in a text editor to make changes to it.

Method 2—Use if you have both DxDesigner and PADS Layout on your computer. This automated method provides the most control of the process. You can use the the tabbed screens in DxDesigner's PCB Interface to make basic changes to the PCB configuration file and control the constraints.

Method 3—Use if you don't have DxDesigner and PADS Layout on your computer and the netlist files are sent to from the schematic engineer to the layout designer.

Method 1—Automated Foward Annotation Using the DxDesigner Link

Use the DxDesigner Link to connect DxDesigner to PADS Layout and forward-annotate design changes from the schematic to the layout. Forward annotation compares data from a newer DxDesigner schematic to an older PADS Layout design file and updates the PADS Layout design to match the schematic.

Requirement

- You must have both DxDesigner and PADS Layout on the same computer.

Procedure

1. **With your design open in PADS Layout, on the Tools** menu, click **DxDesigner**.

   Result: In the DxDesigner Link, on the Documents tab, the pathname of your design appears in the PADS Layout Design box and the Connect button has changed to a Disconnect button since you are connected to the PADS Layout design.

2. In the DxDesigner Project File area, accept the file shown or browse for the corresponding DxDesigner project file (*.prj). Once your DxDesigner project pathname is in the box, click **Connect.**

3. If there are multiple designs available in the DxDesigner project, select the design you want from the **Design Name** list.

4. In the Forward/Backward configuration file area, click **Browse** to locate the pads<*latest_version*>.cfg file to use for forward or backward annotation between the two applications.

   Tip: Create a copy of the default configuration file and put it in your DxDesigner project directory. Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.

5. If you need to modify the config file settings, click **Edit** to modify the file in your default text editor.

   See also: PCB Interface Configuration Files chapter in the DxDesigner PCB Interfaces User Guide.

6. On the Library tab, select an import mode for Library Parts and choose a library.

   Tip: You can click New Library to create a new PADS library. By default, libraries are located at C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Libraries. The new library is added at the bottom of the library list and search order.

7. On the Preferences tab, select items to be compared during forward annotation.

   Tip: DxDesigner Link stores your preferences for future operations. You need to perform this step only if you want to change the preferences you selected previously.

8. On the Documents tab, click **Forward to PCB**.

9. In the Forward Annotation dialog box, select **Update PCB.**

10. In the Data to pass to PCB area, select the schematic data to send to the PADS Layout design.

Requirement: To include design rules in the forward annotation operation, select the Design Rules option, even if you already selected **Compare Design Rules** on the Preferences tab.

Optional. Select the **Pause before Updating PCB Design** check box. Selecting this option gives you the chance to review the ECO file containing changes that the update will make to the PADS Layout design. Then you can click Continue to update the layout or cancel it.

11. Click **OK.**

12. In the Process Indicator dialog box, after the forward process if finished, click **Close**.

Result

DxDesigner Link compares the data in the schematic to the existing layout design and updates the layout design with changes.

Related Topics

Cross-Probing with DxDesigner

Method 2—Automated Foward Annotation Using DxDesigner's PCB Interface Script

Use the DxDesigner PCB Interface to set basic options in the configuration file and use the command line script to automate the forward annotation between DxDesigner and PADS Layout. Forward annotation compares data from a newer DxDesigner schematic to an older PADS Layout design file and updates the PADS Layout design to match the schematic.

Requirement

You must have both DxDesigner and PADS Layout on the same computer.

Procedure

1. **DxDesigner Tools** menu > **PCB Interface.**

2. In the PCB Interface dialog box, on the Basic tab, click **Create Netlist for Layout**.

3. Create a copy of the default configuration file and put it in your project directory.

   Tip: Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.

4. Make necessary settings on the Advanced and Constraints tabs of the dialog box according to the requirements of your design.

5. If other modifications to the configuration file are needed, open a text editor and make them.

6. Select the **Run command line after processing** check box. For more infomation on the script, see Using the PCB Interface PADS layout Script.

7. Click **OK**.

   Result: The PCB Interface creates the netlisting files and the DxDesigner To PADS Layout dialog box appears.

8. In the DxDesigner To PADS Layout dialog box:

   a. Click **Compare Schematic with existing PCB design**.

   b. If your design pathname doesn't appear in the PCB Design Filename (*.pcb) box, type the pathname or browse for your PCB design.

   c. Set the remaining options to the requirements of your design.

   d. Click **Next**.

9. In the Library Import Options dialog box if you want to transfer part types from DxDesigner to PADS Layout, select the **Update Part Types in Library** check box. If not, clear the check box and click Next.

   a. Browse for the Part Type ASCII File if it's not already listed. This .p file, along with the rest of the netlist files, is created in your DxDesigner Project folder by the PCB Interface prior to running this script.

   b. In the Import to Library box, select the PADS Layout library to which you want to add the DxDesigner parts.

      Tip: You can click New Library to create a new PADS library. By default, libraries are located at C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Libraries. The new library is added at the bottom of the library list and search order.

   c. Set the Import Mode for Library Parts.

   d. Click **Next**.

10. In the ECO Compare Options dialog box, set the options to the requirements of your design.

11. Click **Finish**.

Result

The .eco file is generated and imported by PADS Layout. If there are errors during the import, an ascii.err file will open in your default text editor. All errors must be fixed, and you must repeat the process until there are no errors. No errors file is generated if no errors are found. The following are reported in the errors report file:

- Library issues
- Single or zero pin nets
- Totally floating connections or subnets
- Unnamed dangling connections (one end floating)
- Power and Ground symbols used on nets whose name is different from the default name on the symbol
- Multiple subnet nets where one or more subnets is missing an off-page symbol
- Single subnet nets with an off-page symbol (lonely subnet warning)
- User named subnets that have no visible net name label

Method 3—Manual Foward Annotation

Use the DxDesigner PCB Interface to set basic options in the configuration file and output the netlist files. Use the Compare/ECO Tools to compare the differences to the PCB design and import the changes into PADS Layout. Forward annotation compares data from a newer DxDesigner schematic to an older PADS Layout design file and updates the PADS Layout design to match the schematic.

See also: Comparing the PCB Layout Against the DxDesigner Design

Procedure

1. **DxDesigner Tools** menu > **PCB Interface.**
2. In the PCB Interface dialog box, on the Basic tab, click **Create Netlist for Layout**.
3. Create a copy of the default configuration file and put it in your project directory.

   Tip: Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.
4. Make necessary settings on the Advanced and Constraints tabs of the dialog box.
5. If other modifications to the configuration file are needed, open a text editor and make them.
6. Clear the **Run command line after processing** check box.

7. Click **OK**.

   Result: The PCB Interface creates the following netlisting files: *.asc, *.fdc, *.ndc, *.p, *.db.

8. In PADS Layout, on the Tools menu, click **Compare/ECO**.

9. In the Compare/ECO Tools dialog box, on the Documents tab, in the New Design with Changes area, clear the **Use Current PCB Design** check box. Browse for the .asc netlist file in your DxDesigner project directory.

   Tip: Don't forget to change the Files of type from PADS Layout Files (*.pcb) to ASCII Files (*.asc).

10. In the Original Design to Compare and Update area, select the **Use Current PCB Design** check box.

11. In the Output Options area, select the **Generate ECO File** check box. Clear the other Output Options check boxes.

12. On the Comparison tab, set the options according to the needs of your design.

13. On the Update tab, in the Update Options area, select the **Update Original Design** check box.

    Alternative: If you don't use this option, you need to import the .asc after you update the library with the .p file.

14. In the Library area, select the **Update Part Types in Library** check box. In the Part Type ASCII File box, browse for the .p netlist file in your DxDesigner project directory.

    Alternative: If you don't use this option, you need to import the .p file into the library using the Library Manager.

15. Choose the library to use and set the Import Mode.

    Tip: You can click New Library to create a new PADS library. By default, libraries are located at C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Libraries. The new library is added at the bottom of the library list and search order.

16. Click Run.

Result

The .p file is imported into the library and the .eco file is generated and imported by PADS Layout. If there are errors during the import, an ascii.err file will open in your default text editor. All errors must be fixed, and you must repeat the process until there are no errors. No errors file is generated if no errors are found. The following are reported in the errors report file:

- Library issues
- Single or zero pin nets

- Totally floating connections or subnets
- Unnamed dangling connections (one end floating)
- Power and Ground symbols used on nets whose name is different from the default name on the symbol
- Multiple subnet nets where one or more subnets is missing an off-page symbol
- Single subnet nets with an off-page symbol (lonely subnet warning)
- User named subnets that have no visible net name label

Back Annotating from PADS Layout to DxDesigner

When laying out a design, you may make engineering changes and want to update the schematic with those changes. This process is called backward annotation. There are three ways to back annotate design changes from PADS Layout to DxDesigner.

Tip: To avoid unexpected changes during back annotation, consider comparing data before you back-annotate. For more information, see Comparing the PCB Layout Against the DxDesigner Design.

Caution

Make sure you record ECO changes in an .eco file as you make them. For more details, see Recording ECO Changes. Recording ECO changes ensures the best results when compared to generating and ECO file by comparing the designs. For more details on the differences of these methods, see Recording Versus Generating an ECO File in the *Concepts Guide*.

Choosing a Method

Method 1—Use if you have both DxDesigner and PADS Layout on your computer. This automated method is the simplest. This method requires you to open the PCB configuration file in a text editor to make changes to it.

Method 2—Use if you have both DxDesigner and PADS Layout on your computer. This automated method provides the most control of the process. You can use the the tabbed screens in DxDesigner's PCB Interface to make basic changes to the PCB configuration file and control the constraints.

Method 3—Use if you don't have both DxDesigner and PADS Layout on your computer and the netlist files are sent from the layout designer to the schematic engineer.

Method 1—Automated Backward Annotation Using the DxDesigner Link

Use the DxDesigner Link to connect DxDesigner to PADS Layout and back-annotate design changes from the layout to the schematic. Back annotation compares data from an older DxDesigner schematic to a newer PADS Layout design file and updates the DxDesigner schematic to match the layout design.

Requirement

- You must have both DxDesigner and PADS Layout on the same computer.

Procedure

1. **With your design open in PADS Layout, on the Tools** menu, click **DxDesigner**.

   Result: In the DxDesigner Link, on the Documents tab, the pathname of your design appears in the PADS Layout Design box and the Connect button has changed to a Disconnect button since you are connected to the PADS Layout design.

2. In the DxDesigner Project File area, accept the file shown or browse for the corresponding DxDesigner project file (*.prj). Once your DxDesigner project pathname is in the box, click **Connect.**

   Result: The DxDesigner window becomes active and may have covered the DxDesigner Link dialog box. Return to the DxDesigner Link.

3. If there are multiple designs available in the DxDesigner project, select the design you want from the **Design Name** list.

4. In the Forward/Backward configuration file area, click **Browse** to locate the pads<*latest_version*>.cfg file to use for forward or backward annotation between the two applications.

   Tip: Create a copy of the default configuration file and put it in your DxDesigner project directory. Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.

5. If you need to modify the config file settings, click **Edit** to modify the file in your default text editor.

   See also: PCB Interface Configuration Files chapter in the DxDesigner PCB Interfaces User Guide.

6. On the Preferences tab, select items to be compared during forward annotation.

Tip: DxDesigner Link stores your preferences for future operations. You need to perform this step only if you want to change the preferences you selected previously.

7. On the Documents tab, click **Backward from PCB**.
8. In the Backward Annotation dialog box, choose one of the ECO preferences.
9. In the Data to pass to PCB area, select the layout data to send to the PADS Layout design.

   Requirement: To include design rules in the forward annotation operation, select the Design Rules option, even if you already selected **Compare Design Rules** on the Preferences tab.

   Optional. Select the **Pause before Updating PCB Design** check box. Selecting this option gives you the chance to review the ECO file containing changes that the update will make to the PADS Layout design. Then you can click Continue to update the layout or cancel it.
10. Click **OK.**
11. In the Process Indicator dialog box, after the forward process if finished, click **Close**.

Result

DxDesigner Link compares the new data in the layout to the older schematic design and updates the schematic design with changes. The pcb.err file opens to display any errors, warnings, or notes regarding the update.

Related Topics

Cross-Probing with DxDesigner

Method 2—Automated Backward Annotation Using DxDesigner's PCB Interface Script

Use the DxDesigner PCB Interface to set basic options in the configuration file and use the command line script to automate the back annotation between PADS Layout and DxDesigner. Back annotation compares data from an older DxDesigner schematic to a newer PADS Layout design file and updates the DxDesigner schematic to match the layout design.

Requirement

You must have both DxDesigner and PADS Layout on the same computer.

Procedure

1. **DxDesigner Tools** menu > **PCB Interface.**

2. In the PCB Interface dialog box, on the Basic tab, click **Back Annotate Information From Layout to Schematic**.
3. Create a copy of the default configuration file and put it in your project directory.

   Tip: Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.
4. Make necessary settings on the Advanced and Constraints tabs of the dialog box according to the requirements of your design.
5. If other modifications to the configuration file are needed, open a text editor and make them.
6. Select the **Run command line before processing** check box. For more infomation on the script, see Using the PCB Interface PADS layout Script.
7. Click **OK**.
8. In the PADS Layout To DxDesigner dialog box.
9. Ensure your .pcb file is listed in the PCB Design Filename (*.pcb) box.
10. Choose an ECO File Option.
    - If you selected Create ECO File Using Netlist Comparison, you must click Next and in the ECO Compare Options dialog box, set the options to the requirements of your design.
11. Click **Finish**.

Result

The script generates the .asc and .eco files required for back annotation and then the PCB Interface back annotates the files into DxDesigner.

Method 3—Manual Backward Annotation

Use the Compare/ECO Tools in PADS Layout to compare the differences between the new PCB design and the older schematic design and generate a .asc and .eco file. Use the DxDesigner PCB Interface to set basic options in the configuration file and process the .asc and .eco file from PADS Layout to create the .baf file and update the schematic design to match the layout design.

See also: Comparing the PCB Layout Against the DxDesigner Design

Procedure

1. PADS Layout **Tools** menu > **Compare/ECO**.
2. In the Compare/ECO Tools dialog box, on the Documents tab, in the New Design with Changes area, select the **Use Current PCB Design** check box.
3. In the Original Design to Compare and Update area, clear the **Use Current PCB Design** check box. Browse for the original .asc netlist file output from the DxDesigner design.

   Tip: Don’t forget to change the Files of type from PADS Layout Files (*.pcb) to ASCII Files (*.asc).
4. In the Output Options area, select the **Generate ECO File** check box and type a pathname or browse for the location to save the file.
5. Select the **Generate ASCII File for Back Annotation to Schematic** check box and type a pathname or browse for the location to save the file.
6. On the Comparison tab, set the options according to the needs of your design.
7. On the Update tab, clear the **Update Part Types in Library** check box.
8. Click **Run**.
9. In the Process Status dialog box, click **Close**.
10. Move the .asc and .eco file to the DxDesigner Project directory and ensure the name of the files matches the name of the DxDesigner project.

    Tip: Rename the original .asc file from the schematic so that you don’t overwrite it with the one being used for back annotation.
11. **DxDesigner Tools** menu > **PCB Interface.**
12. In the PCB Interface dialog box, on the Basic tab, click **Back Annotate Information From Layout To Schematic**.
13. Clear the **Run command line after processing** check box.
14. Create a copy of the default configuration file and put it in your project directory.

    Tip: Default configuration files are located at C:\MentorGraphics\<latest_version>PADS\SDD_HOME\standard. There are often configuration files for several versions. Use the latest version. You copy the configuration file to your DxDesigner project directory to customize the configuration file for your local project and to prevent overwriting the master copy that came with the software.
15. On the Advanced tab, select the **Automatically Back Annotate Changes to Schematic** check box.

16. Make any other necessary settings on the Advanced and Constraints tabs of the dialog box.

17. If other modifications to the configuration file are needed, open the file in a text editor to make the changes.

18. If your changes in the PCB Interface dialog box have changed the configuration file, click Save Pcb Configuration.

19. Click **OK**.

Working with Variants

- Importing Variant Data to PADS Layout
- Importing Variant Data to PADS Layout Manually
- Exporting Variant Data to DxDesigner
- Comparing Variant Data Files

Importing Variant Data to PADS Layout

Using the DxDesigner Link, you can import Variant Manager data into the PADS Assembly Variants feature.

Requirements

- Both the DxDesigner design and PADS Layout design must be open.
- Before you can transfer variant data, the DxDesigner design and PADS Layout design must be synchronized. In other words, the reference designators must match in both designs. If not, you must perform regular forward or backward annotation to synchronize the design.

Procedure

1. On the Tools menu, click **DxDesigner**.

2. In the DxDesigner Link, on the Documents tab, connect to the DxDesigner Project File.

3. On the Variants tab, click the **Create DxD AV file** button. This creates the <project_name>_DxDVariants.asc and adds the path to the DxDesigner AV ASCII file field in preparation for the next step.

   Alternative: Using a more manual process, you can create the .asc file from DxDesigner and then type or browse for the path in the Variants tab DxDesigner AV ASCII file field. For more information, see the Variant Manager Users Manual for the

Export data to PADS button. This method could be used if your setup does not allow a direct connection between DxDesigner and PADS using the DxDesigner Link.

4. Optionally, click the **Show DxD Variants** button to preview the assembly variant data file.
5. Click **Import AV to PADS Layout** to import the assembly variant data into the Assembly Variants feature.

Checking the Result

You can check to ensure the variant data imported properly.

1. On the Tools menu, click Assembly Variants.
2. In the Name list, select a variant.
3. In the list of reference designators, check the status of components and compare the results to Variant Manager in DxDesigner.

Related Topics

DxDesigner Dialog Box, Variants Tab

DxDesigner's Variant Manager Users Manual

Importing Variant Data to PADS Layout Manually

Exporting Variant Data to DxDesigner

Comparing Variant Data Files

Importing Variant Data to PADS Layout Manually

You can import variant data manually by exporting and then importing the data file.

Requirements

- Before you can transfer variant data, the DxDesigner design and PADS Layout design must be synchronized. In other words, the reference designators must match in both designs. If not, you must perform regular forward or backward annotation to synchronize the design.

Procedure

1. Open the design in DxDesigner, and open Variant Manager.
2. On the Variant Manager toolbar, click the **Export data for PADS** button.
3. Open the design in PADS Layout, and on the File menu, click Import.

4. In the Import dialog box, browse for the location of the <project_name>_DxDVariants.asc file. It is located in the same folder as the .prj file of the DxDesigner project.

5. Import the file.

Checking the Result

You can check to ensure the variant data imported properly.

1. On the Tools menu, click Assembly Variants.

2. In the Name list, select a variant.

3. In the list of reference designators, check the status of components and compare the results to Variant Manager in DxDesigner.

Related Topics

DxDesigner Dialog Box, Variants Tab

DxDesigner's Variant Manager Users Manual

Importing Variant Data to PADS Layout

Exporting Variant Data to DxDesigner

Comparing Variant Data Files

Exporting Variant Data to DxDesigner

Using the DxDesigner Link, you can export Assembly Variants data into the DxDesigner Variant Manager.

Requirements

- Both the DxDesigner design and PADS Layout design must be open.
- Before you can transfer variant data, the DxDesigner design and PADS Layout design must be synchronized. In other words, the reference designators must match in both designs. If not, you must perform regular forward or backward annotation to synchronize the design.

Procedure

1. On the Tools menu, click **DxDesigner**.

2. In the DxDesigner Link, on the Documents tab, connect to the DxDesigner Project File.

3. On the Variants tab, click the **Create PADS AV file** button. This creates the <project_name>_PADSVariants.asc and adds the path to the PADS Layout AV ASCII file field in preparation for the next step.
4. Optionally, click the **Show PADS Variants** button to preview the assembly variant data file.
5. Click **Import AV to DxDesigner** to import the assembly variant data into DxDesigner's Variant Manager.

Checking the Result

You can check to ensure the variant data imported properly.

- In DxDesigner, in Variant Manager, check the status of components and compare the results to the Assembly Variants feature in PADS Layout.

Related Topics

DxDesigner Dialog Box, Variants Tab

DxDesigner's Variant Manager Users Manual

Importing Variant Data to PADS Layout

Importing Variant Data to PADS Layout Manually

Comparing Variant Data Files

Comparing Variant Data Files

In the DxDesigner Link, you can compare the DxDesigner assembly variant data file from Variant Manager with the PADS Layout assembly variant data file from Assembly Variants.

Requirement

In the DxDesigner Link, you must be connected to at least one or both of DxDesigner or PADS Layout to enable the Variants tab.

Procedure

1. On the Tools menu, click **DxDesigner**.
2. In the DxDesigner Link, click the **Variants** tab.
3. Add the path to the DxDesigner assembly variant data file into the DxDesigner AV ASCII file field.
4. Add the path to the PADS assembly variant data file into the PADS Layout AV ASCII file field.

5. Click the Compare AV ASCII button.

   Result: The VariantCheck.txt report file opens.

Related Topics

DxDesigner Dialog Box, Variants Tab

DxDesigner's Variant Manager Users Manual

Importing Variant Data to PADS Layout

Importing Variant Data to PADS Layout Manually

Exporting Variant Data to DxDesigner

Chapter 11
Working with PADS Logic

In this chapter:

Creating a New PCB Layout from a PADS Logic Design

There are two ways to create a new PCB design from a PADS Logic design.

Caution

Forward annotation of changes to an *existing* design requires a different process. See Forward-Annotating Design Changes from PADS Logic.

Restriction

Transferring non ECO registered parts and non electrical parts is constrained by settings in the Options. See the Options dialog box, Design page in PADS Logic for details.

Choosing a Method

Method 1—Use if you have both PADS Logic and PADS Layout on your computer. This automated method is the simplest. It uses the PADS Layout Link within PADS Logic. For more information, see Method 1—Automatic Netlist Process Using the PADS Layout Link in the *PADS Logic User's Guide*.

Method 2—Use if you don't have both PADS Logic and PADS Layout on your computer. This manual method requires you to manually export the netlist from PADS Logic and then import it into PADS Layout.

Method 2—Importing the PADS Logic Netlist

Requirement

This is a continuation of a process started in PADS Logic. You must have the netlist (*.asc) file. For more information, see Method 2—Manual Netlist Process Between PADS Logic and PADS Layout in the *PADS Logic User's Guide*.

Procedure

1. In PADS Layout, click **File menu** > **Import**.
2. Select **ASCII Files (*.asc)** in the Files of type box.
3. Navigate to the location of the netlist file created from the PADS Logic design, select it, and click **Open**.

Result

The import process sources all the part types and decals from the library and stacks the decals at the origin. You can then use the Tools menu > Disperse Components command to spread out the components.

If an errors report file (ascii.err) is generated and errors are found in the netlist, see Toubleshooting the Netlist Process for more information.

Toubleshooting the Netlist Process

If errors are found in the netlist import process, an errors report file (ascii.err) is generated, and the error report file is displayed in a Notepad window. No errors file is generated if no errors are found. The following are reported in the errors report file:

- Library issues
- Single or zero pin nets
- Totally floating connections or subnets
- Unnamed dangling connections (one end floating)
- Power and Ground symbols used on nets whose name is different from the default name on the symbol
- Multiple subnet nets where one or more subnets is missing an off-page symbol
- Single subnet nets with an off-page symbol (lonely subnet warning)
- User named subnets that have no visible net name label

If an ascii.err file is generated, perform the following steps:

1. **PADS Layout File menu > New**
2. Click No when you are prompted to save the design.
3. Add any missing components listed in the ascii.err file to your library, either by adding a library which contains the missing components to your library list, or by creating the missing part types and decals. (See Adding Libraries to the Library List, Creating and Modifying Part Types, and Creating and Editing PCB Decals for information on how to do this.)
4. Resolve any other errors found in the ascii.err file.
5. When all the errors have been resolved, repeat the procedure you used to pass the netlist from PADS Logic to PADS Layout.

Related Topics

Creating a New PCB Layout from a PADS Logic Design

Cross Probing with PADS Logic

Cross probing with PADS Logic is controlled using the PADS Layout Link within PADS Logic. For instructions, see Cross-probing Between PADS Products in the *PADS Logic User's Guide*.

Forward-Annotating Design Changes from PADS Logic

You can import design changes from PADS Logic into PADS Layout. The ECO file, containing the design changes, is created by comparing the schematic and the layout design. There are three ways to bring forward design changes from the PADS Logic design.

Tip: If you're creating a new PCB by importing a netlist for the first time, see Creating a New PCB Layout from a PADS Logic Design.

Caution

Design changes resulting from forward-annotating can cause existing associated nets to be truncated, split, or deleted altogether. It can also cause existing associated nets to be extended or new associated nets to be created if the refdes prefixes of updated components are specified in the Associated Nets dialog box. For more information, see the Associated Nets chapter in this manual.

Restrictions

- During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.

- Transferring non-ECO-registered parts and non-electrical parts is constrained. See the override settings in the Design page of the PADS Logic Options.

Choosing a Method

- Use Method 1 if you have both PADS Logic and PADS Layout on your computer. This automated method is simpler and faster.
- Use Method 2 if you don't have both PADS Logic and PADS Layout on your computer, and you want to compare the designs in PADS Layout to generate the .eco file. This method is somewhat quicker than Method 3 since the Compare/ECO tool in PADS Layout can automatically import the file after the designs are compared.
- Use Method 3 if you don't have both PADS Logic and PADS Layout on your computer, and you want to compare the designs in PADS Logic to generate the .eco file.

Method 1—Automated Forward Annotation Process

See Method 1—Automated Forward Annotation Process in the PADS Logic User's Guide. This process is controlled entirely by the PADS Layout Link within PADS Logic.

Method 2—Generate the ECO File in PADS Layout

This is a continuation of the procedure Method 2—Generate the ECO File in PADS Layout in the *PADS Logic User's Guide*. You receive an updated netlist file from PADS Logic and compare it to the current PCB layout to generate an .eco file that you import to update the PCB layout.

Requirement

You must acquire an updated schematic netlist (.asc) file, and have the PCB design open in PADS Layout.

Procedure

Tip: There is no undo after an .eco file import. If you think you might want to revert to the state of the design before the import, you should save a copy of the PCB layout.

1. **PADS Layout Tools menu** > **Compare/ECO**
2. In the Compare/ECO dialog box, click the **Documents** tab.
3. In the Original Design to Compare and Update area, select the **Use Current PCB Design** check box.

   Tip: If the check box is unavailable, clear the check box in the New Design with Changes area.

4. In the New Design with Changes area, browse for the updated schematic netlist (asc.) file.

5. Click the Comparison Tab, and select the options you want to use for design comparison.

6. **Optional:** If you want to check the design differences before creating the ECO file, select only the **Generate Differences Report** check box in the **Documents** tab, and click **Run**.
   The netlist and PCB files are compared and differences written to Layout.rep in the \PADS Projects folder. To see the differences, click **Show Report** in the Process Status dialog box.

7. Select the **Generate ECO File** check box, and verify the ECO Filename.
   Tip: Give the file a unique name to avoid overwriting any existing ECO files.

8. Click the Update Tab, and select the **Update Original Design** check box.

   Tip: This saves you from manually importing the generated ECO file.

9. Set the other options for updating.

10. Click **Run.** Output files are written to the \PADS Projects folder.

    Tip: In addition to the files listed above, messages or errors that occur during comparison are also written to Layout_Session.log and Layout.err in the \PADS Projects folder.

Results

The files generated are:

<*pcb_name*>mine.eco	The ECO file. Contains ECO commands that describe the changes needed to update the original design to match the new design. Generated when you select Generate ECO File from the Compare/ECO Tools Documents Tab. **See also:** ECO File Format in the Concepts Guide.
Layout.rep	The Differences Report file. Describes the differences between the "old" and the "new" compared files. Generated when you select Generate Differences Report from the Compare/ECO Tools Documents Tab. **See also:** Differences Report in the Concepts Guide.

ecogtmp0.asc *ecogtmp1*.asc	temporary copy of the “old” netlist temporary copy of the “new” netlist
ecogtmp[0|1].err	Generated only if errors are found in the netlist. A link to this file is displayed in the Output Window.

Method 3—Import the ECO File from PADS Logic

This is the PADS Layout portion of the procedure Method 3—Generate the ECO File in PADS Logic in the *PADS Logic User’s Guide*. You import the .eco file of design changes into PADS Layout.

Tip: As long as the PCB design hasn’t undergone engineering changes, the last .asc file from PADS Logic can be compared to the current design in PADS Logic to generate the .eco file of the engineering changes in the design. If the last exported .asc file is lost, you can export a .asc file from PADS Layout to compare against the current schematic to generate the .eco file that gets imported into PADS Layout to update the PCB layout. The same effect is created using Method 2 and the process is semi-automated by automatically importing the .eco file.

Requirement

You must acquire an .eco file, and have the PCB design open in PADS Layout.

Procedure

Tip: There is no undo after an .eco file import. If you think you might want to revert to the state of the design before the import, you should save a copy of the PCB layout.

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **ECO Files (*.eco)** file type.
3. Browse to the file to import and click **Open**.

Result

If there are errors during the import, an eco.err file will open in your default text editor. No chnages are made to the PCB layout until the .eco file is imported without errors. A link to the ECO Import errors file, is also written to the Output window.

Backward Annotating from PADS Layout to PADS Logic

You can export your PCB layout changes “back” to the schematic. This process is called *backward annotation*. There are three methods to backward annotate design changes.

You can backward annotate part, gate, pin, net, and attribute changes. For details, see Backward Annotation Results.

Restrictions

- Transferring non-ECO-registered parts and non-electrical parts is constrained. See Options dialog box, Design page for details.
- During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.
- You cannot perform backward annotation between PADS Logic and PADS Router because PADS Router does not export ECO files.

Choosing a Method

- If you have both PADS Logic and PADS Layout on your computer, you can use Method 1. While this automated method is the simplest and fastest, this method is not recommended since it generates its own .eco file by design comparison. You should use Method 2 to record the .eco file in PADS Layout and manually import it into PADS Layout for the best results. For more information, see Recording Versus Generating an ECO File in the *Concepts Guide*.
- If PADS Layout is on another computer and you want the layout designer to generate the ECO file, use Method 2. This is the most accurate method. You must manually import the .eco file into the PADS Logic design.
- If PADS Layout is on another computer and you must compare the designs and generate the ECO file within PADS Logic use Method 3. This method is not recommended since you are generating the .eco file by design comparison. You should use Method 2 to record the .eco file in PADS Layout and manually import it into PADS Layout for the best results. For more information, see Recording Versus Generating an ECO File in the *Concepts Guide*.

Method 1—Automated Backward Annotation Process

If PADS Logic and PADS Layout are on the same computer, you can use the PADS Layout Link dialog box to compare a newer PCB design with an older schematic and update the older schematic from the newer PCB design. You can also create a differences report.

Caution

This method generates a new .eco file and does not use a recorded .eco file. Recording the exact changes in an .eco file gives the best back annotation results. Use Method 2 for the best results. For more information, see Recording Versus Generating an ECO File in the *Concepts Guide*.

Requirement

You must have the older schematic open in PADS Logic, and the newer PCB design open in PADS Layout.

Procedure

1. **PADS Logic Tools** menu > **PADS Layout**

   Tip: If PADS Layout is not open, the Connect to PADS Layout dialog box appears. Click **Open** to open the PCB design you want to annotate *from* in PADS Layout. In the File Open dialog box, select the .pcb file and click **Open**.

2. In the PADS Layout Link dialog box, click the Design Tab.

3. **Optional:** If you want to check the design differences before updating, click the Compare PCB button
   The two versions are compared and differences written to Logic.rep in the \PADS Projects folder. To see the report, click the logic.rep link in the Output Window.

4. On the Preferences Tab, set the appropriate options.

5. On the ECO Names Tab, set the appropriate options.

6. On the Design Tab:

   a. If needed, check the **Compare Design Rules** and **Show Net List errors report** check boxes.

   b. Click the **ECO From PCB** button.

Related Topics

Cross-probing Between PADS Products

Method 2—Creating the ECO File in PADS Layout

Create the .eco file of design changes using PADS Layout and then import it into the PADS Logic design.

This procedure only documents recording the .eco file as you made engineering changes to your design since it creates a perfect before and after record of changes. Generating an .eco file by comparing designs will be electrically correct but it does not create a perfect before and after record of components that are exactly identical in their part types and connections. For more information, see Recording Versus Generating an ECO File in the *Concepts Guide*.

If you neglected to record the .eco changes and you must generate the .eco file by comparing two designs, see Creating an ECO File by Comparing Two Versions of a Design.

Procedure

1. Record the engineering/netlist changes in an .eco file. For more details, see Recording ECO Changes**.**
2. **PADS Logic Tools** menu > **Options** > **Design page**
3. Set the **Allow overwriting of attribute values in design with blank values from library** attributes check box appropriately to allow or prevent **overwriting of non-blank attribute values with blank ("placeholder") values from the library**.
4. With your design open in PADS Logic, on the **PADS Logic File** menu**,** click **Import.**
5. In the File Import dialog box, in the **Files of Type:** list, select **ECO Files (*.eco)**.
6. Browse for and select the ECO file to import.
7. Click **Open**. If no errors occur, the schematic is updated. If errors occur, the schematic is not updated, and the errors, along with a link to the ECO Import errors file, are written to the Output window.

Method 3—Creating the ECO File in PADS Logic

Acquire the layout design in a .asc file format and compare it to the schematic design with the PADS Logic Compare/ECO tools and then import .eco file of design changes into the PADS Logic schematic design.

Caution

This method generates a new .eco file and does not use a recorded .eco file. Recording the exact changes in an .eco file gives the best back annotation results. Use Method 2 for the best results. For more information, see Recording Versus Generating an ECO File in the *Concepts Guide*.

Requirement

You must have a .asc file exported from PADS Layout. For more details, see Exporting an ASCII File.

Procedure

1. **PADS Logic Tools Menu** > **Options** >**Design page**
2. Set the **Allow overwriting of attribute values in design with blank values from library** attributes check box appropriately to allow or prevent **overwriting of non-blank attribute values with blank ("placeholder") values from the library**.
3. **PADS Logic Tools menu** > **Compare/ECO**
4. In the Compare/ECO dialog box, click the Documents Tab.

5. In the Original Design to Compare and Update area, select the **Use Current Schematic Design** check box.

6. In the New Schematic Design with Changes area, browse for the new .asc file exported from PADS Layout.

7. Click the Comparison Tab, and select the options you want to use for design comparison.

8. **Optional:** If you want to check the design differences before creating the ECO file:

    a. Select the **Generate Differences Report** check box in the **Documents** tab.

    b. Clear the **Generate ECO File** check box.

    c. Click **Run**.
    The netlist and PCB files are compared and differences written to Logic.rep in the \PADS Projects folder. To see the differences, click **Show Report** in the Process Status dialog box.

9. Select the **Generate ECO File** check box, and verify the ECO Filename.
   Tip: Give the file a unique name to avoid overwriting any existing ECO files.

10. Click **Run.** Output files are written to the \PADS Projects folder.

    Tip: In addition to the files listed above, messages or errors that occur during comparison are also written to Logic_Session.log and Logic.err in the \PADS Projects folder.

11. **PADS Logic File menu** > **Import**

12. In the File Import dialog box, in the Files of Type: list, select **ECO Files (*.eco)**.

13. Browse for and select the ECO file to import.

14. Click **Open**. **If no errors occur, the schematic is updated**. If errors occur, the schematic is not updated, and the errors, along with a link to the ECO Import errors file, are written to the Output window.

Related Topics

Backward Annotation Results

Backward Annotation Results

This section describes what happens to layout changes that are back-annotated to the schematic.

- Attribute Level Backward Annotation
- Part Level Backward Annotation
- Gate Level Backward Annotation

- Net Level Backward Annotation
- Pin Level Backward Annotation

Attribute Level Backward Annotation

You can backward annotate new attributes and deleted attributes.

New Attributes

A new attribute in a part updates all parts of the same type. If the attribute name does not exist, it is added with the assigned value.

An error is created if the part does not exist.

Tip: Unsupported attribute types, such as net or net class, are ignored.

Deleted Attributes

Deleting an attribute for a part-type deletes the attribute on all parts of that type in the design.

An error message is generated if the part or attribute name does not exist.

If the attribute command specifies an object type not supported for general attributes, such as net or net class, the attribute command is ignored.

Related Topics

Method 2—Creating the ECO File in PADS Layout

Part Level Backward Annotation

You can backward annotate added parts, changed parts, deleted parts, and the reference designator name.

Added Parts

A new sheet is created and all new parts are added to the sheet. Parts are placed on a grid so that parts of a medium size do not overlap. No attempt is made to avoid overlapping of larger parts.

An error message is generated if the reference designator of the newly added part already exists or if the part does not exist in the Library.

Tip: The part is not added to the schematic if the reference designator already exists.

If the part contains Signal Pins, these pins are included in the add pin function. Backward annotation does not currently support signal pins so an error message is created.

Changed Parts

If the changed part is a multigate part, all gates are updated to the new part type.

An error message is generated if the new part does not exist in the design or in the Library, or if the gate or pin count is incompatible.

Deleted Parts

If the deleted part is a multigate part, all gates are deleted.

An error message is generated if the part is still connected to a net or the part does not exist.

Reference Designator Name

If the part being renamed is a multigate part, all gates are updated. An error message is generated if the old reference designator does not exist.

Related Topics

Method 2—Creating the ECO File in PADS Layout

Gate Level Backward Annotation

You can backward annotate swapped gates.

PADS Logic creates an offpage symbol at each swapped gate. An error message is created if the gate does not exist.

Related Topics

Method 2—Creating the ECO File in PADS Layout

Net Level Backward Annotation

You can backward annotate joined nets, nets created by splitting an existing net, and renamed nets.

Joined Nets

The first net is renamed to be the same name as the second net.

Nets Created by Splitting an Existing Net

A Delete Pin from Net operation is performed to remove them from the existing net, followed by an Add Pin to Net operation to add the pin to the new net.

Renamed Nets

All subnets of the old net on all sheets are renamed. If any of the subnets contain Power or Ground symbols without netnames, netnames are added to these symbols.

An error message is created if the new net already exists.

Related Topics

Method 2—Creating the ECO File in PADS Layout

Pin Level Backward Annotation

You can backward annotate swapped pins, pins added to a net, and pins disconnected from a net.

Swapped Pins

PADS Logic creates an offpage symbol at each swapped pin.

Pins Added to a Net

A pin can be added only if it is not already connected to another net. If the pin is a gate pin (a visible terminal pin on the gate symbol), an offpage symbol is created.

An error is created if pin is already connected or the pin is a signal pin already assigned to a net.

Pins Disconnected from a Net

If the pin is a gate pin, the connection is deleted if it connects to a tie-dot or offpage symbol. If the connection goes to another gate pin, the connection is broken and, an offpage symbol is added.

This command generates an error message if the pin is not connected to the net in question.

Related Topics

Method 2—Creating the ECO File in PADS Layout

Chapter 12
Setting up the Design Environment

Panning

The most efficient ways to pan are using the middle mouse button, or using the mouse wheel. But there are also many alternatives.

Panning Using the Middle Mouse Button

To center and pan the workspace:

1. Point in the workspace to where you want to locate the new center.
2. Middle-click. The screen repaints the design with the point you chose at the center of the screen.

Panning Using the Wheel

You can pan horizontally and vertically with the wheel in the workspace.

To pan vertically:

- Rotate the wheel.

  Tip: To scroll up, rotate the wheel away from you. To scroll down, rotate the wheel toward you.

To pan horizontally:

- Press **Shift** and rotate the wheel.

  Tip: To scroll to the left, rotate the wheel button up. To scroll to the right, rotate the wheel button down.

Alternative Panning Methods

There are many ways to pan around the design without using the mouse.

- Use the scroll bars to pan the view.
- To move the view, in the Status window click on the part of the postage stamp that represents the area you want to view. The center of view pans to that area.

- To pan the viewing window, moving one-half the screen width in the direction of the arrow use the arrow keys with NumLock On.
- To move the pointer on grid unit use the arrow keys with NumLock Off. Moving the pointer off the viewable area pans the view.
- To center the view at the current pointer location without zooming, press Ins on the numeric keypad.

Zooming to Specific Views

You can zoom to the board outline, the extents of all design items, or a selected item.

Zooming to the Board

This command zooms to and centers the board outline in the workspace.

- Press **Ctrl+B**.

  Alternatives:

  - On the standard toolbar, click the **Board** button.
  - On the View menu, click **Board**.

Zooming to the Extents

This command zooms to fit all objects, including those outside the board outline, into the workspace.

- Press **Ctrl+Alt+E**.

  Alternatives:

  - On the standard toolbar, click the Extents button.
  - On the View menu, click **Extents**.

Zooming to the Selection

Specific Object Zoom

Any object you select becomes the center view in the workspace.

1. Select an object.
2. Press **Alt+Z**.

Alternative: On the View menu, click **Selection**.

In the Project Explorer

You can also zoom to the selection when you select an object in the Project Explorer.

1. Right-click in the Project Explorer, and click **Allow Selection**.

   Tip: This option may already be selected, if you click it again, you will turn it off.

2. Right-click again, and click **Zoom to Selection**.

   Tips:

   - This option may already be selected, if you click it again, you will turn it off.
   - If Zoom to Selection is already enabled, when you right-click an object, that object becomes the center view in the workspace.

3. Select an object.

Zooming Using the Mouse

You can zoom using the middle mouse button or using a combination of toolbar button and mouse buttons, or keyboard key and the wheel.

Using the Middle Mouse Button

You can use the middle mouse button to zoom. **Zoom in** magnifies the design view. **Zoom out** shrinks the design view.

To magnify the design view, zoom in:

1. With the cursor in the workspace, hold down the middle mouse button.
2. Drag the cursor diagonally up across the design, indicating both the horizontal and vertical limits of the bounding box that defines your next view.
3. When the area of the design you want to magnify is within the bounding box, release the middle mouse button.

   Alternative: Instead of zooming from the center as in the above, you can also zoom from the corner by holding down the Shift key while you drag the cursor.

To shrink the design view, zoom out:

1. With the cursor in the workspace, hold down the middle mouse button.
2. Drag the cursor diagonally down across the design, indicating both the horizontal and vertical limits of the bounding box that defines your next view. The line that expands

from the box represents the new view size in proportion to the old. The zoom-out ratio also appears with the cursor.

3. When the area you want to shrink is within the bounding box, release the middle mouse button.

   Alternative: Instead of zooming from the center as in the above, you can also zoom from the corner by holding down the Shift key while you drag the cursor.

Zooming Using the Zoom Button

You can activate a zoom mode with the Zoom button and zoom incrementally with the left and right mouse buttons.

1. On the standard toolbar, click the **Zoom** button. This button is a toggle.
2. Position the cursor at the desired view center.
3. Click the left mouse button to zoom in, or click the right mouse button to zoom out.
4. Press **Esc** to exit zoom mode and return the mouse to its normal functionality.

   Alternative: You can also click the Zoom button again to exit zoom mode.

Zooming Using the Wheel

You can zoom using the wheel button in combination with the Ctrl key.

- Press and hold down **Ctrl** and rotate the wheel button:

  Tips: To zoom in, rotate the wheel button up. To zoom out, rotate the wheel button down.

Zooming Using Keyboard Keys

- To draw a zoom rectangle, press **5** on the numeric keypad with NumLock on. When you zoom out, a solid box shows at the pointer. This represents the current view size. The thin line that expands from the solid box represents the new view size in proportion to the old. The zoom-out ratio also displays with the pointer.
- To zoom in centered at the pointer location press **Pg Up** on the numeric keypad.
- To zoom out centered at the pointer location press **Pg Dn** on the numeric keypad.

Pan and Zoom Shortcuts

Table 12-1 provides a list of pan and zoom shortcut keys.

Table 12-1. Pan and Zoom Shortcut Keys

PADS Logic Command/Operation	PADS Layout Command / Operation	Accelerator Keys	Common Keyboard/ Mouse	PADS Router Command/ Operation	Comments
Auto Pan Display On/Off	Auto Pan Display On/Off	Ctrl & Alt	A	Auto Pan display On/Off	
View Sheet	View Board	Ctrl	B	View Board	
Zoom Selection	Zoom Selection	Alt	Z	Zoom Selection	
Status	Status	Ctrl & Alt	S		
Zoom Mode on/off	Zoom Mode on/off	Ctrl	W	Zoom Mode on/off	
View Sheet	View Board		Home	View Board	
Zoom in at cursor (Zoom mode only)	Zoom in at cursor (Zoom mode only)		Spacebar	Zoom in at cursor (Zoom mode only)	
Zoom in	Zoom in		PgUp	Zoom in at cursor	
Zoom out	Zoom out		PgDn	Zoom out at cursor	
Move Cursor Up 1 Grid Unit	Move Cursor Up 1 Grid Unit		Up Arrow	Move Cursor Up 1 Grid Unit	
Move Cursor Down 1 Grid Unit	Move Cursor Down 1 Grid Unit		Down Arrow	Move Cursor Down 1 Grid Unit	
Move Cursor Left 1 Grid Unit	Move Cursor Left 1 Grid Unit		Left Arrow	Move Cursor Left 1 Grid Unit	

Table 12-1. Pan and Zoom Shortcut Keys (cont.)

PADS Logic Command/Operation	PADS Layout Command / Operation	Accelerator Keys	Common Keyboard/ Mouse	PADS Router Command/ Operation	Comments
Move Cursor Right 1 Grid Unit	Move Cursor Right 1 Grid Unit		Right Arrow	Move Cursor Right 1 Grid Unit	
Center View at Cursor	Center View at Cursor		KeyPad0	Center View at Cursor	
Redraw	Redraw		KeyPad1	Redraw	
Pan Down	Pan Down		KeyPad2	Pan Down	NumLock On
Move Cursor Down 1 Grid Unit	Move Cursor Down 1 Grid Unit			Move Cursor Down 1 Grid Unit	NumLock Off
Zoom Out at Cursor	Zoom Out at Cursor		KeyPad3	Zoom Out at Cursor	
Pan Left	Pan Left		KeyPad4	Pan Left	NumLock On
Move Cursor Left 1 Grid Unit	Move Cursor Left 1 Grid Unit			Move Cursor Left 1 Grid Unit	NumLock off
Zoom Area In/Out	Zoom Area In/Out		KeyPad5	Pan to Center of Board	NumLock On
Unassigned in PADS Logic	Unassigned in PADS Layout			Unassigned in PADS Router	NumLock Off
Pan Right	Pan Right		KeyPad6	Pan Right	NumLock On
Move Cursor Right 1 Grid Unit	Move Cursor Right 1 Grid Unit			Move Cursor Right 1 Grid Unit	NumLock Off
Zoom Sheet	Zoom Board		KeyPad7	Zoom Board	

Table 12-1. Pan and Zoom Shortcut Keys (cont.)

PADS Logic Command/Operation	PADS Layout Command / Operation	Accelerator Keys	Common Keyboard/ Mouse	PADS Router Command/ Operation	Comments
Pan Up	Pan Up		KeyPad8	Pan Up	NumLock On
Move Cursor Up 1 Grid Unit	Move Cursor Up 1 Grid Unit			Move Cursor Up 1 Grid Unit	NumLock Off
Zoom in at Cursor	Zoom in at Cursor		KeyPad9	Zoom in at Cursor	
Zoom from Window corner	Zoom from Window corner		KeyPad Decimal	No equivalent in PADS Router	NumLock On
Delete Selected	Delete Selected			No equivalent in PADS Router	NumLock Off
Zoom in at Cursor (Zoom Mode Only)	Zoom in at Cursor (Zoom Mode Only)		LMButton click	Zoom in at Cursor (Zoom Mode Only)	
Center View (Do not move cursor)	Center View (Do not move cursor)		MMButton click	Center View (Do not move cursor)	
Dynamic Pan	Dynamic Pan	Alt	MMButton drag	Dynamic Pan (Was shift+MMB click)	New Assignment
Zoom from Window Corner (Alternate Zoom)	Zoom from Window Corner (Alternate Zoom)	Shift	MMButton drag	Zoom from Window Corner (Alternate Zoom)	New Assignment
Zoom Area In/Out	Zoom Area In/Out		MMButton drag	Zoom Area In/Out	
Zoom Out at Cursor (Zoom Mode)	Zoom Out at Cursor (Zoom Mode)		RMButton click	Zoom Out at Cursor (Zoom Mode)	

Table 12-1. Pan and Zoom Shortcut Keys (cont.)

PADS Logic Command/Operation	PADS Layout Command / Operation	Accelerator Keys	Common Keyboard/ Mouse	PADS Router Command/ Operation	Comments
Zoom in at Cursor	Zoom in at Cursor	Ctrl	Wheel forward	Zoom in at Cursor	
Zoom out at Cursor	Zoom out at Cursor	Ctrl	Wheel back	Zoom out at Cursor	
Scroll up one line	Scroll up one line		Wheel forward	Scroll up one line	
Scroll down one line	Scroll down one line		Wheel back	Scroll down one line	
Scroll left one line	Scroll left one line	Shift	Wheel forward	Scroll left one line	
Scroll right one line	Scroll right one line	Shift	Wheel back	Scroll right one line	
Scroll up 1 pixel	Scroll up 1 pixel	Ctrl &Alt	Wheel forward	Scroll up 1 pixel	
Scroll down 1 pixel	Scroll down 1 pixel	Ctrl &Alt	Wheel back	Scroll down 1 pixel	
Scroll left 1 pixel	Scroll left 1 pixel	Shift & Alt	Wheel forward	Scroll left 1 pixel	
Scroll right 1 pixel	Scroll right 1 pixel	Shift & Alt	Wheel back	Scroll right 1 pixel	

Controlling views

Defining a Specific View Area

- To define a specific view area, hold the mouse button down, move the mouse to define the view rectangle, and release the mouse button.

Redrawing a View

- To redraw the current view press **End** on the numeric keypad.

Saving a View

You can save a work area view:

1. Arrange the work area to the view you want to capture.
2. On the **View** menu click **Save View**. The Save View dialog box appears.
3. Click **Capture**.
4. Type a name in the **Capture a New View** box and Click **OK**.

You can create up to nine views. The view names appear at the bottom of the View menu.

Tip: Capture is not available in the PCB Decal Editor.

Restoring a View

To restore a previously captured view:

1. On the **View** menu, select the name of the captured view.
2. Click **Apply**.

Tip: When you change the view, PADS Layout automatically saves the previous view. Use Previous View on the View menu to restore this view.

Creating a Board Outline

Use the Board Outline and Cut Out button on the Drafting toolbar to create a board outline. You can create only one board outline per design.

Procedure

1. **Drafting Toolbar** button > **Board Outline and Cut Out** button.
2. Right-click and click the shape of your board outline.
3. Draw your outline.
   See also: Creating a Rectangle Drafting Object, Creating a Circle Drafting Object, Creating a Polygon or Path Drafting Object
4. **To change the color of the board outline, click Setup menu > Display Colors. Then set the color in the Other area.**
   Tip: Any cut outs you create will be the same color.

Result

The board outline is created.

Related Topic

Importing a Board Outline and Cut Out from AutoCAD

Reusing a Board Outline

Creating a Board Cut Out

Use the Board Outline and Cut Out button on the Drafting toolbar to also create a board cut out. You can create only one board outline per design, so the same command and creation process is used for creating cut outs.

Restriction

You cannot create a notch using a board cut out. Edit the board outline instead.

Requirement

The board outline must exist before you can create a cut out.

Procedure

1. **Drafting Toolbar** button > **Board Outline and Cut Out** button.
   Tip: If a board outline already exists, the message "Board Outline exists. Would you like to create a Board Cut Out?" appears. Click **Yes** to continue.
2. Right-click and click the shape of your board cut out.
3. Draw your cut out.
   See also: Creating a Rectangle Drafting Object, Creating a Circle Drafting Object, Creating a Polygon or Path Drafting Object
4. **To change the color of the board cut out, click Setup menu > Display Colors. Then set the color in the Other area using the Board Outline tile.**
   Warning: Changing the color of the cut out changes the color of board outline.

Result

The board cut out is created.

A board cut out cannot intersect another cut out or the board outline. If intersections occur, the message "Selected point is outside the board area" appears in the Status Bar. If you move a cut out and it intersects another cut out or the board outline, the message "Board cut out intersects the board outline" appears.

Importing a Board Outline and Cut Out from AutoCAD

You can import a board outline from AutoCAD, but it must be created according to the following procedure.

Procedure

1. Create a layer within AutoCAD entitled BOARD_OUTLINE_00.
2. Draw your board outline using the Polyline button in AutoCAD. Make sure you close the Polyline shape, and assign a real width to the closed polyline.

   Restriction: Only one polyline object can be used for the board outline since PADS Layout only allows a single board outline shape.

   If you did not create the board outline shape on the new layer, you can select the board outline, and within the Properties of AutoCAD, assign the shape to the BOARD_OUTLINE_00 layer.
3. If you do not require a cut out, simply save the file as a DXF format file and import into a blank PADS Layout design file. If you have a cut out(s), proceed with the next steps.
4. Create a layer within AutoCAD entitled BOARD_CUTOUT_00.
5. Draw your cut out(s) using a closed Polyline for each cutout.

   If you did not create the board cutout shape on the new layer, you can select the cut out(s), and within the Properties of AutoCAD, assign the cut out(s) to the BOARD_CUTOUT_00 layer.
6. Select the board outline and cut out(s), and make a Block out of them. In the Block Definition dialog box, name the block BOARD_1.
7. Save the file in AutoCAD as a DXF format file and import into a blank PADS Layout design file.

Result

- Did the board outline or cutout import as a 2D line? If so, it wasn't assigned to the correct layer. In AutoCAD, go into the object's properties and assign it to the correct layer.
  You can determine if the object imported as a board outline or a 2D line by its color. Check the color of the object against the color set for the Board Outline in the Display Colors Setup dialog box. (The color set for the Board Outline must be different than the color set for Lines.)
- If the import of the board and cutout does not proceed correctly, explode the block in AutoCAD, recreate it and save a new DXF file.

- Are you getting an error about a polyline that isn't closed? In AutoCAD, edit the polyline and close it.

Related Topic

Creating a Board Outline

Creating a Board Cut Out

Reusing a Board Outline

Reusing a Board Outline

To reuse a board outline, export the outline and then import it into a new design.

Procedure

1. Create a board outline.
   Tip: If you have a design with a board outline you like, you can use this design; remove any extra line items so they are not included in the export in Step 2.
2. Export to an ASCII file.
   Tip: In the ASCII Output dialog box, select the **Lines** check box only.
3. Import the saved ASCII file into a new design.

Result

The board outline is in the new design.

Moving a Board Cut Out

When you move a board cut out, the Move Preference is important; if the move preference is set to Move by Origin, the cut out may snap off screen. The pointer snaps to the origin of the board outline and not the cut out.

Procedure

1. **Tools** menu > **Options** > **Design page**.
2. In the Move Preference area, select either **Move by cursor location** or **Move by midpoint**.
3. Click **OK**.
4. Select the cut out.
5. Right-click and click **Move**.

Result

The cut out attaches to the pointer so you can place it where you want.

If you move a cut out and it intersects another cut out or the board outline, the message “Board cut out intersects the board outline” appears.

Related Topic

Creating a Board Cut Out

Setting the Design Origin

There are two ways to set your Design Workflow Origin: click-to-set or precise.

Setting the Origin by Click

Use the Set Origin command to use your mouse to click the location you want the Design Workspace origin.

Procedure

1. **Setup** menu > **Set Origin.**
2. Click where you want the new origin to be.

Setting a More Precise Origin Location

If you want a more precise location, you can set the origin to any of the following: a component origin, a pin, a drawing corner, a via, text, the center of a circle, or a line (or arc) Intersection.

Procedure

1. Select the object with the point you want the new origin to be.
2. **Setup** menu > **Set Origin.**
   Alternative: Use the SO Modeless command

Examples

The following are examples of line (or arc) intersections.

Chapter 13
Importing and Exporting

Importing Files

The Import command inserts design data from various formats into a .pcb file.

- **File** menu > **Import**.

PADS Layout supports the formats listed in Table 13-1.

Table 13-1. PADS Layout Import Data Formats

Format	Extension	Description
ASCII	.asc	PADS-ASCII format text **Restriction:** PADS Layout no longer imports Perform 6 ASCII format files.
Data eXchange Format, or DXF	.dxf	A graphical information format used by AutoCAD 2004 and other CAD systems.
Engineering Change Order	.eco	Contains forward annotation or backward annotation information generated in schematic capture package, concerned with logic changes to the design. **See also:** Working with PADS Logic, Working with DxDesigner
Intermediate Data Format, or IDF	.emn	Import data from IDF 2.0 and 3.0 format. You must have the IDF Interface option in PADS Layout to import IDF files. **See also:** Intermediate Data Format in the *Concepts Guide*
OLE	.ole	PADS Layout allows you to embed files from other applications as OLE objects in your design using Insert New Object. Once you have an OLE object in your design, you can export the object as a singular item to an .ole file using File Export. You can then import the OLE file into other PADS Layout designs. **See also:** Inserting OLE Objects in PADS Layout

Table 13-1. PADS Layout Import Data Formats (cont.)

Format	Extension	Description
Collaboration	.clb, .cle	Import markups in the collaboration file (.clb) and encrypted collaboration file (.cle) formats.
Protel 99SE design database	.ddb	Protel 99 design files in the binary format as well as in the ASCII format.
Protel PCB98 design	.pcb	Protel 98 design files in the binary format as well as in the ASCII format.
Protel DXP / Altium Designer design	.pcbdoc	Altium DXP/2004/2006/Altium Designer design files in the binary format as well as in the ASCII format.
P-CAD design	.pcb	P-CAD design files in binary and ASCII formats generated in P-CAD 2001, 2002, 2004, 2006.
CADSTAR PCB design	.pcb	CADSTAR design files in binary format generated in CADSTAR 5.0, 6.0, 7.0, or 8.0.
CADSTAR PCB archives	.cpa	CADSTAR files in ASCII format generated in CADSTAR 5.0, 6.0, 7.0, or 8.0.
OrCAD Board	.max	OrCAD Layout design files in binary generated in OrCAD Layout 9.X, 10.X.
Allegro Board	.brd	Cadence Allegro design files.

Exporting Files

The Export command selectively extracts data from the open .pcb file and saves it into other formats.

- **File** menu > **Export**.

PADS Layout supports the formats listed in Table 13-2.

Table 13-2. PADS Layout Export Data Formats

Format	Extension	Description
ASCII Export	.asc	PADS-ASCII format text **Restriction:** PADS Layout does not export Perform 6 ASCII format files.
Data eXchange Format, or DXF	.dxf	A graphical information format used by AutoCAD 2004 and other CAD systems.

Table 13-2. PADS Layout Export Data Formats (cont.)

Intermediate Data Format, or IDF	.emn and .emp	Export data to IDF 2.0 and 3.0 format. You must have the IDF Interface option in PADS Layout to export IDF files. **See also:** Intermediate Data Format in the *Concepts Guide*
OLE	.ole	You can embed files from other applications as OLE objects in your design using Insert New Object. Once you have an OLE object in your design, you can export the object as a singular item to an .ole file using Export. You can then import the .ole file into other PADS Layout designs.
HyperLynx HYP	.hyp	You can export the .hyp file if you only want the exported file. **Tip:** You can also create .hyp, or HyperLynx files during the process of translating the design to BoardSim using Tools menu > Analysis > Signal/Power Integrity.
IPC-D-356 or IPC-D-356a	.ipc	Export your data to IPC-D-356 or IPC-D-356 revision A formats.
ODB++	.tgz	Export your data to an ASCII open format. This format is self-contained and can be transferred between computers without any loss of data.
Collaboration Files	.cle	Export markups in the encrypted collaboration file format for use in collaboration software.
CCE Files	.cce	Export the design in compressed (.cce) format for use in design review and collaboration software. For example, CAMCAD, visECAD, or visEDOC.
CAM350 File	.cam	Launch the CAM350 link to export the file or to open the file in CAM350 if it’s installed.
SPECCTRA	.do	Launch the Specctra Translator to create the .do file for the Specctra autorouter.

Importing an ASCII File

You can import an ASCII file created with previous versions of PADS Layout. If format or other changes, such as the addition of an attribute dictionary, are needed to update your data to the current PADS release, they are made. If the ASCII file contains test points, PADS Layout removes the test points that are not accessible and lists them in the report file. Physical design reuses are preserved when you import an ASCII file.

Restriction

You cannot import the ASCII file if any of the following conditions are true:

- The design is in default layer mode and the ASCII file is in increased layer mode. Use the Layers Setup dialog box to change the design to increased layer mode.
- The ASCII file that has fewer electrical layers than the current design. Use the Layers Setup dialog box to increase the electrical layer count to match the number in the imported ASCII file.
- Perform 6 ASCII created the ASCII file.

Notes:

- Improvements in thermal relief definition in PADS 9.2 may cause changes in thermal pad appearance, flooding results and CAM output (for CAM Plane layers) when an ASCII file from a previous version is imported into PADS 9.2. If incompatibilities are found when such a file is imported, a warning prompt is displayed and the incompatibilities are written to the ascii.err file.
- Beginning with PADS 9.0, die parts and flip chips are no longer identified by their family designations (DIE or FLP), but instead by the Special Purpose settings in the General tab of the Part Information dialog box. When you import an ASCII file created by a previous PADS version, these Special Purpose settings are automatically set for parts having the logic family DIE or FLP. The part's family designation remains the same.

Procedure

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **ASCII Files (*.asc)** file type.
3. Browse to the file to import and click **Open**.

Result

Import process information is written to ascii.err, located in the \PADS Projects folder.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Exporting an ASCII File

Exporting an ASCII File

You can use ASCII files to exchange design data between PADS Layout and external translators or previous versions of PADS Layout.

Tips:

- Improvements in thermal relief definition in PADS 9.2 may cause changes in thermal pad appearance, flooding results and CAM output (for CAM Plane layers) when an design is exported from PADS 9.2 to a previous version. If incompatibilities are found when the design is exported, a warning prompt is displayed and the incompatibilities are written to the Powerpcb.Rep file.
- Beginning with PADS 9.0, die parts and flip chips are no longer identified by their family designations (DIE or FLP), but instead by the Special Purpose settings in the General tab of the Part Information dialog box. With this change, the following changes occur in a design when you export it to an ASCII file of a previous PADS version:
    - The Special Purpose settings of any die parts and flip chips are cleared.
    - Die parts and flip chips having a family designation other than DIE and FLP lose their die part or flip chip special purpose and become normal parts.
    - Any normal parts that have the DIE or FLP family designation are treated as die parts or flip chips in the previous PADS version.

Procedure

1. **File** menu > **Export**.
2. On the File Export dialog box, select the **ASCII Files (*.asc)** file type.
3. Specify the file to export and click **Save**.
4. On the ASCII Output dialog box, specify the items that you want to write to the ASCII file by doing one of the following:
    - Select the check boxes for the items. For information about each item, see Exported Item Descriptions.
    - Click **Select All**.
5. In the Format list, select the format version.

Warning: Exporting to older formats will result in the loss of newer software features not supported by older formats.

Tips:

- Use PowerPCB V3.0 to export to both PowerPCB and PowerBGA V3.0 ASCII formats.
- PADS Layout does not export to PADS-Perform 6 ASCII format.

6. In the Units list, select the units to export.

   Tip: If you plan to use the ASCII file for an external translator or another ASCII-reading program, select Current. If you plan to re-import the ASCII file and save it as a .pcb database, select Basic. Basic units represent how values are stored in the software. They do not use standard units of measure, but they record precise positioning values for database items.

7. If you want to export attributes assumed from higher levels in the attribute hierarchy, select the **Parts** and **Nets** check boxes as needed.
8. Click **OK**.

Result

The .asc file is created. An export status log file— ascii.err—and an export report file—Powerpcb.Rep—are created in the \PADS Projects folder. Physical design reuse information, if any, is retained.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Importing an ASCII File

Importing OLE Files

You can import an .ole file (containing OLE objects) into the design. This is useful when another design contains OLE objects that you want to include in the current design.

Procedure

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **OLE Files (*.ole)** file type.
3. Browse to the file to import and click **Open**.

Result: The OLE objects import into the current design.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Exporting OLE Files

Exporting OLE Files

You can export OLE objects to an .ole file. This is useful when the current design contains OLE objects that you want to include in another design.

Procedure

1. Select an OLE object > **File** menu > **Export**.
2. On the File Export dialog box, select the **OLE Files (*.ole)** file type.
3. Specify the file to export and click **Save**.

Result: The .ole file is created.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Importing OLE Files

Importing DXF Files

You can import DXF files of the AutoCAD 2004 DXF format.

Recommendation: There are two interfaces to import DXF files. This procedure uses an interface designed for advanced import of any and all design objects. It imports the design objects from specially named block and layers names in AutoCAD. This mapping is hard-coded and can’t be changed.
If you need to simply import RF shapes into the decal or into the design, see Importing RF Shapes in DXF Format.

Restriction: You cannot import a DXF file to a design if the design is in default layer mode and the DXF file is in increased layer mode. Use the Layers Setup dialog box to change the design to increased layer mode.

Procedure

1. On the **File** menu, click **Import**.
2. In the File Import dialog box, in the file type list, select **DXF Files (*.dxf)**.
3. Browse to the file to import and click **Open**.

4. In the DXF Import dialog box, in the Layer Selection area, select layers from which to import information.

   Tip: To add individual layers, select the layer names in the Available list and click the Add button. To remove individual layers, select the layer names in the Selected list and click the Remove button.

5. In the Select Input Items area, specify the items that you want to import by doing one of the following:

   - Select the check boxes for the items.
   - Click **All Items**.

6. In the DXF-File Unit list, select the units to use in the DXF file.

   Tip: This list is unavailable if it is not necessary to set the units.

7. In the Mode Area, click an import mode.
8. Click **OK**.

Result

The .dxf file is imported and opened in PADS Layout.†

If you need your imported geometries to be single objects, but they have been imported as multiple line items, see Joining and Closing 2D Lines and Copper Shapes.

Warning: Nets bridged with copper are not supported in the .dxf file format. If you are importing a file previously exported with bridged coppers, they will import as regular copper shapes.

Related Topics

Importing a Board Outline and Cut Out from AutoCAD

DXF Format in the *Concepts Guide*

Importing and Exporting Files in the *Concepts Guide*

Exporting DXF Files

You export DXF files to transfer design elements to AutoCAD 2004. For example, you can export the design with component height information in order to create a 3D model.

Procedure

1. On the **File** menu click **Export**.
2. In the File Export dialog box, in the Save as type list, select **DXF Files (*.dxf)**.

3. Browse to overwrite a file or type a new file name. Click **Save**.
4. On the DXF Export dialog box, in the Export Type area, choose between standard and flat output.
   Restriction: If you select flat output, the setup of DXF drill sizes and symbols is unavailable.
5. In the Layer Selection area, select layers from which to export information.

   Tip: To add individual layers, select the layer names in the Available list and click the Add button. To remove individual layers, select the layer names in the Selected list and click the Remove button.
6. In the Select Input Items area, specify the items that you want to export by doing one of the following:
   - Select the check boxes for the items.
   - Click **All Items**.
7. In the DXF-File Unit list, select the units to use in the DXF file.

   Tip: This list is unavailable if it is not necessary to set the units.
8. To define DXF drill sizes and symbols, click **Setup** and do the following on the Setup DXF Drill Size and Symbols dialog box (unavailable if exporting a flat type file):
   a. If you want to specify a drill size for a 2D line library item, click **Add**, type a value into the Drill Size box, and then type the library item name into the Library Equivalent box.

      Tip: If you do not associate a drill size to a 2D line library item, the default drill symbol is exported for that item.
   b. If you want to remove a drill size for a 2D line library item, select the item in the Drill Size and Library Equivalent area, and then click **Delete**.
   c. Click **OK**.
   d. If you want to change symbol units, select the units from the **System Units** list.
   e. If you want to change the appearance of the default drill symbol, which appears in the DXF file as a plus sign +, type the length/width of the plus sign into the Symbol Size box, and then type the plus sign line width into the Line Width box.
9. Click **OK**.

Result

The .dxf file is created.

Warning: Nets bridged with copper are not supported in the .dxf file format and will export as regular copper shapes.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

DXF Format in the *Concepts Guide*

Specifying DXF Drill Sizes and Symbols

Use the Setup DXF Drill Size and Symbols dialog box to specify drill sizes for 2D line library items that you export to DXF.

Procedure

1. **File** menu > **Export** > save as DXF type > Export dialog box > **Setup** button.
2. If you want to specify a drill size for a 2D line library item, click **Add**, type a value into the Drill Size box, and then type the library item name into the Library Equivalent box.

   Tip: If you do not associate a drill size to a 2D line library item, the default drill symbol is exported for that item.
3. If you want to remove a drill size for a 2D line library item, select the item in the Drill Size and Library Equivalent area, and then click **Delete**.
4. If you want to change symbol units, select the units from the **System Units** list.
5. If you want to change the appearance of the default drill symbol, which appears in the DXF file as a plus sign +, type the length/width of the plus sign into the Symbol Size box, and then type the line width of the plus sign into the Line Width box.
6. Click **OK**.

Related Topics

Setup DXF Drill Size and Symbols Dialog Box

DXF Export Dialog Box

Importing IDF Files

You can use IDF files to exchange design data between PADS Layout and a mechanical design system. You can import an IDF file to import board outlines, keepouts, components, and holes from a mechanical design system. PADS Layout cannot import the information in the .emp library file.

Some of the components you import are mechanical objects that are not ECO registered, such as board mounting holes and card guides.

See also: Importing Holes and Adding Components During Importing in the *Concepts Guide*

Requirement: Before importing an IDF file into PADS Layout, first export IDF files from the mechanical design system.

Procedure

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **IDF Files (*.emn)** file type. You can import IDF 2.0 or 3.0 .emn files.
3. Browse to the file to import and click **Open**.
4. On the IDF Import dialog box, in the Select Import Items area, specify the items that you want to import by doing one of the following:
   - Select the check boxes for the items.
   - Click **All Items**.

   See also: IDF Import Dialog Box
5. Click **OK**.

   Result: The .emn board file imported and opened in PADS Layout. A status log file is written to *C:\PADS Projects\idfimport.sts*.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Exporting IDF Files

Exporting IDF Files

You can use IDF files to exchange design data between PADS Layout and a mechanical design system. You can export IDF files to export board outlines, keepouts, components, and holes to a mechanical design system.

Tip: Set any IDF-specific part height information, drilled hole information, or part outline information for a more accurate IDF export.

Procedure

1. **File** menu > **Export**.
2. On the File Export dialog box, select the **IDF Files (*.emn, *.emp)** file type.

3. Specify the file to export and click **Save**.
4. On the IDF Export dialog box, in the **Shape Layer** list, select the layer containing the outline information for the decals in your design that you want to send to the mechanical design system.
5. In the **Format** list, select the IDF version to use.
6. In the Select Export Items area, specify the items that you want to export by doing one of the following:
    - Select the check boxes for the items.
    - Click **All Items**.
7. Specify the minimum height of components you want to export by typing the minimum part heights in the Top and Bottom boxes.

    Tip: Components less than these heights are not exported. If a part is not exported because of a minimum height value, a message is written to the status log file.
8. Click **OK**.
9. If the Geometry.Height attribute does not exist or is set to zero height for any exported part type and decal pairs, the Missing Height dialog box appears for each pair. If you want to specify height information prior to exporting the part type and decal pair, type the height into the **Height** box.

    Tip: If you do not specify the height, the mechanical design system may prompt you to enter a height when importing the IDF files.

Result: The .emn (board and placement) and .emp (part library) files are created. A status log file is written to C:\PADS Projects\idfexport.sts.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Importing IDF Files

To Set Part Outlines for IDF Export

The IDF library file requires part outline information for all parts.

Procedure

1. **Tools** menu > **PCB Decal Editor**.
2. Open the decal for which you want to define an outline.
3. Click the layer on which to define the outline. For IDF, you must use the same layer in each decal to define all part outlines.

4. Define the part outline.

   The outline must be a single, special kind of closed curve that is either a circle or a sequence of non-self-intersecting arcs and contiguous line segments.

5. Save the decal.

Related Topics

Part Outline Information in the *Concepts Guide*

To Add Drill Hole Information to IDF Files

Using the HOLE part attribute, you can enable the export of drill hole information for single-pin parts to IDF files. Vias and multiple-pin parts do not use the HOLE attribute, even though they are displayed as drilled holes in the mechanical design system.

Mechanical design systems interpret drill holes as board drills.

Procedure

1. **File** menu > **Library**.
2. On the Library Manager dialog box, select and edit the single-pin part for which you want to add drill hole information.

   Tip: If you use the same part type for holes and nonholes, but use different decals, set the HOLE attribute in the part type.

3. Add the HOLE part attribute using the Attributes tab on the Part Information dialog box.
4. Save your changes.

Related Topics

Exporting Holes in the *Concepts Guide*

Exporting Part Height Information to IDF Files

To export part heights to IDF:

- Assign the Geometry.Height attribute at the Decal or Part Type hierarchy level.

Related Topics

Working with Object Attributes

Using the Attribute Manager

Part Height Information in the *Concepts Guide*

Importing Protel 99SE Design Database Files

You can import a .ddb file directly into PADS Layout.

Procedure

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **Protel 99SE design database (*.ddb)** file type.
3. Browse to the file to import and click **Open**.

Result: The .ddb file is imported and opened in PADS Layout. The resulting PADS Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original file name is ProtelTest.ddb, the new filename will be ProtelTest_pads.pcb.

If a file with the new name already exists in the folder, a sequential number is added to the file name. For example, ProtelTest_pads_1.pcb, ProtelTest_pads_2.pcb, and so on.

A log file, with the same name as the resulting .pcb file, is saved in the \PADS Projects folder by default.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Importing Protel PCB98 Design Files

You can import a Protel .pcb file directly into PADS Layout.

Procedure

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **Protel PCB98 Design (*.pcb)** file type.
3. Browse to the file to import and click **Open**.

Result: The .pcb file is imported and opened in PADS Layout. The resulting PADS Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original file name is ProtelTest.pcb, the new filename will be ProtelTest_pads.pcb.

If a file with the new name already exists in the folder, a sequential number is added to the file name. For example, ProtelTest_pads_1.pcb, ProtelTest_pads_2.pcb, and so on.

A log file, with the same name as the resulting .pcb file, is saved in the \PADS Projects folder by default.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Importing Protel DXP / Altium Designer Design Files

You can import a .pcbdoc file directly into PADS Layout.

Procedure

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **Protel DXP / Altium Designer design (*.pcbdoc)** file type.
3. Browse to the file to import and click **Open**.

Result: The .pcbdoc file is imported and opened in PADS Layout. The resulting PADS Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original file name is ProtelTest.pcbdoc, the new filename will be ProtelTest_pads.pcb.

If a file with the new name already exists in the folder, a sequential number is added to the file name. For example, ProtelTest_pads_1.pcb, ProtelTest_pads_2.pcb, and so on.

A log file, with the same name as the resulting .pcb file, is saved in the \PADS Projects folder by default.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Importing P-CAD Design Files

You can import a .pcb file directly into PADS Layout.

Procedure

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **P-CAD design files (*.pcb)** file type.
3. Browse to the file to import and click **Open**.

Result: The .pcb file is imported and opened in PADS Layout. The resulting PADS Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original file name is PCADTest.pcb, the new filename will be PCADTest_pads.pcb.

If a file with the new name already exists in the folder, a sequential number is added to the file name. For example, PCADTest_pads_1.pcb, PCADTest_pads_2.pcb, and so on.

A log file, with the same name as the resulting .pcb file, is saved in the \PADS Projects folder by default.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Importing CADSTAR PCB Design Files

You can import a .pcb file directly into PADS Layout.

Procedure

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **CADSTAR PCB design files (*.pcb)** file type.
3. Browse to the file to import and click **Open**.

Result: The .pcb file is imported and opened in PADS Layout. The resulting PADS Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original file name is CADSTARTest.pcb, the new filename will be CADSTARTest_pads.pcb.

If a file with the new name already exists in the folder, a sequential number is added to the file name. For example, CADSTARTest_pads_1.pcb, CADSTARTest_pads_2.pcb, and so on.

A log file, with the same name as the resulting .pcb file, is saved in the \PADS Projects folder by default.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Importing CADSTAR Archives

You can import a .cpa file directly into PADS Layout.

Procedure

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **CADSTAR archive (*.cpa)** file type.
3. Browse to the file to import and click **Open**.

Result: The .cpa file is imported and opened in PADS Layout. The resulting PADS Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original file name is CADSTARTest.cpa, the new filename will be CADSTARTest_pads.pcb.

If a file with the new name already exists in the folder, a sequential number is added to the file name. For example, CADSTARTest_pads_1.pcb, CADSTARTest_pads_2.pcb, and so on.

A log file, with the same name as the resulting .pcb file, is saved in the \PADS Projects folder by default.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Importing OrCAD Board Files

You can import a .max file directly into PADS Layout.

Procedure

1. **File** menu > **Import**.
2. On the File Import dialog box, select the **OrCAD Board files (*.max)** file type.
3. Browse to the file to import and click **Open**.

Result: The .max file is imported and opened in PADS Layout. The resulting PADS Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original file name is OrCADTest.max, the new filename will be OrCADTest_pads.pcb.

If a file with the new name already exists in the folder, a sequential number is added to the file name. For example, OrCADTest_pads_1.pcb, OrCADTest_pads_2.pcb, and so on.

A log file, with the same name as the resulting .pcb file, is saved in the \PADS Projects folder by default.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Importing Allegro Board Files

You can import a prepared Allegro .brd file directly into PADS Layout. The file must be prepared by running a SKILL script.

Procedure

1. Prepare the Allegro file as described in Migrating Allegro Designs to PADS Layout in the *Allegro to PADS Translator User's Guide*.
2. **File** menu > **Import**.
3. On the File Import dialog box, select the **Allegro Board files (*.brd)** file type.
4. Browse to the prepared Allegro file to import and click **Open**.

Result: The .max file is imported and opened in PADS Layout. The resulting PADS Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original file name is OrCADTest.max, the new filename will be OrCADTest_pads.pcb.

If a file with the new name already exists in the folder, a sequential number is added to the file name. For example, AllegroTest_pads_1.pcb, AllegroTest_pads_2.pcb, and so on.

A log file, with the same name as the resulting .pcb file, is saved in the \PADS Projects folder by default.

Related Topics

Importing and Exporting Files in the *Concepts Guide*

Exporting ODB++ Files

You can export an entire design to an ODB++ job, which can be transferred from computer to computer without any loss of data.

Warning: ODB++ export is based on the CAM documents for your design. If you do not create the CAM documents and proceed without them, a default set of CAM documents will be generated and used for the ODB++ export. The default CAM documents may not be similar to what you would create.

Procedure

1. **File** menu > **Export**
2. In the File Export dialog box, select the location for the job from the **Save in** list.
3. In the Save as type list, select **ODB++ (*.tgz)**.

   Requirement:

   - If you have copper pours or plane areas that are not filled, they are poured automatically when the CAM documents are being generated.
   - If you haven't created all the required CAM documents for your design you are prompted with a list of missing documents. You can cancel the operation and create

those documents, or you can continue and use CAM documents created using some default settings.

- If you haven't created any of the required CAM documents for your design you are prompted to generate the default documents.

4. Make appropriate settings in the ODB++ Export dialog box.
5. Click **Save.**

Result

The ODB++ job, consisting of a .tgz file and associated report file (_odbppExport.rep), is created and placed in the location indicated in step 2. For example, if your file is preview.pcb, the resulting files will be preview.tgz and previewodbpp_Export.rep in the same directory.

Exporting CCE Files

You can export design elements in CCE (formerly CC/CCZ) Files for import to CAMCAD, visECAD, and visEDOC.

Procedure

1. **File** menu > **Export**.
2. In the File Export dialog box, in the Save as type list, select **CCE Files (*.cce)**.
3. Browse to overwrite a file or type a new file name.
4. Click **Save**.
5. In the CCE Export dialog box, specify the Solder Mask and Paste Mask options.
6. Choose the output file format.
7. Click **OK**.

Related Topics

CCE Export Dialog Box

Creating HyperLynx BoardSim - HYP Files

You can export the current design to a HYP file to perform high-speed analysis in BoardSim. You can also launch and open the file in BoardSim from the Tools menu if you have it installed. For information about the format of the HYP file and creating a "BoardSim-friendly" design, see the HyperLynx online Help.

PADS Layout passes the Value, Tolerance, Voltage, HyperLynx, and PowerGround attributes to the HYP file. BoardSim uses these attributes to obtain values for resistors and capacitors, and to transfer information about fixed voltage nets.

Procedure

1. **There are two methods to start the process:**
    - **If you have HyperLynx BoardSim installed, on the Tools** menu, point to **Analysis** and click **Signal/Power Integrity**. This opens the BoardSim Dialog Box.
    - If you don't have HyperLynx BoardSim installed, on the **File** menu, click **Export.** In the File Export dialog box, in the Save as type list, select **HYP Files (*.hyp)**. Browse to overwrite a file or type a new file name. Click **Save**. This opens the HYP Export Dialog Box.
2. In the BoardSim or HYP Export dialog box, in the Output area, specify the information that you want to export.
3. If you selected the Unrouted check box, do the following:
    a. In the Assumed Layer list, select the layer on which to implement the unrouted nets.
    b. In the Percent Excess over Manhattan Length box, type a value to estimate the routing lengths.

       Tip: This value adds a percentage of the Manhattan length to the route length, to account for indirect routing paths. Net lengths are based on the Manhattan distance between pin pairs, which is Delta X plus Delta Y.
4. Click **OK**.

Result

If you used the BoardSim dialog box, BoardSim will start up and open the file. If you used the HYP Export dialog box, the .hyp file is created. If you selected the .REF IC Automapping file option, the .ref file is created and has the same location and name as the .hyp file.

Warning: Nets bridged with copper are combined when you export the design into the .hyp (HyperLynx) file format.

Related Topics

BoardSim Dialog Box

HYP Export Dialog Box

Exporting an IPC-D-356 Netlist

You can export either an IPC-D-356 netlist or an IPC-D-356 Revision A netlist.

Procedure

1. On the **File** menu, click **Export**.
2. In the File Export dialog box, in the Save as type list, select **IPC356 Files (*.ipc)**.
3. Browse to overwrite a file or type a new file name. Click **Save**.
4. In the IPC Export dialog box, select one of the two netlist formats.
5. Click **OK**.

Result

Warning: Nets bridged with copper are combined when you export the design into the .ipc (IPC356) file format.

Related Topics

The IPC-D-356 Netlist

Chapter 14
Design and Editing Basics

In this chapter:

Selection

Selecting Items

Most design work involves editing the database and adding, modifying, and deleting items. There are two ways to edit items. You can first select the object to edit and then select the command, select mode; or you can first select the command and then select the object to edit, verb mode.

Verb mode attaches a command to the pointer and performs that command on any items subsequently selected. The advantage of using verb mode is that it automatically sets the selection filter to items that respond to the active command.

To Select Using Select Mode

To select the object and perform a command:

1. Point to the object and click. The selected item highlights.
2. Use one of the following:
3. Right-click to open a menu of commands that are appropriate for the selected item.
4. Click a menu command.

5. Click a toolbar button to start a command.

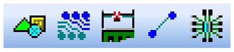
Advantages: Convenient for objects that are already selected.

To Select Using Verb Mode

To perform a command and select the object:

1. Open one of the four toolbars by clicking one of the following buttons on the main toolbar. Each toolbar provides a specific set of additional buttons.

Figure 14-1. Toolbar Buttons



2. Click one of the buttons on the newly opened toolbar. The command attaches to the pointer and the message line displays the command name. Now you can apply the command to objects that you select. When you move the pointer off the toolbar, a small V appears on the pointer to show that the selected command is active.
3. Select objects on which to perform the active command. An object is one discreet item in the design, such as a route segment or part. Click the **Select** button to cancel the command.

   Example:

   From the **Routing** toolbar, click the **Spin** button. The small V attaches to the pointer. Select a component. The component is in Spin Mode. Indicate a new orientation or press **Esc** to cancel the spin.

Tip: Press Esc before you click a button to make sure nothing is selected. If something is selected when you click a button, the command acts on the selected item.

To select all objects in an area, hold the mouse button down and drag a selection rectangle around one or more objects; start at one corner of the area and drag to the diagonally opposite corner.

When you release the button, all objects contained within the rectangle are selected.

You can add additional objects to the selection or remove objects from the selection by pressing Ctrl while you click.

Tip: The Drag Moves area in the Global tab of the Options dialog box can affect your ability to area select in dense designs. If a selected object starts to move when you area select, click Cancel from the shortcut menu and try starting in a different area. To disable drag moves, set the Drag Moves area to No Drag Moves.

To Select a Single Object

To select a single object, point to the object and click. The object becomes selected and highlighted. Any previously selected objects are no longer selected. If you click over empty space, all previously selected objects are no longer selected. If you try to select an object in a dense or crowded area, use the Selection Filter to disable other items for selection.

To Select Multiple Objects

To select multiple objects, press and hold Ctrl while you click over each item in sequence.

If an object was not previously selected, it is added to the set of selected objects. If an object was previously selected, it is removed from the set of selected objects.

Using the Selection Filter

Sometimes you cannot easily select the object you want because there are several objects at the same location. Use the Selection Filter to help solve this problem. Use the Selection Filter to specify objects on certain layers that you can or cannot select. The Selection Filter has two tabs, the Object tab and the Layer tab.

Tip: You can also set the Selection Filter using a shortcut menu. With nothing selected, right-click, then set the filter.

To Select Using the Selection Filter Object Tab

To select using the Selection Filter Object tab:

1. **Edit** menu > **Filter**.
2. Select the design and drafting items in the Selection Filter Object tab by clicking the appropriate check box.
3. Use the Anything or Nothing buttons as necessary.
4. To select layers, click on the **Layer** tab and make your selections.
5. Click **Close**.

To Select Using the Selection Filter Layer Tab

To select using the Selection Filter Layer tab:

1. **Edit** menu > **Filter**.
2. Click on the **Layer** tab.

3. Select the layers from the layer list by clicking on the appropriate check box.
4. Use the Anything or Nothing buttons as necessary.
5. Click **Close**.

Related Topics

To Cycle Pick

To Cycle Pick

When you cannot easily select the object you want because several objects occupy the same location use cycle picking:

1. Move the pointer over the object to select, and left-click.
2. If you did not select the correct object, you can either:
   - Click the **Cycle Pick** button from the standard toolbar.

     or
   - Click **Cycle** on the **Edit** menu.

     or
   - Press **Tab**.

     or
   - Right-click and click **Cycle**.

     Any of these actions deselects the currently selected object and selects a new object at the same location.
3. You can continue to click the **Cycle Pick** button to cycle the current selection through all objects at the pointer location.

Related Topics

Using the Selection Filter

Selecting Stitching Vias

When you are working with vias in a design, you may want to select all of the stitching vias in a shape in order to modify them. You can select all stitching vias contained by a copper, copper pour, or hatch outline.

To select all stitching vias in a shape:

1. Select the copper, copper pour or hatch outline.

2. Right-click and click **Select Stitching Vias**.

Selecting Isolated Stitching Vias

An isolated stitching via is not connected to any hatch outline or copper area. You can locate such vias by selecting them. You can select all isolated stitching vias in the entire design or just those associated with a specific net.

To select isolated stitching vias in the entire design:

- With no object selected, right-click in the design area and click **Select Isolated Stitching Vias**.

To select isolated stitching vias associated with a net:

1. Select a net.
2. Right-click and click **Select Isolated Stitching Vias**.

Restriction: The search for isolated stitching vias ignores connections to CAM plane layers.

Tip: When you verify your design, you can find isolated stitching vias by performing a Connectivity Check; however, you first need to set up that check. For information, see Setting Up Checking for Isolated Stitching Vias.

Running the Selection Report

The selection report is a quick way to determine everything you have selected in your design. This report is a great way to ensure that you are moving and editing exactly what you want.

Tip: The Selection Report is available only when at least two objects of different types are selected in your design.

Procedure

- **Select more than one type of object, right-click and click Selection Report.**

  Or

- Select more than one type of object, **View** menu > **Selection Report**.

Result

report.rep opens in your default text editor and displays everything you have selected in your design at this moment.

Finding Objects

Use Find to find and select single or multiple objects by reference designator, part type, line width, or other attributes.

Find works two ways, depending on your selection mode:

- **Select mode** — Find ignores the Selection Filter settings and selects whatever you ask it to.
- **Verb mode** — Find only looks for items that are logical for verb mode.

Commands for Find By

A few of the Find By commands present more complex commands for finding. Those commands are:

- Find by Attribute
- Find by keepout
- Find by fonts
- Find by physical design reuse
- Find by test point types
- Find by thermal attributes
- Find by pour area and isolated pour

To Find By Attribute

You can find by attribute name or attribute value.

1. **Edit** menu > **Find**.
2. Click **Attribute** from the **Find By** list, a list of attributes in the design appears in the Attribute list box.
3. To filter the attributes, select an attribute in the Attribute list or type a wildcard or expression in the Value box and click **Apply**.
4. Click an attribute, attribute values appear in the list box on the right.

Because the list box on the right allows multiple selections, you can find by multiple values.

To Find By Keepout

1. **Edit** menu > **Find**.

2. Click **Keepouts** from the **Find By** list, a list of keepout restrictions in the design appears in the list box on the left.
3. To filter the keepouts, type a wildcard or expression in the Value box and click **Apply**.
4. If you select a Component Height restriction, a list of available heights appears in the list box on the right.

Finding Fonts

Use the Find dialog box to find fonts in labels and text strings in your design. Then use the Properties dialog box to set up new fonts and font styles for:

- Finding Label Fonts
- Finding Text Fonts

Finding Label Fonts

1. **Edit** menu > **Find**.
2. Select **Label Fonts** from the Find by list.
3. In the Text fonts area, click the font to find. The list includes all fonts in the design.

   Tip: Once you select a text font, the Styles area is populated with the list of font styles used in the design.
4. In the Styles area, click the style to find. The list includes all font styles in the design.
5. Click **OK**. All labels with the chosen font characteristics are selected.
6. Use the Properties dialog box to apply a new font and/or style to the highlighted selections.

Finding Text Fonts

1. **Edit** menu > **Find**.
2. Select **Text Fonts** from the Find by list.
3. In the Text fonts area, click the font to find. The list includes all fonts in the design.

   Tip: Once you select a text font, the Styles area is populated with the list of font styles used in the design.
4. In the Styles area, click the style to find. The list includes all font styles in the design.
5. Click **OK**. All text strings with the chosen font characteristics are selected.
6. Use the Properties dialog box to apply a new font and/or style to the highlighted selections.

See also: Replacing Fonts

To Find By Physical Design Reuse

You can find by reuse type or by reuse name.

1. **Edit** menu > **Find**.
2. Click **Reuse Type** from the **Find By** list, a list of reuse types in the design appears in the list box on the left.
3. To filter the reuse, type a wildcard or expression in the Value box and click **Apply**.
4. Click a reuse type, a list of reuse names (names for each instance of the type) appears in the list box on the right.

To Find By Test Point Types

1. **Edit** menu > **Find**.
2. Click **Test Point Types** in the **Find By** list.
3. Click to search by **Via**, **Component pin**, or **Net** in the **Test Point Types** list. The contents of the list box at the right of the dialog box changes depending on what you select. Table 14-1 shows what the List Box displays for each of the available selections

Table 14-1. Test Point Types List

Selection	List Box Contents
Via	Lists and sorts, in order, all types for test points on vias.
Component pin	Lists and sorts, in order, all nets containing test points on component pins. Jumper pins are treated as component pins by Find. If the design has at least one unused component pin with a test point attribute, the name Unused Pins appears at the bottom of the list box. All unused pins with the test point attribute are also searched for test points.
Nets	Lists all nets with test points, either on component pins or vias. If the design has at least one unused component pin with a test point attribute, the name Unused Pins appears at the bottom of the list box.

4. To filter the test points, type a wildcard or expression in the Value box and click **Apply**.

Tip: Find treats jumper pins as component pins.

The search differs depending on which list box has the current selection, the Test Point Types list or the one to the right of it. If the list on the right is active, Find searches only for test points from selected nets. Otherwise, Find searches for all test points of selected types from all nets.

To Find by Thermal Attributes

1. **Edit** menu > **Find**.
2. Click **Thermal Attributes** from the **Find By** list.
3. To filter the attributes, type a wildcard or expression in the Value box and click **Apply**.

The list on the left lists thermal data by inner pad size. The list on the right lists a breakdown of thermals using the following format: outer/number of spokes/spoke width/rotation format.

To Find By Pour Area and Isolated Pour

The Find By list box contains two commands for handling copper pour: Pour Area and Isolated Pour.

Pour Area

Selects copper pour based on area size. Type the size in the Value field. All copper pour areas of equal size or less are selected. Press Delete to delete them.

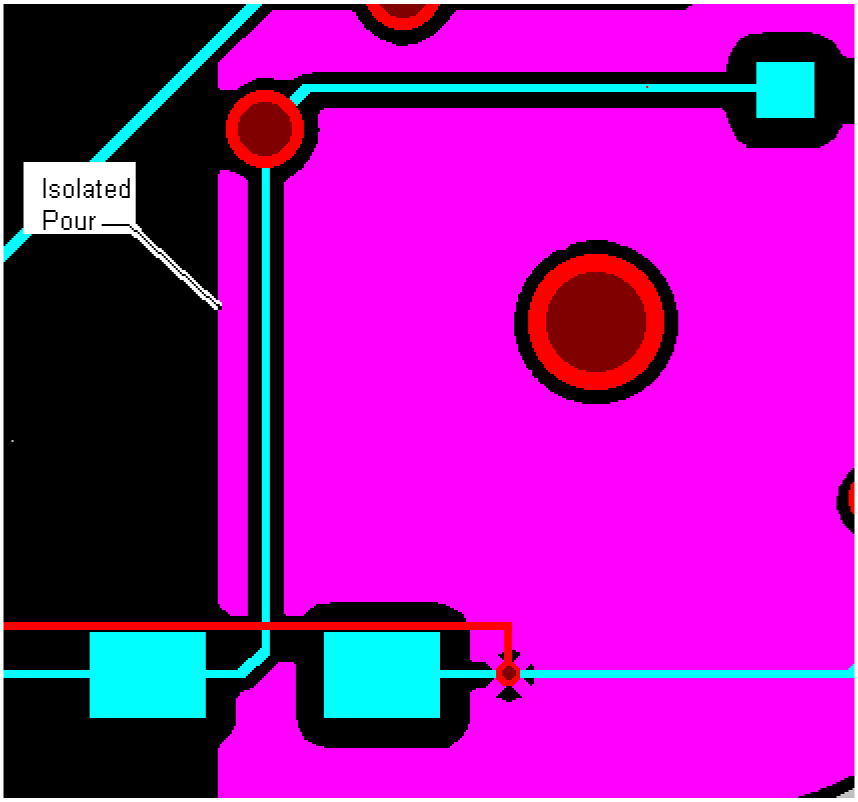
Isolated Pour

Locates areas of copper pour that have been isolated from the main pour area, but are still part of the same netname. Cleared obstacles that are too close to the edge of the pour outline usually cause isolated copper pour.

After you select the isolated copper areas, press Delete to delete them.

Tip: Pour areas not assigned a netname are not considered isolated; they are excluded from this operation.

Figure 14-2. Isolated Pour



Related Topics

Find Dialog Box

Using the Find Dialog Box During Placement

To Highlight Objects

You can highlight an object without performing an operation on it.

1. Select the objects to highlight.
2. Click **Highlight** from the **Edit** menu to show the object as highlighted. Subsequent edit operations do not apply to these highlighted objects.
3. Click **Unhighlight** from the **Edit** menu to remove highlighting from all objects.

To Use Transparent View Mode

Use Transparent Mode to view traces on several layers simultaneously. Transparent mode can show you obstacles that may be hidden directly under the current active layer.

To turn on and off transparent view mode:

- Use the Modeless Command T.

Related Topics

To Use Outline View Mode

Viewing Protected Routes with Outline Mode

To Use Outline View Mode

Use Outline mode to speed redraw time by displaying traces and pads as outline objects instead of filled objects. Traces appear as two parallel lines, separated by the established trace width. Pads appear as outlines of their shapes.

To turn on and off outline view mode:

- Use the Modeless Command O.

Related Topics

To Use Transparent View Mode

Viewing Protected Routes with Outline Mode

Measuring

You can measure clearance distances or objects in your design using two modeless commands.

Using Quick Measure

Use the Q modeless command to measure using a dynamic ruler or measuring stick.

Restriction: This modeless command is restricted by the Snap to Design Grid setting. To get the most accurate measurement, do not snap to the design grid.

1. Locate your pointer at the starting location of your measurement. Without further movement of the pointer, type the **Q** modeless command and then type **Enter**.
2. Move the pointer in any direction and a line will be drawn indicating the span of your measurement. Three measurement values are displayed as you move the mouse pointer.
    - dx - displays the delta measurement along the x axis.
    - dy - displays the delta measurement along the y axis.
    - d - displays the delta of the euclidean measurement between the start and end point.
3. When finished, type **Esc** to exit the Quick Measure mode.

Using Quick Length

Use the QL modeless command to measure route objects you've selected in the design.

1. Select a net, pin pair, or one or more routed segments in the design.
2. Type the **QL** modeless command and then type **Enter**.

   The measurement of the selected route objects is given in a Layout.err report file which opens. The Output Window also logs a link to the report file for easy return-access.

Related Topics

Modeless Commands

Clearances and Keepouts

Using the Trace Length Monitor

Chapter 15
Setting Up Layers

Use the following steps to set your layers up. Based on your needs, you may or may not need to do every step.

Procedure

1. **Setup** menu > **Layer Definition**.
2. Set the number of layers for the design.
   - Increase the total number of layers available to the design.
   - Designate that this is a single-sided board.
   - Change the number of electrical layers.
3. Set up the outer layers.
4. Set up the inner layers.
5. Determine the visibility of non-electrical layers.
6. Define layer and substrate thickness.

Increasing the Maximum Number of Available Layers

PADS Layout supports two layer modes, default layer mode and increased layer mode.
See also: Layer Modes in the *Concepts Guide*

When you switch a design to increased layer mode, you cannot return to default layer mode.

Requirements

You must set the proper number of electrical layers in PCB Decal Editor before switching layer mode.

Procedure

1. **Setup** menu > **Layer Definition**.
2. Click **Max Layers**. The Increase Maximum Layer Number dialog box opens.
3. Click **OK** to switch to increased layer mode.

Designating a Board as Single-sided

You can only designate a board as single-sided if there are no more than two electrical layers.

Procedure

1. **Setup** menu > **Layer Definition**.
2. Click **Single-sided Board**.
3. Click **OK**.

Result

- Connectivity checking no longer reports connectivity errors for component pins with non-plated drill holes. Components and jumpers placed on the top layer are considered as connected to pads on the bottom layer with solder joints.
- In CAM output, all through-hole pins and vias are treated as non-plated regardless of the definition in pad stacks.

Modifying the Number of Electrical PCB Layers

You can change the number of electrical layers in a design in either default layer mode or increased layer mode.

Requirement

Before changing the number of electrical layers, delete all partial vias and their definitions from your design. Also, you can't modify the layer definition for electrical layers if a physical design reuse exists on the board.

Procedure

1. In the Layers Setup dialog box, click **Modify**. The Modify Electrical Layer Count dialog box opens.
2. Type the new number of electrical layers in the Modify Electrical Layer Count within the specified range.
3. Click **OK**. The Reassign Electrical Layers dialog box opens.
4. If necessary, reassign the electrical information on any existing electrical layer to a new layer. The new electrical layers are created in the database with default parameter values.
   See also: Reassigning Electrical Layers
5. Click **OK** to return to the Layers Setup dialog box.

Tip: If you increase your electrical layer count, and intend to use partial vias, you should also update your drill pairs layer settings.

Setting Up an Outer Layer

The outer layers of your design are the Top and Bottom layers. You can designate an outer layer to have components or not.

Procedure

1. In the Layers Setup dialog box, select an outer layer. For example, select Top.
2. In the Name box, type the name of the layer.
   Tip: By default, PADS Layout assigns the outer layer names as Top and Bottom. Rename the layer to something that lets you know what the layer is; for example, Top Component Layer 1.
3. In the Electrical Layer Type area, click **Component** if you want to allow components on the layer; otherwise click **Routing**. If you select Component, you can change the layer associations:
   a. Click the **Associations** button.
   b. In the Component Layer Associations dialog box, set the associations for each documentation layer type.
   c. Click **OK** to close the dialog box.

      Tip: When a top or bottom layer is set as a component layer, you can associate, or otherwise map, which documentation layers go with the selected layer.

      Whatever layer associations are made here are used by CAM routines for output. For example, when you output a silkscreen for the top, any items on the documentation layer you associated for silkscreen are automatically added to the CAM document.
4. In the Layers Setup dialog box, click the Plane Type you want. If you select CAM or Split/Mixed Plane, you must assign the plane net(s) to the layer:
   a. Click the **Assign Nets** button.
   b. Click a net from the **All Nets** list.
   c. Click **Add** to move the net to the Assigned Nets list.
   d. You can associate additional nets if the layer is a Split Mixed Plane type.
   e. Click **OK** to close Plane Layer Nets dialog box.

      You can assign a net to as many layers as required.
5. In the Layers Setup dialog box, click the Routing Direction you want.

Tip: You must assign a primary routing direction to all electrical layers. Nonelectrical layers are not assigned a routing direction. The routing direction affects the manual and autorouting performance. For example, if you select Horizontal but most of the traces on the layer need to be vertical, route editing performance is slow. Also, selecting Any can adversely affect route editing performance.

6. Click **OK**.

Related Topics

Drawing a Plane or Using a CAM Plane

Setting Up an Inner Layer

Setting Up a Documentation Layer

Associating Component and Documentation Layers in the *Concepts Guide*

Setting Up an Inner Layer

The inner layers of your design are always electrical layers.

Procedure

1. In the Layers Setup dialog box, select an inner layer. For example, select Ground Plane.
2. In the Name box, type the name of the layer.
   Tip: By default, PADS Layout assigns the layer number as its name; for example, Inner Layer 2. Rename the layer to something that lets you know what the layer is; for example, Inner Signal Layer 2.
3. Click the Plane Type you want. If you select CAM or Split/Mixed Plane, you must assign the plane net(s) to the layer:
   a. Click the **Assign Nets** button.
   b. Click a net from the **All Nets** list.
   c. Click **Add** to move the net to the Assigned Nets list.
   d. You can associate additional nets if the layer is a Split Mixed Plane type.
   e. Click **OK** to close Plane Layer Nets dialog box.

   You can assign a net to as many layers as required.
4. In the Layers Setup dialog box, click the Routing Direction you want.

   Tip: You must assign a primary routing direction to all electrical layers. Nonelectrical layers are not assigned a routing direction. The routing direction affects the manual and autorouting performance. For example, if you select Horizontal but most of the traces on the layer need to be vertical, route editing performance is slow. Also, selecting Any can adversely affect route editing performance.

5. Click **OK**.

Related Topics

Drawing a Plane or Using a CAM Plane

Setting Up an Outer Layer

Setting Up a Documentation Layer

Setting Up a Documentation Layer

PADS Layout assigns layers 21 to 30 (in default layer mode) as default documentation layers for manufacturing plot types, including two for silkscreen: top and bottom.

Procedure

1. In the Layers Setup dialog box, select a documentation layer.
   Tip: Documentation layers are listed after all electrical layers. A quick way to tell if a layer is a documentation layer is the absence of a letter in the Direction (dir) column.
2. In the Name box, type the name of the layer.
   Tip: By default, PADS Layout assigns the layer number as its name; for example, Layer_3. Rename the layer to something that lets you know what the layer is; for example, Gold Mask.
3. Click the Fab. Assembly and Documentation Layer Type you want.
   Tip: If you click SilkScreen, Paste Mask, Solder Mask, or Assembly, this layer is available for association with component layers.
4. Click **OK**.

Related Topics

Associating Component and Documentation Layers in the *Concepts Guide*

Setting Up an Inner Layer

Setting Up an Outer Layer

Component Layer Associations dialog box

Hiding or Displaying Non-electrical Layers

Default PADS Layout designs come with many layers. If your design only uses a few layers, there might be a large number of layers that are unused. You can hide those layers from view to shorten the list of layers wherever they're displayed - for example, the Layer list of the standard toolbar, the Layers Setup and Display Colors dialog boxes. You can set the visibility for non-electrical layers only.

Tip: The best use of this feature is if you never intend to use the layer; if you need to use a layer, but want to switch the visibility of it, use the Display Colors Setup dialog box instead.

Procedure

1. In the Layers Setup dialog box, click **Enable/Disable**.
2. Select the appropriate check box in the **Enabled** column to either enable or disable a layer.

   Tip: While the **Enabled** column is the only editable column, you can sort the displayed layers by any other column. You can click a column heading to sort by that column's values. Click the same column heading again to sort in the reverse order.
3. Click **OK** to return to the Layers Setup dialog box.

Setting Layer Thickness

Use the Layer Thickness dialog box to define electrical layer and dielectric material layer thickness and dielectric constant information. When you verify your design, the electrodynamic check uses this information.

Traces on high-speed printed circuit boards can act like transmission lines that “broadcast” interference to adjacent conductors. Using the high-speed rules module, you can use Rules to set clearances on a net class, net, or pin to pin connection basis; then use high-speed checking to report on properties such as impedance, delay, track length, daisy chaining, and parallel routing. These issues cause interference and create costly problems in prototyping. You can run checks against the entire board or against specific nets.

Requirements

- Set these definitions before you run an electrodynamic check.
- You must specify your plane layers in the Layers Setup dialog box before you run an electrodynamic check. For a two-layer board, temporarily identify one of the layers as a plane layer.

Procedure

1. In the Layers Setup dialog box, **click the Thickness** button. The Layer Thickness dialog box opens.
2. For each dielectric material layer, double-click the **Type cell** to select whether the “layer” is a Prepreg or Substrate layer.

3. For each layer, double-click the **Thickness cell** and type a value.

   Tip: If no coating is required, set thickness to zero.

4. For dielectric material layers, double-click the **Dielectric cell** and type a dielectric constant value.

5. To view and edit copper thicknesses by ounces per square foot, click **Weight (oz)**. Otherwise, click **Design (")** to view and edit in the same unit of measure as the current database.

Tip: Board Thickness is the total value of material and layer thicknesses in the current design units.

Related Topics

Verify the Design

Setting Up High Speed (Electrodynamic) Checking

Setting Up EDC Parameters

Unassigning a Netname from a Plane Layer

Procedure

1. In the Layers Setup dialog box, select a layer defined as a CAM Plane or a Mixed Plane.
2. Click **Assign Nets**. The Plane Layer Nets dialog box opens.
3. Click the netname from the Assigned Nets list.
4. Click **Remove**. The netname is moved back to the All Nets list.

Reassigning Electrical Layers

When reassigning layers, component keepouts are ignored. They remain on the layer on which you created them. Also, you can't modify the layer definition for electrical layers when a physical design reuse exists on the board.

Be careful when you reassign a layer. If a route is attached to a pad that is not available on the new layer, such as a surface mount pin or a partial via, the program places a zero-length unroute from the end of the trace to the component pin.

Procedure

1. In the Layers Setup dialog box, click **Reassign**. The Reassign Electrical Layers dialog box opens.
2. Click the number of the layer you want to reassign from the **Old** list.

3. Type the layer number you want to assign it to in the **New Layer #** box. You cannot merge items from an old to a new layer, but you can swap layers. Additionally, the target layer must be empty.

4. The following information moves from the old layer to the new layer:

   - Traces and vias
   - Drafting objects
   - Layer name, layer type, routing direction, and component/plane layer parameters

5. If you decrease the number of layers and the selected layer has no data, you can remove the layer. To remove it, click **Delete**.

6. Click **OK**. You return to the Layers Setup dialog box.

Chapter 16
Via Setup

In this section, you learn how to create, edit, and delete vias. You also learn how to set up Drill Pairs and tent vias.

- Creating a Drill Pair
- Creating a Through-hole Via
- Creating a Partial Via
- Editing a Via
- Deleting a Via
- Tenting Vias With Solder Mask

Creating a Drill Pair

Layers are expressed as a pair of numbers, or a drill pair. Define drill pairs first to prevent defining or installing partial vias that cross layers that don't drill together.

You must create a drill pair for each partial via in your design. If you are using through-hole vias in addition to partial vias, you must also create an “all-layers” drill pair; once you create a partial via, PADS Layout no longer assumes you are using the through-hole via default.

Procedure

1. **Setup** menu > **Drill Pairs**. The Drill Pairs Setup dialog box opens.
2. Click **Add** to add a line to the Drill Pairs list.
3. Select the starting layer from the **Starting Layer** list. The layer number is automatically indicated to the left of the list.
4. Select the ending layer from the **Ending Layer** list. The layer number is automatically indicated to the left of the list.
5. Click **OK** to save the drill pairs.

You can edit the starting or ending layer for a drill pair, and you can delete drill pairs.

Tip: You can use up to 30 electrical layers when adding a partial via in default layer mode. You can use up to 64 electrical layers when adding a partial via in increased layer mode.

Related Topics

Creating a Partial Via

Creating a Through-hole Via

Drill Pairs Setup Dialog Box

Creating a Through-hole Via

You can create a through-hole via or a partial via.
See also: Creating a Partial Via

Requirements

- If you are using partial vias in addition to through-hole vias, you must create an “all-layers” drill pair; once you create a partial via, PADS Layout no longer assumes you are using the through-hole via default.
  See also: Creating a Drill Pair

Procedure

1. **Setup** menu > **Pad Stacks**. The Pad Stacks Properties dialog box opens.
2. In the Pad Stack Type area, select **Via**.
3. Click **Add Via** to add a line to the Decal name list.
   Tip: A list of all previously defined vias appears in the Decal Name list.
4. In the Vias area, type the name of the via in the **Name** box.
5. Click **Through** to indicate that this is a Through-hole via.
6. *Select the layer of the pad you want to customize* from the ***Sh: Sz: Layer:* list**.

   See also: Pad Stack Default Layers, Control of Solder Mask and Paste Mask
7. In the Parameters area, specify the settings for all three **Pad**, **Thermal**, and **Antipad** pad styles if needed. The style of pad used is selected automatically in the design depending on the situation. The thermal and antipad styles are used when the pin is located within a plane.

   See also: Design Rule Versus Pad Stack - Thermals and Antipads
8. Click **OK** to save the via pad stack.

Related Topics

Pad Sizes and Pad Stacks

Editing a Via

Creating a Drill Pair

Creating a Partial Via

Pad Stacks Properties Dialog Box

Creating a Partial Via

You can create a through-hole via or a partial via. Partial vias are also called *blind via* or *buried via* depending on how visible they are.
See also: Creating a Through-hole Via

Requirements

- You must first create a drill pair for each partial via.
  See also: Creating a Drill Pair
- If you are using partial vias in addition to through-hole vias, you must create an "all-layers" drill pair; once you create a partial via, PADS Layout no longer assumes you are using the through-hole via default.

Procedure

1. **Setup** menu > **Pad Stacks**. The Pad Stacks Properties dialog box opens.
2. In the Pad Stack Type area, select **Via**.
3. Click **Add Via** to add a line to the Decal name list.
   Tip: A list of all previously defined vias appears in the Decal Name list.
4. In the Vias area, type the name of the via in the **Name** box.
5. Click **Partial** to indicate that this is a blind or buried via.
6. Select the starting and ending layers from the **Start Layer** and **End Layer** lists.
7. *Select the layer of the pad you want to customize* from the ***Sh: Sz: Layer:*** **list**.

   See also: Pad Stack Default Layers, Control of Solder Mask and Paste Mask
8. In the Parameters area, specify the settings for all three **Pad**, **Thermal**, and **Antipad** pad styles if needed. The style of pad used is selected automatically in the design depending on the situation. The thermal and antipad styles are used when the pin is located within a plane.

   See also: Design Rule Versus Pad Stack - Thermals and Antipads
9. Click **OK** to save the via pad stack.

Troubleshooting

If you are unable to use the new partial via, check the following in the Default Routing Rules:

- Is your new via in the Selected vias list and available for use in the design?
- Are both the from and to layers listed in the Selected layers list and available for routing?

Related Topics

Editing Pad Stacks

Editing a Via

Creating a Drill Pair

Creating a Through-hole Via

Pad Stacks Properties Dialog Box

Editing a Via

To edit a via pad stack and apply the changes to the part you selected or all parts:

1. Select a via > right-click > **Properties** > **Pad Stack** button.
2. Click the via type to edit in the **Decal Name** list.
3. Change via settings as needed.
4. When you finish making changes, click **OK**.
5. Click **Yes** to change all vias of the selected type.

Tip: Use the Parameters and Drill Size areas to reset the size and shape options.

Related Topics

Editing Pad Stacks

Creating a Through-hole Via

Creating a Partial Via

Pad Stacks Properties Dialog Box

Deleting a Via

1. **Setup** menu > **Pad Stacks**.
2. Click **Via** in the **Pad Stack Type** area.

3. Click a via name in the **Decal Name** list.
4. Click **Delete Via**.
5. Click **OK**.

Related Topics

Editing Pad Stacks

Creating a Through-hole Via

Creating a Partial Via

Pad Stacks Properties Dialog Box

Tenting Vias With Solder Mask

You can cover vias with solder mask (a process known as "tenting" the via.)

Choosing a Method

Method 1—You use the Pad Stacks dialog box to add a custom solder mask shape to the solder mask layer. Use this method if you want to be able to see the various sizes of solder mask shapes in the design.

Method 2—You use the Object Attributes dialog box to add an attribute that controls the solder mask openings. This is the quickest method, but you will be unable to see the shapes in the design--you have to create a CAM document to see them.

Warning: If a solder mask shape exists in the solder mask layer of the pad stack, and a via has an assigned **CAM.Solder mask.Adjust** value, and both values are used in the CAM document, they will be combined. See Mask Hierarchy in the PADS Layout Concepts Guide.

Method 1 Procedure

1. **Setup** > **Pad Stacks**
2. In the Pad Stack Type area of the Pad Stacks Properties Dialog Box, click the **Via** radio button.
3. Add a solder mask layer to the via you want to be tented, as follows:
   a. In the Decal name list click the name of the via.
   b. In the Vias area Name box, append **_OPEN** to the via name.
   c. In the Sh: Sz: Layer area, click **Add**, and from the Add Layer dialog box select either **Solder Mask Top** or **Solder Mask Botto**m.

   d. With the new Solder Mask layer selected, set the pad diameter to the oversize required for the via.

   e. If you need to add the other solder mask layer, repeat steps c and d.

4. Create a duplicate of the via you want to be tented, as follows:

   a. With the <*VIA_NAME*>_ OPEN via selected, click the **Add Via** button.

   b. In the Vias area, type ***<ORIGINAL_VIA_NAME>*_TENTED**.

   c. In the Sh: Sz: Layer area, click **Add**, and from the Add Layer dialog box select either **Solder Mask Top** or **Solder Mask Botto**m.

   d. With the new Solder Mask layer selected, set the pad diameter to zero.

   e. If you need to add the other solder mask layer, repeat steps c and d.

   f. Click OK to save the settings and close the dialog box.

5. Tent the vias in the design, as follows:

   a. In the design, select the vias you want to be tented.

   b. **Right-click > Properties**

   c. From the Via Name list of the Via Properties Dialog Box, select the name of the new tented via.

   d. Click **OK**.

Results

To verify your results:

1. **File > CAM**
2. In the Define CAM Documents Dialog Box, click the **Add** button.
3. In the Add/Edit CAM Document Dialog Box, from the Document Type list, select **Solder Mask**.
4. In the Layer Association dialog box, select the top/bottom layer.
5. Click the **Layers** button.
6. In the Select Items Dialog Box Selected list, select **Solder Mask Top**.
7. In the Items on Primary area, select the **Vias** check box.
8. Click the **Preview** button. The output will show openings for the “open” vias, but the “tented” vias will not appear.

Method 2 Procedure

1. In the design, select the vias you want to be tented.
2. **Right-click** > **Attribute**
3. In the Object Attributes Dialog Box, click the **Add** button.
4. From the drop-down list in the Attribute column, select **CAM.Solder mask.Adjust**.
5. Double-click in the edit box under the Value header, type a negative value that represents the via pad size, and press **Enter**. For example, enter -20 to tent a 20 mil via pad.
6. Click Close.

Results

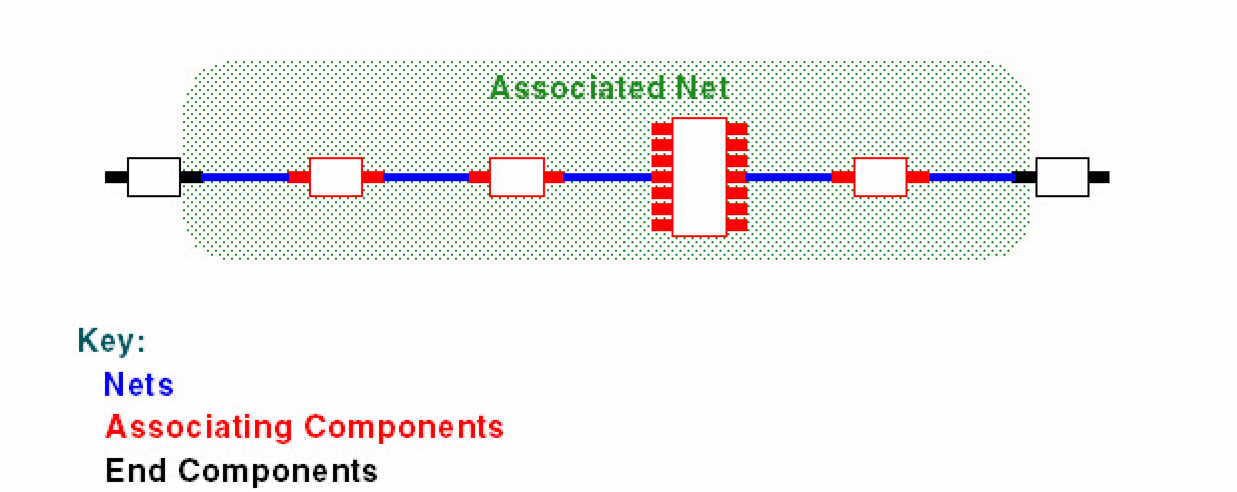
To verify your results:

1. **File** > **CAM**
2. In the Define CAM Documents Dialog Box, click the **Add** button.
3. In the Add/Edit CAM Document Dialog Box, from the Document Type list, select **Solder Mask**.
4. In the Layer Association dialog box, select the top/bottom layer.
5. Click the **Layers** button.
6. In the Select Items Dialog Box Selected list, select **Solder Mask Top**.
7. In the Items on Primary area, select the **Vias** check box.
8. Click the **Preview** button. The output will show openings for the “open” vias, but the “tented” vias will not appear.

Chapter 17
Associated Nets

You can associate an array of nets joined by discrete components, creating an *associated net*, to which you can apply rules as you would to a single net. The length of an associated net is the combined lengths of the nets and discrete components of which it is composed. The following figure shows an example of an associated net.

Figure 17-1. Associated Net Components and Nets



Length, differential pair and matched length rules can be applied to an associated net as though it were a single net.

There are three ways to create and modify your associated nets;

- **By net**
  Select nets and associate them, either by settings in the Net Properties dialog box or by popup commands.
- **Manually by component**
  Select discrete components and associate their attached nets, either by settings in the Component Properties dialog box or by popup commands.
- **Automatically by refdes prefix**
  Specify the refdes prefixes of the components whose nets you want to associate in the Associated Nets dialog box.

By default these methods are independent of each other (that is, you can use any of the methods by itself), because the commands/properties by which they interact are defaulted to allow you to choose one method to use, and ignore the others. Choosing one method and sticking with it is the simplest course. If you mix methods in your design, you will need to understand how the methods interact, described in Conditions Governing Associated Net Creation.

The following topics describe the creation and management of associated nets:

Creating Associated Nets

There are three methods you can use to create associated nets:

- By net
- Manually by component
- Automatically by component refdes prefix

General Restrictions

The following restrictions apply to all 3 net association methods.

- A plane net cannot be part of an associated net.
- A net having more pins than the limit specified in the *Maximum non-plane-net pin count* field of the Associated Nets dialog box is considered a plane net, and cannot be part of an associated net. (See General Prerequisite (Recommended) below.)
- An associated net that would have more nets than the limit specified in the *Maximum net count per associated* field of the Associated Nets dialog box is not created. (See General Prerequisite (Recommended) below.)
- Components through which an associated net passes must be either discrete two-pin components, or multiple pin components that conform to the following conditions:
  - All the pins must connect to a gate.
  - Each gate must have exactly two pins.

No component that does not conform to these conditions can be an associating component, that is, no associated net can go through it.

General Prerequisite (Recommended)

This prerequisite applies to all 3 net association methods.

Before you begin creating associated nets, you should make sure that the *Maximum net count per associated net* and *Maximum non-plane-net pin count* fields in the Associated Nets dialog box are set appropriately for your design. These fields set limits on the number of nets allowed in an associated net, and on the number of pins allowed in a single net included in an associated net. If these values are set inappropriately for your design, associated nets that would otherwise be created are not created, or are created differently than you expect, as follows:

- If the potential associated net exceeds the *Maximum net count per associated net* limit, it is not created.
- If any net in the potential associated net exceeds the *Maximum non-plane-net pin count* limit, the associated net can be split, truncated, or not created at all, depending on the location of the faulty net.

For more information, see Conditions Governing Associated Net Creation.

Method 1—Net Association by Net

Use this method if you want to create associated nets by selecting nets and associating them.

Tip: Whenever associated nets are created, deleted or changed, all associated nets in the design are regenerated. So you should always verify that the results of an associated nets action are what you expected by reading the messages in the Output Window.

Procedure

1. In the workspace, select the nets you want to associate.
2. Right-click > **Associate Nets**

Alternative: In the Net Properties dialog box, check the *Associate net* checkbox.

Results

The selected nets are associated, and the Net Properties *Associate net* checkbox is set for all the selected nets.

Are the results different from what you expected?

- An expected associated net might have been split or truncated, or not created at all, because one or more of its nets or components did not conform to the General Restrictions for creating associated nets.

- Some nets cannot be associated; for example, two selected nets from opposite sides of the board will have the Associate net checkbox checked, but no associated net will be created.
- **Mixed-method restriction:** If a component to which one of the selected nets is attached has been excluded from net association, it cannot be an associating component, that is, no associated net can go through it. (A component or net is excluded from net association if its *Associate nets* and *Allow net association...* checkboxes are both cleared.)
  For more information, see Conditions Governing Associated Net Creation.

Related Topics

Cancelling Net Association for Nets Associated by the Net Method

Excluding Nets from Net Association

Method 2—Net Association by Component

Use this method if you want to create associated nets by selecting components and associating their nets.

Tip: Whenever associated nets are created, deleted or changed, all associated nets in the design are regenerated. So you should always verify that the results of an associated nets action are what you expected by reading the messages in the Output Window.

Procedure

1. In the workspace, select the components whose nets you want to associate.
2. Right-click > **Associate Nets**

Alternative: In the Component Properties dialog box, check the *Associate nets* checkbox.

Results

The nets attached to the selected components are associated, and the Component Properties *Associate nets* property is set for all the selected components.

Are the results different from what you expected?

- An expected associated net might have been split or truncated, or not created at all, because one or more of its nets or components did not conform to the General Restrictions for creating associated nets.
- **Mixed-method restriction:** If a net attached to a selected component is excluded from net association, it cannot be included in an associated net. (A component or net is excluded from net association if its *Associate nets* and *Allow net association...*

checkboxes are both cleared.)
For more information, see Conditions Governing Associated Net Creation.

Related Topics

Excluding Components from Net Association

Cancelling Net Association for a Component Associated by the Component Method

Method 3— Automatic Net Association by Refdes Prefix

Use this method if you want to create associated nets automatically by specifying the refdes prefixes of the associating components in the Associated Nets dialog box.

Tip: Whenever associated nets are created, deleted or changed, all associated nets in the design are regenerated. So you should always verify that the results of an associated nets action are what you expected by reading the messages in the Output Window.

Prerequisite (Recommended)

This procedure creates associated nets for *all* components that have the specified refdes prefixes, and conform to the General Restrictions for creating associated nets.

So you will want the refdes prefix of any set of components you plan to group into associated nets to be unique.

Procedure

1. **Setup** menu > **Associated Nets**
2. In the **Maximum net count per associated net** box, enter the maximum number of nets to be allowed in associated nets.
3. In the **Maximum non-plane-net pin count** box, enter the maximum number of pins allowed in nets belonging to associated nets. Nets that would have pin counts higher than this maximum are considered plane nets, and are not included in associated nets.
4. In the **Discrete component prefixes** area, enter the refdes prefixes of the components you want to be associating components in associated nets.
5. **Tip:** The text entry field names are only for convenience; prefixes for any type of component can be entered in any field.

Results

When you click OK or Apply, associated nets are regenerated based on the new or changed set of specified refdes prefixes.

Rules (min/max length, matched length groups, differential pairs) for associated nets that are unchanged in the regeneration are kept. Associated nets that are changed (including changes to the set of components they pass through), or deleted:

- Lose their rules
- Are removed from matched length groups, and
- Diff pairs containing them are removed.

Are the results different from what you expected?

- An expected associated net might have been split or truncated, or not created at all, because one or more of its nets or components did not conform to the General Restrictions for creating associated nets.
- **Mixed-method Restriction:** If any of the selected components, or any net attached to a selected component, has been excluded from net association, it cannot be included in an associated net. (A component or net is excluded from net association if its *Associate nets* and *Allow net association...* checkboxes are both cleared.)

  For more information, see the Conditions Governing Associated Net Creation.

Related Topics

Deleting Associated Nets

Excluding Nets from Net Association

Excluding Components from Net Association

Deleting Associated Nets

You can delete associated nets manually, or automatically if they have been created automatically in the Associated Nets dialog box.

Deleting Associated Nets Manually

Tips:

- Deleting an associated net deletes only the association, not the nets.

Procedure

1. In the workspace, select the associated nets you want to delete.
2. Right-click > **Delete Associated Net**

Results

The Associate Nets checkbox is cleared for all the nets of the selected associated nets. If any nets have also been associated by component (including by refdes prefix), for those nets the *Allow net association by component* checkbox is cleared as well.

Related Topics

Cancelling Net Association for Nets Associated by the Net Method

Excluding Nets from Net Association

Deleting Associated Nets “Automatically”

You can “automatically” delete a group of associated nets which you have created by the refdes prefix method.

Tip: Whenever associated nets are created, deleted or changed, all associated nets in the design are regenerated. So you should always verify that the results of an associated nets operation are what you expected by reading the messages in the Output Window.

Procedure

1. **Setup** menu > **Associated Nets**
2. Delete the refdes prefixes of the associating components.
3. Click **Apply** or **OK**.

Results

The associated nets associated by components having the specified refdes prefix are deleted, split or truncated.

Are the results different from what you expected?

- **Mixed-method restrictions:**
    - If a component having the deleted refdes prefix has the *Associate nets* checkbox checked, the nets of that component will remain associated.
    - If a component having the deleted refdes prefix has the *Allow net association by refdes prefix and by net* checkbox checked, and both nets attached to a two-pin component (or to a single gate of a multiple-pin component) have the *Associate net* checkbox checked, those nets will remain associated.

  For more information, see Conditions Governing Associated Net Creation.

Related Topics

Creating Associated Nets

Excluding Components from Net Association

Excluding Nets from Net Association

You can exclude individual nets from inclusion in any associated net.

Tips:

- This procedure clears both the Net Properties *Associate net* checkbox and the *Allow net association by component* checkbox, removing the selected nets from any current net association, and preventing net association by component (including by refdes prefix). For more information, see Conditions Governing Associated Net Creation.
- Whenever associated nets are created, deleted or changed, all associated nets in the design are regenerated. So you should always verify that the results of an associated nets operation are what you expected by reading the messages in the Output Window.

Procedure

1. In the workspace, select the net(s) you want to exclude from net association.
2. Right-click > **Disable Net Association**

Alternative: In the Net Properties dialog box, clear both the *Associate net* and *Allow net association by component* checkboxes.

Results

The selected nets are removed from existing associated nets, and the Net Properties *Associate net* and *Allow net association by components* checkboxes are cleared for all the selected nets.

Related Topics

Conditions Governing Associated Net Creation

Excluding Components from Net Association

Cancelling Net Association for Nets Associated by the Net Method

Excluding Components from Net Association

You can prevent an individual component from being an associating component, that is, a component through which an associated net passes. If you exclude a component that is not already part of an associated net, it cannot become part of an associated net created by net or by refdes prefix. In addition, if the component is already part of an associated net, it is removed from the associated net.

Tips:

- This procedure clears both the Component Properties *Associate nets* and *Allow net association...* checkboxes, preventing the selected components from being associating components. For more information, see Conditions Governing Associated Net Creation.
- Whenever associated nets are created, deleted or changed, all associated nets in the design are regenerated. So you should always verify that the results of an associated nets operation are what you expected by reading the messages in the Output Window.

Procedure

1. In the workspace, select the components you want to exclude from net association.
2. Right-click > **Disable Net Association**

Alternative: In the Component Properties dialog box, clear both the *Associate nets* and *Allow net association by refdes prefix or by net* checkboxes.

Results

The selected components are excluded from net association. If a component is part of an existing associated net, it is removed, that is, the associated net no longer goes through it. Removing a component from an associated net splits, truncates, or deletes the associated net, depending upon its configuration and the position of the removed component.

Are the results different from what you expected?

- Nets attached to the excluded component may still be associated through other components.

Related Topics

Conditions Governing Associated Net Creation

Excluding Nets from Net Association

Cancelling Net Association for Nets Associated by the Net Method

You can cancel the net association of nets associated by the net method.

Tip: Whenever associated nets are created, deleted or changed, all associated nets in the design are regenerated. So you should always verify that the results of an associated nets operation are what you expected by reading the messages in the Output Window.

Procedure

1. In the workspace or Project Explorer, select the nets you want to remove from associated nets.
2. Right-click > **Properties**
3. In the **Associated Nets** area, clear the **Associate Net** checkbox.

   Tip: If you want to exclude the nets from net association altogether, also clear the *Allow net association by components* checkbox.

Results

The selected nets are removed from existing associated nets.

Are the results different from what you expected?

- **Mixed-method restriction:** Nets that are also associated by the component and/or the refdes prefix method remain associated.

Related Topics

Conditions Governing Associated Net Creation

Cancelling Net Association for a Component Associated by the Component Method

You can cancel the net association of a component associated by the component method.

Tip: Whenever associated nets are created, deleted or changed, all associated nets in the design are regenerated. So you should always verify that the results of an associated nets operation are what you expected by reading the messages in the Output Window.

Procedure

1. In the workspace or Project Explorer, select the components whose net association you want to cancel.
2. Right-click > **Properties**
3. In the **Associated Nets** area, clear the **Associate nets** checkbox.

   Tip: If you want to exclude the components from net association by any method, also clear the **Allow net association by refdes prefix and by net** checkbox.

Results

Net association by the selected component is cancelled.

Are the results different from what you expected?

- **Mixed-method restriction:** Nets that:
  - Are attached to selected components, and
  - Are also associated by the net and/or the refdes prefix method

  remain associated through the selected components.

Selecting Associated Nets

Procedure

1. In the workspace, select one of the nets in the associated net you want to select.
2. Right-click **> Select Associated Net**

Creating a Matched Length Group of Associated Nets

Procedure

1. **Setup** menu > **Design Rules** > **Associated Nets** button
2. In the Associated Net Rules dialog box, select the associated nets you want to include in the matched length group, and click the **HiSpeed** button.
3. Make appropriate settings in the HiSpeed Rules dialog box, and click **OK**.

Creating a Differential Pair of Associated Nets

Procedure

1. **Setup** menu > **Design Rules** > **Differential Pairs** button
2. In the Differential Pairs dialog box, select the **Associated Nets** tab.
3. Select the associated nets to make up the diff pair:
   a. Select the first associated net from the Available list, and click the top Select button.
   b. Select the second associated net from the Available list, and click the bottom Select button.
4. Click **Add**.
5. Set the properties of the diff pair as appropriate, and click **OK**.

Creating Associated Net Design Rules

Use the Associated Net Rules dialog box to set or modify associated nets design rules.

Tip: By default, associated nets have no rules (that is, the default is no rules). If you do not specify rules for an associated net, it has none.

Procedure

1. You can assign rules to one or more associated nets at a time. There are two ways to select the associated net(s) before accessing the rule categories.

    - Select associated nets in the design:

        i. In the design area, select one of the nets in the associated net to which you want to apply rules.

        Tip: You can also use the Find dialog box to find nets and use wildcards in the Value field to filter the selection.

        ii. Right-click and click **Select Associated Net**.

        iii. Right-click and click **Show Rules**.

    - Select associated nets from the list in the Associated Net Rules dialog box:

        iv. **Setup** menu > **Design Rules** > **Associated Nets** button.

        v. In the Associated Net Rules dialog box, in the Associated Nets box, click one or more associated nets.

    Tip: To display all associated nets, clear the Show Associated Nets with Rules check box.

2. Click the **HiSpeed** button, and specify Length and/or Matching rules for the selected associated net(s).

Result

After you define rules, , HiSpeed Rules indicators (H) appear beside the selected associated nets in the Associated Nets list.

Modifying Associated Net Design Rules

Follow the procedure in the topic Creating Associated Net Design Rules.

Clearing Associated Net Rules

You can clear the rules for selected associated nets.

Tip: By default, associated nets have no rules (that is, the default is no rules). If you do not specify rules for an associated net, it has none.

Procedure

1. **Setup** menu > **Design Rules** > **Associated Nets** button.
2. In the Associated Net Rules dialog box, select one or more associated nets in the associated Nets list, click **Default**, and then click **Yes**.

   Tip: The Default button is unavailable if the selected associated net has no rules assigned to it.

   Result: You know you've successfully reset the net rules if the HiSpeed Rules indicator (H) is removed, and the Default button is no longer available.
3. Close all rules dialog boxes to return to the design.

Conditions Governing Associated Net Creation

Associated net creation is controlled by the following dialog box settings:

- The *Associate net* and *Allow net association by components* checkboxes in the Net Properties dialog box.
- The *Associate nets* and *Allow net association by refdes prefix or by net* checkboxes in the Component Properties dialog box.
- Refdes prefixes you specify in the Associated Nets dialog box.
- The *Maximum net count per associated net* and *Maximum non-plane-net pin count* fields in the Associated Nets dialog box.

The default settings for these checkboxes are:

- The *Associate nets* checkboxes, which create associated nets, are **cleared** by default for all nets and components. So by default, no nets are associated.
- The *Allow net association...* checkboxes, which allow the selected net or component to be associated by other methods, are **set** by default for all nets and components. So by default, you can associate any net or component by any of the three methods.
- The *Maximum net count per associated net* default value is 5.
- The *Maximum non-plane-net pin count* default value is 25.

A discrete component and its attached nets become part of an associated net only if the following conditions 1 and 2 are true:

1. The component and net properties are set so as to meet one of the following conditions:

- COMPONENT *Associate nets* is set **and** neither attached net is excluded from net association (A component or net is excluded from net association if its *Associate nets* and *Allow net association...* checkboxes are both cleared.)
- NET *Associate net* is set for both nets **and** the component is not excluded from net association (A component or net is excluded from net association if its *Associate nets* and *Allow net association...* checkboxes are both cleared.)
- COMPONENT *Allow net association...*is set **and** no attached net is excluded from net association **and** the component's refdes prefix is specified in the Associated Nets dialog box. (A component or net is excluded from net association if its *Associate nets* and *Allow net association...* checkboxes are both cleared.)

2. The component, its nets, and the potential associated net it will be a part of, conform to the General Restrictions for creating associated nets.

Chapter 18
Setting Colors

In this chapter:

Setting Colors

There are four different ways to set colors in PADS Layout: In the Design Editor, to individual nets, in the Decal Editor, and when you want to produce a CAM document.

Restriction: You can change colors in the Decal Editor; however, the colors remain for the duration of your editing session with the decal only. When you change the decal, it reverts back to the defaults.

- **Design**—Use the Display Colors Setup dialog box to set colors to objects on each layer.
- **Nets**—Use the View Nets dialog box to set colors to individual nets.
- **Decal**—Use the Display Colors Setup dialog box within the Decal Editor.
  Important: Once you bring the decal into a design, the decal assumes the design colors.
- **CAM**—Use the Select Items dialog box to set colors to objects in CAM before printing the CAM document.
  Restriction: You must have a color printer for the colors to appear in the Select Items dialog box.

Setting Colors of Objects in the Display

As you work on a layout design, you can customize display colors to make it easier to see objects as you place them. Using the Display Colors Setup dialog box, you can:

- Set and change the color of objects on a per layer or per object type basis.
- Make objects visible or invisible in the display (also on a per layer or per object type basis).
- Customize the palette of color selections.
- Save your customizations in a configuration file.

Procedure

1. **Setup** menu > **Display Colors**.
2. Set the color(s) you want:

    a. To set the color for one object, in the **Selected Color** area, click a color tile. Then in the Layers/Object Types table, click the tile for the object type in the correct layer row.
    Tip: To change the palette of colors from which you can select, click **Palette**.
    See also: To Change the Color Palette

    b. **To make all objects on a layer the same color,** click the layer number to select the entire row, and then click the color tile in the Selected Color area.

    c. **To make an object the same color on all layers,** click the object name to select the entire column, and then click the color tile in the Selected Color area.

    d. **To make an object type invisible,** set its tile to the background color. You can make multiple objects invisible (for example, all objects on the same layer).
    See also: Making Objects Invisible in the Display.

    Tip: To make an object type the same color on all layers, use the Assign Color to All Layers dialog box.

3. Click **Apply**.

Result: The display reflects your color settings. When you save the design, PADS Layout stores the settings with design data.

Tips:

- You can save display color settings you commonly use as you work on a design. See To Save Color Assignments to a File.
- You can use modeless commands to make pin numbers and net names visible or invisible. See Making Pin Numbers and Net names Visible or Invisible.

Color configurations saved in PowerPCB versions 2.1 and lower cannot be used in PowerPCB versions 3.0 and higher.

Related Topics

To Change the Color Palette

To Save Color Assignments to a File

Display Colors Setup Dialog Box

To Change the Color Palette

You can change the colors that appear in the color palette.

Procedure

1. **Setup** menu > **Display Colors**.
2. In the Selected Color area, click the color tile that you want to change.
3. Click **Palette**.
4. In the Color dialog box, click the color value you want to use.

   Tip: You can also create a custom color. Refer to the Windows Help for more information on changing the Windows color palette.
5. Click **OK** to return to the Display Colors Setup dialog box.

Result

The new color appears in the tile you selected.

Related Topics

Setting Colors of Objects in the Display

Creating a New Default Decal Editing Environment

To Save Color Assignments to a File

Display Colors Setup Dialog Box

Making All Objects Visible

To make objects visible:

1. **Setup** menu > **Display Colors**.
2. Click **Assign All** to open the Assign Color to All Layers dialog box.

3. Click **Automatically Make Objects Visible**.
4. Select one of the following:
   - **One Color Per Object Type** assigns color for a certain object type, which is currently set to the background color, on all layers,
   - **One Color for a Layer** assigns color for all objects, which are currently set to the background color, on the same layer,
   - **Selected Color** assigns a color you select from the Display Colors Setup dialog box color matrix to all objects, which are currently assigned the background color, on all layers.
5. Click **OK**. You return to the Display Colors Setup dialog box.

You can also make objects invisible. See Making Objects Invisible in the Display.

Making Objects Invisible in the Display

When you are placing or routing components in a layout design, it can be helpful to have certain objects visible in the display and others invisible. For example, you might want to make objects on the power, ground, and bottom layers invisible when you are routing on the top layer.

To make all objects invisible:

1. **Setup** menu > **Display Colors**.
2. Use the method that matches what you want to do.
   - **To make one object type invisible,** set its tile to the background color.

     Tip: When you are working in the design, you can make pin numbers or net names invisible (or visible again) using modeless commands. See Making Pin Numbers and Net names Visible or Invisible.
   - **To make all objects on a layer invisible,** clear the check box to the right of the layer name. To make all objects on a layer visible, select the check box.
   - **To make an object type invisible on all layers,** clear the check box above the column for that object type. To make an object type visible on all layers, select the check box.
   - **To make all objects invisible,** change their colors on all layers to the background color. You might use this option as a quick way to clear the display of objects except for board outline and connections, whose colors are set in the Other area.

     Click **Assign All**. On the Assign Colors to All Layers dialog box, click **Assign Background Color to All Objects**.

Tip: Make sure you save your color configuration so you can revert to the design colors you were using.

- To list only the layers with visible objects, select the **Visible only** check box.

3. Click **Apply**.

Making Pin Numbers and Net names Visible or Invisible

While working in a layout design, you can use the following modeless commands to turn display of net names and pin numbers on or off:

PN
Toggles pin number display on/off.
Tip: This command toggles the Pin Num. column check box in the Display Colors Setup Dialog Box.

NN
Enables/disables the settings of the NNP, NNT, and NNV commands. That is:
- When net name display is turned off with the NN command, no net names are displayed, irrespective of the NNP, NNT and NNV settings.
- When net name display is turned on with the NN command, net names are displayed or not displayed, according to the NNP, NNT and NNV settings.

Tip: This command toggles the Pin Num. check box in the Display Colors Setup Dialog Box.

NNP
Toggles display of net names on pins on/off.
Tip: This command toggles the Net names on...Pins check box in the Display Colors Setup Dialog Box.

NNT
Toggles display of net names on traces on/off.
Tip: This command toggles the Net names on...Traces check box in the Display Colors Setup Dialog Box.

NNV
Toggles display of net names on vias on/off.
Tip: This command toggles the Net names on...Vias check box in the Display Colors Setup Dialog Box.

Related Topics

Typing Modeless Commands

Setting Pin Number and Net Name Display Options

Setting Pin Number and Net Name Display Options

Use the Display page of the Options dialog box to set the text size of displayed net names and pin numbers, and the maximum allowable gap between net names on traces.

To Assign Colors to Objects on Different Layers

You can assign various colors to different objects on different layers. You can assign colors for pads and component pins, pin numbers, net names, traces, vias, lines (nonelectrical drawn items), text, copper and copper pour, errors, labels (reference designator, part type, and attribute labels), keepouts, and outlines (top and bottom).

Procedure

1. **Setup** menu > **Display Colors**.
2. Click a color from the **Selected Color** area.
3. Click a tile in an item column and a layer row. The color for that object changes for that layer.
4. To make all objects on a layer invisible, click to clear the check box to the right of the layer name. Click the check box to view the objects.
5. To make an object type invisible on all layers, click to clear the check box on top of the column for the object type. Click the check box to view the objects.

Tip: You can't set up colors per attribute type, only by layer.

Related Topics

To Change the Color Palette

To Save Color Assignments to a File

Setting Colors of Objects in the Display

Display Colors Setup Dialog Box

To Save Color Assignments to a File

You can save color assignments you have specified in the Display Colors Setup dialog box to a file.

Procedure

1. **Setup** menu > **Display Colors**.

2. Set the color assignments you want.
3. Click **Save** in the **Configuration** area.
4. Type the configuration name in the Save Configuration dialog box. The current color settings are saved under the new name. Configurations are saved in the C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Settings folder as the saved name with a .ccf extension. To change the default display colors setup, name the color configuration default.ccf. The Save Configuration dialog box appears.

Warning: You are not prompted if you overwrite an existing color configuration.

To use a previously saved configuration, select the configuration name from the Configuration list.

To save the current color settings under an existing configuration, select the configuration name from the configuration list and click **Save**. Name the new configuration with the old file name. The file is overwritten.

To delete a configuration, select the configuration in the list and click **Delete**.

Related Topics

Setting Colors of Objects in the Display

Display Colors Setup Dialog Box

Making Permanent Changes to the Display Colors

The Display Colors of the Design Editor and the PCB Decal Editor only change for the current Design or Decal. To make permanent changes to the colors, you must create a new default start-up file for the Design Editor or the PCB Decal Editor.

For the Design Editor, see Customizing PADS Layout Default Settings

For the PCB Decal Editor, see Creating a New Default Decal Editing Environment

Chapter 19 Setting Options

In this chapter:

Creating a Backup File

PADS Layout automatically backs up your files based on the settings you choose.

Procedure

1. **Tools** menu > **Options** > **Global** > Backups page.
2. In the Interval box, type the time in minutes between backups.
3. In the Number of Backups box, type the quantity (1-9) of different backup files to create.

   Tip: Backup files are named <design_name>.#, where # is a sequential number. For example, Layout1.pcb, Layout2.pcb, and so on.
4. If you want to change the folder or name of the backup file, click **Backup File**.

   Result: The Backup File dialog box appears. Browse to the folder, type the file name, and then click Save.
5. Select the **Use design name in backup file name** check box to use the design name instead of the product name as the file name.

   Example: preview_Layout1.pcb, preview_Layout2.pcb instead of layout1.pcb, layout2.pcb.
6. Select the **Create backup files in design directory** check box to place all of your backup files in the same directory as the design.

   Tip: Click to clear if you want your backup files in one, common backup directory.

Results

Table 19-1. Backup File Creation

If this is selected:		**The Backup File is Saved**
Create backup files in design directory	Use design Name in backup file name	
		in one common directory without the design name.
X	X	in the design directory using the design name.
X		in the design directory without the design name.
	X	in one common directory using the design name.

To Set the Display Grid

Procedure

1. **Tools** menu > **Options** > **Grids tab**.
2. Use the Grids page of the Options dialog box to set the display grid.

Tip: The fastest and most convenient way to change a grid is to use the Modeless Command; type **GD <x> <y>** and press **Enter**.

To Set the Design Grid

To set or reset the design grid:

- Use the Modeless Command; type **GR <x> <y>** and press **Enter**.

Tip: You can also change the design grid and the via grid by typing **G <x> <y>** and pressing **Enter**.

DRC and the Via Stitching and Shielding Operations

The Design Rule Checking (DRC) setting determines whether the Via Stitch and Add Via Shield operations can add vias. The DRC setting works with these operations in the following ways:

- Even when the DRC setting is Off, the Via Stitch and Add Via Shield operations do not add vias that violate clearance rules for pins, coppers, keepouts, texts, and board outline. However, the operations may create violations with traces if the DRC setting is Off.

- If the DRC setting is Prevent Errors, DRC prevents the via stitching and shielding operations from placing any via that violates design rules, including traces.
- If the DRC setting is Warn Errors, when the stitching and shielding operations place vias that violate design rules, DRC displays the warning “Clearance violations detected. Permit?”

Related Topics

See the On-line DRC setting on the Design page of the Options dialog Box.

Setting Via Shielding Options

Use options in the When Shielding area of the Via Patterns tab to specify the via type for vias used for shielding.

Requirement: You must have a design open in order to set Via Patterns options.

Tip: See for a description of each option, see the Via Patterns page.

Procedure

1. **Tools menu** > **Options** > **Via Patterns**.
2. In the When Shielding area, from the **Add vias from net:** list, select the net you want to associate with the vias used in shielding. Then select a Via type. (Design rules determine available via types.)
3. In the Shielding spacing area, specify the distance from the shielding vias to the trace or shape they shield.

   Select **Use design rules** to have design rules determine the distance or select either **Via to edge** or **Via to ground edge** and specify a value. (For information on these options, see the Via Patterns page.)
4. In the Via Spacing box, specify a distance between vias (center to center).

   Note: The value you specify applies to both Via Stitch and Add Via Shield operations.
5. To glue each via added by the Add Via Shield or Via Stitch operation, select **Glue vias as they are added**.
6. Make sure that the **Ignore Via Grid** check box is selected; otherwise vias snap to the grid.
7. Click **OK**.

Tips:

- Set Via Patterns options before using **Add Via Shield** or **Via Stitch** commands.

- PADS Layout stores your settings in the design for future operations; you need to set them again only if you want to change the settings.
- The Add Via Shield and Via Stitch operations do not move traces in order to make room for the vias. For best results, you should add via shielding or via stitching early in design layout when your design has fewer traces.

Related Topics

Adding a Via Shield

Setting Shape Stitching Options

Use options in the When stitching shapes area of the Via Patterns tab to specify the via type for stitching a copper shape for a given net.

Requirement: You must have a design open in order to set Via Patterns options.

Procedure

1. **Tools menu** > **Options** > **Via Patterns tab**.
2. In the When stitching shapes area, click **Add** to add a new line to the Nets/Via Type list.
3. Select a net for stitching and then select a via type.

   Tip: Design rules determine available via types.
4. In the Pattern area, select the mode for placing vias in a shape.
   - Fill—Choose the Aligned or Staggered pattern.
   - Perimeter

   Result: The Preview area displays your selections.
5. In the Via to shape box, specify the distance from the edge of the shape to the edge of the via pattern used in stitching the shape.
6. Select the **Fill selected hatch outline** option to fill only a selected hatch outline or clear the check box to fill all hatch outlines in the shape.
7. In the Via Spacing box, specify a distance between vias (center to center).

   Note: The value you specify applies to both Via Stitch and Add Via Shield operations.
8. To glue each via added by the Via Stitch or Add Via Shield operation, select **Glue vias as they are added**.
9. To have Via Stitch and Add Via Shield operations ignore the via grid setting, select the **Ignore Via Grid** check box. To have vias snap to the grid, clear the check box.
10. Click **OK**.

Tips:

- Set Via Patterns options before using **Add Via Shield** or **Via Stitch** commands.
- PADS Layout stores your settings in the design for future operations; you need to set them again only if you want to change the settings.
- The Add Via Shield and Via Stitch operations doe not move traces in order to make room for the vias. For best results, you should add via shielding or via stitching early in design layout when your design has fewer traces.

Related Topics

Filling a Shape with a Pattern of Vias

Placing Vias Inside the Perimeter of a Shape

Chapter 20
Controlling Attributes

In this chapter:

Using the Attribute Dictionary

Although you can add new attributes to design objects using the Object Attributes dialog box (select object > right-click > Attribute), you must use the Attribute Dictionary to set the properties for attribute values. It is recommended that you use the Attribute Dictionary to create new attributes for, or to edit and delete attributes in, your design. You can also use the Attribute Dictionary to assign attributes for the design, or remove attributes from objects.

In this topic:

Creating Attributes for the Design

PADS Layout provides default attributes that are applied to every new design you create. Although the attributes are provided, they are not assigned to any objects.

See also: Default Attributes in the *Concepts Guide*

To create an attribute:

1. **Edit** menu > **Attribute Dictionary**.
2. Click **New**.
3. Add the attribute and set up the attribute properties following the steps in Setting Attribute Properties.
4. When you finish setting attribute properties, click **OK** to return to the Attribute Dictionary dialog box.
5. Repeat Steps 2 to 4 for each new attribute you want to create.
6. Click **Close**.

Modifying Design Attribute Properties

Use the Attribute Dictionary to modify design attribute properties.

Requirement: If an attribute is ECO-Registered, you must be in ECO mode to modify the attribute.

1. **Edit** menu > **Attribute Dictionary**.
2. You can select an attribute group from the Group list. This list acts as a filter and allows you to view only a select group of attributes. Attributes are grouped if you they are structured attributes.

   Tip: If you select the **Show Hidden** check box, you can view attribute groups that have no visible attributes. You set whether an attribute is hidden on the Objects tab of the Attribute Properties dialog box.
3. Click the attribute to modify from the list. Default attributes and the design attributes are listed.

   Tip: You can modify the default attributes; however, it is not recommended.
4. Click **Properties**. The Attribute Properties dialog box appears.
5. Modify the attribute properties following the steps in Setting Attribute Properties.
6. When you finish setting attribute properties, click **OK** to return to the Attribute Dictionary dialog box.

7. Repeat Steps 2 to 5 for each attribute you want to modify.
8. Click **Close**.

Deleting Design Attributes

Use the Attribute Dictionary to delete attributes in your design. If you delete a decal attribute, any labels associated with it are now associated with non-decal attributes.

1. **Edit** menu > **Attribute Dictionary**.
2. You can select an attribute group from the Group list. This list acts as a filter and allows you to view only a select group of attributes. Attributes are grouped if you they are structured attributes.
3. Clear the **Show Hidden** check box. Only attribute groups that contain at least one visible attribute appear in the list.
4. Click the attribute to delete from the list.

   Tip: You can delete the default attributes; however, it is not recommended. Because the default attributes are only provided for your design and not assigned to objects, you do not need to delete these attributes.
5. Click **Delete**. The message “Are you sure you want to delete attribute type XXX?” appears.

   Exception: If you select a hidden attribute, this button is unavailable.
6. Click **Yes** to delete the attribute.

Working with Pre-3.0 Designs

Designs created in PADS software before version 3.0 will not list any provided Default Attributes.

- In the Update from lib. area, click **Load Now** to automatically load attributes for part types and decals from the current libraries to the Attribute Dictionary.
- Click **AutoLoad for pre-3.0 Designs**, and then open (load) a design file; the attributes are updated as soon as the file loads.

Setting Attribute Properties

Use the Attribute Properties dialog box to set the properties of attribute values. Properties are assigned to attributes, not to objects; therefore attributes cannot have properties that change depending on the object. Attribute properties include the type of value you want the attribute to have, whether values are case sensitive, and the objects to which you want to apply the attribute.

If the attribute is a system attribute, all of the options in this dialog box are unavailable, except the System attribute option on the Objects tab (which you would use to turn off the system attribute flag - not recommended).

In this topic:

Adding a New Attribute

1. **Edit** menu > **Attribute Dictionary** > click **New**.
2. Type the name of the new attribute to create in the **Attribute** box.

   Tips: Attributes names can be 255 characters long. You can use any printable character, including spaces, in an attribute name. A space, however, cannot be the first or last character, or the character after a dot in the attribute name (for example, xxx. xxx is illegal). Attribute names are not case sensitive, and they are defined for the entire design, not per object.
3. On the Type tab, assign a type to the attribute and assign settings to the type if applicable. The default type is **Free Text**. Choose from:

   Free Text—Allows you to type a text string as the attribute value.

   Yes/No—Creates a list box with Yes and No as the attribute values.

   Number—Allows you to type a number as the attribute value.

   Decimal Number—Allows you to type a decimal number for the attribute value.

   Measure—Allows you to determine a measurement for the attribute value. It is a physical value associated with units.

   List—Allows you to create a list from which you choose the value.
4. On the Objects tab, assign settings and hierarchy to the objects to which you want to apply the attribute.
5. Click **OK** to return to the Attribute Dictionary.

Modifying Attribute Properties

Attribute properties include the type of value you want the attribute to have, whether values are case sensitive, and the objects to which you want to apply the attribute. Properties are assigned to attributes, not to objects; therefore, attributes cannot have properties that change depending on the object.

To set attribute properties:

1. **Edit** menu > **Attribute Dictionary**.
2. Click an attribute for which to set properties from the list.
3. Click the **Properties** button. The Attribute Properties dialog box appears.
4. Click the **Types** tab, and click the type for the attribute. Choose from:

   Free Text—Allows you to type a text string as the attribute value.

   Yes/No—Creates a list box with Yes and No as the attribute values.

   Number—Allows you to type a number as the attribute value.

   Decimal Number—Allows you to type a decimal number for the attribute value.

   Measure—Allows you to determine a measurement for the attribute value. It is a physical value associated with units.

   List—Allows you to create a list from which you choose the value.

   If you use the design unit Mil, assign the attribute at the board level. Otherwise the unit will not change when you change units for the design (using the Global / General page of the Options dialog box).
5. Click the Objects tab, and click the objects you want to restrict. You can't assign the attribute to a restricted object. For example, if you are defining the properties for a Manufacturer attribute, you may want to restrict nets so they can't have this attribute. Therefore, you would disable the Net object for the Manufacturer attribute.
6. To use the default hierarchy, click **Use Default Hierarchy**. If you choose not to use the default hierarchy, you can modify it. Modifying the hierarchy is considered an advanced procedure.

   See also: Using the Objects Tab
7. If you want to enable ECO Registration of the attribute, click **ECO Registered**. If you enable ECO Registration, changes made to the attributes are recorded in the ECO file. Enabling ECO Registration also restricts you to modifying attributes while the ECO toolbar is active (ECO mode).
8. Click **OK** when you finish setting attribute properties to return to the Attribute Dictionary dialog box.

Using the Free Text Attribute Type

You can select the Free Text attribute type to use any text as the attribute value. This is the default. Free text is not “intelligent.” You can type the name of the net as an attribute, but the attribute will not update if you rename the net. With the Free Text type you can select the Case-sensitive parameter to preserve the letter case of Free Text entries. This setting affects sorting and matching in the Find dialog box and the Attribute Manager dialog box.

Using the Yes/No Attribute Type

You can select the Yes/No attribute type to create a list where you can select Yes or No as the attribute value.

Using Other Attribute Types

You can select other attribute types which require additional parameters. See the following topic for specific information regarding the other attribute types. **See also:** Using the List Attribute Type, Using the Measure Attribute Type, Using the Number or Decimal Number Attribute Type

Using the List Attribute Type

Use this tab to set the attribute type.

In this topic:

- Selecting the List Type
- Adding List Entries
- Deleting List Entries

Selecting the List Type

You can select the List attribute type to create a list of options for the value. For example, you can create a list of all of your part manufacturers that is used every time you assign the attribute.

Adding List Entries

You can create a list of entries as options for the attribute value.

1. Type an attribute value in the box.
2. Click **Set** to add the item to the list.
3. Repeat steps 1 and 2 as necessary.
4. You can select the Case-sensitive check box to preserves the letter case of List entries. This setting affects sorting and matching in the Find dialog box and the Attribute Manager dialog box.

Result: The List box contains the items you entered as possible values for attributes. The items in the list appear as a list from which you can click a value in the Object Attributes or Attribute Manager dialog boxes.

Deleting List Entries

You can click Clear or Clear All to delete an item or all items from the list.

Using the Measure Attribute Type

Use this tab to set the attribute type.

In this topic:

- Selecting the Measure Type
- Using a Listed Set of Units

- Adding a New Set of Units
- Setting Limits for the Measure Attribute Type
- Deleting a Set of Units

Selecting the Measure Type

You can select the measure attribute type to set measurement parameters for the attribute value. It is a physical value associated with units. You can set up the unit of measurement and set a minimum and maximum for the value. You can click a unit of measure from the predefined list or you can add a new unit to the list. You can also customize units. You can either use an existing measurement unit or add a new measurement unit to the design.

Attribute values for the Measure type properties are automatically converted during the ECO process.

Using a Listed Set of Units

All of the default units (and unit prefixes) of measurement appear in the list.

- In the Measure list, select a measurement unit. The abbreviation appears in the Abbr box, the unit in the Unit box, and the measure or quantity in the Quantity box.

Adding a New Set of Units

You can also permanently change the default units that appear in this list.

To add a new unit to the list:

1. In the **Abbr** box, type the abbreviation to use for the unit.
2. In the **Unit** box, type the name of the unit.
3. In the **Quantity** box, type the quantity, or what it measures.
4. Click **Set**.

Tip: If you use the unit **Mil**, assign the attribute at the board level. Otherwise the unit will not change when you change units for the design (using the global / General page of the Options dialog box).

Setting Limits for the Measure Attribute Type

You can specify a range for the Measure attribute type. Type in the Min and Max boxes to set the range. PADS Layout checks against the Limits area values. Therefore, if an attribute has the Number, Decimal Number, or Measure type property, PADS Layout checks the attribute value against this property. Leading zeros are removed, trailing zeros after the decimal point are removed, and numbers greater than 6 characters are rounded.

- Type a minimum value in the Min box and/or a maximum value in the Max box.

Deleting a Set of Units

You can click Clear or Clear User to delete an item or all user items from the list. Clear User deletes only user-defined units from the list. The default units remain in the list.

Related Topics

Default Units in the *Concepts Guide*

Customizing Units for Attributes in the *Concepts Guide*

Using the Number or Decimal Number Attribute Type

Use this tab to set the attribute type.

In this topic:

- Selecting the Number or Decimal Number Type
- Setting Limits for the Measure Attribute Type

Selecting the Number or Decimal Number Type

You can type an integer number (Number) or a number with decimal or floating point numbers (Decimal Number) for the attribute value. Attribute values for the Number or Decimal Number type properties are automatically converted during the ECO process.

Setting Limits for the Number or Decimal Number Type

You can specify a range for the Number or Decimal Number attribute type. Type in the Min and Max boxes to set the range. PADS Layout checks against the Limits area values. Therefore, if an attribute has the Number or Decimal Number type property, PADS Layout checks the attribute value against this property.

Number—you can type any number between -232 and 232 -1. Leading zeros are removed. Numbers with more than 6 zeros may be converted to scientific notation.

Decimal Number—you can type any number between 1.7E +/- 308. Leading zeros are removed, trailing zeros after the decimal point are removed, and numbers greater than 6 characters are rounded.

- Type a minimum value in the Min box and/or a maximum value in the Max box.

See also: Number/Decimal Number Attribute Values and ECO in the *Concepts Guide*

Using the Objects Tab

Use the Objects tab to assign the attribute to objects and set up the hierarchy for the attribute.

If the attribute is a system attribute, all of the options in this dialog box are unavailable, except the System attribute option on the Objects tab (which you would use to turn off the system attribute flag).

In this topic:

- Applying the Attribute to Objects
- Changing the Default Attribute Hierarchy
- Selecting ECO-Registered Status
- Selecting System Status
- Selecting Read-Only Status
- Selecting Hidden Status

Applying the Attribute to Objects

You can apply or restrict the attribute by selecting design objects to which you want to apply the attribute.

1. **Edit** menu > **Attribute Dictionary** > click **New** or [or select an attribute > Properties].
2. Type the name of the new attribute in the Attribute box. If you are not creating a new attribute, ensure the attribute name of the attribute you are modifying appears in the Attribute box. You can use the Browse Lib Attr button to select an attribute from the Attribute Dictionary list of library attributes.
3. On the Objects tab, in the Objects list, select the check box beside objects to which you want to make available the new attribute.

Tip: The Part object is the component in the design. The PCB object is the PCB design as a whole. If you select the Via check box, the ECO Registered check box is unavailable.

Changing the Default Attribute Hierarchy

You can change the default hierarchy of attribute values. For example, an attribute value can be assigned to a decal, part type, and to the component in the design. A component level value overrides a part type value which in turn, overrides a decal level value. If you assign attributes to multiple levels and then delete an attribute, the attribute from the highest possible hierarchy level is assumed.

1. **Edit** menu > **Attribute Dictionary** > click **New** or [or select an attribute > Properties].
2. Clear the **Use Default Hierarchy** check box.

   Requirement: You must have selected enough objects in the objects list to activate a hierarchy. The order of the objects in the Objects list shows the default hierarchy.
3. Select an object in the Objects list.
4. Click the hierarchy level you want to move.
5. Click **Up** or **Down** to move the hierarchy level. The object at the top of the list has the highest priority.

Restriction: Hierarchy modification is restricted. PADS Layout automatically sets a certain logical hierarchy. For example, you cannot place Net Class above Net in the hierarchy because a net cannot usually be derived from a Net Class. PADS Layout automatically places Net above Net Class in the hierarchy.

Selecting ECO-Registered Status

Allows you to specify if the attribute is ECO registered. If so, changes to the attribute are registered in the ECO file. If this check box is clicked, you can only modify attributes when the ECO toolbar is open (ECO mode).

- **Edit** menu > **Attribute Dictionary** > click **New** or [or select an attribute > Properties].

ECO registration of attributes is not forward or backward annotated. The value of an attribute is backward annotated only if the attribute is ECO registered; turn on ECO Registered for attributes to backward annotate. When forward annotating, a report automatically appears in your default text editor indicating the ECO Registration of imported attributes. If an attribute does not exist in the Attribute Dictionary, it is added with ECO Registered turned off. If the attribute already exists in the dictionary, the existing attribute and ECO Registered status in the dictionary are used.

Exceptions:

- If you click the **Via object** check box only, the ECO Registered check box is unavailable.
- Attributes modified using Automation are never registered in the ECO file, regardless of this setting.

Selecting System Status

Shows whether the attribute is a system attribute. System attributes are used by PADS Layout, an external program, or an Automation script (such as Sax Basic). The System check box prevents you from modifying an attribute that is internally set by, and critical to, PADS Layout operation.

- **Edit** menu > **Attribute Dictionary** > click **New** or [or select an attribute > Properties].

Exceptions: Automation ignores this setting and can change a system attribute. This setting is also ignored in the PCB Decal Editor and the library. This setting does not effect opening .pcb files, importing .asc files, importing .dxf files, importing .eco files, or interactive ECO operations (such as Add Part from Library, Update Part from Library, or Change Part).

The System check box is automatically selected if an attribute requires a specific type for processing. You can also turn on the **System** check box to prevent accidental modification of an Attribute Dictionary entry, which may be useful if external programs use the attribute.

You can modify the value of a system attribute. You cannot modify the Attribute Dictionary entry of a system attribute. For new attributes, the System check box is turned off by default.

Warning: Do not modify the properties or Attribute Dictionary entry of a system attribute. Severe program or script errors can occur.

Selecting Read-Only Status

Shows whether the attribute value is read-only, which means it cannot be changed outside of the library. However, you can modify the attribute properties. If you want to modify the attribute value, do so in the library.

- **Edit** menu > **Attribute Dictionary** > click **New** or [or select an attribute > Properties].

Exceptions: Automation ignores this setting and can change a read-only attribute. This setting is also ignored in the PCB Decal Editor and the library. This setting does not effect opening .pcb files, importing .asc files, importing .dxf files, importing .eco files, or interactive ECO operations (such as Add Part from Library, Update Part from Library, or Change Part).

If you are responsible for setting attribute values, you may find the Read-Only status useful because it prevents other users from changing a value. The Read-Only check box can also protect part and decal library attribute data from modification.

For new attributes, this option is cleared by default. The Read-Only check box is unavailable if the **Hidden** check box is clicked because you cannot edit the value of a hidden attribute.

Selecting Hidden Status

Hides the attribute, so that it is not visible or editable. It will not appear in any dialog boxes.

- **Edit** menu > **Attribute Dictionary** > click **New** or [or select an attribute > Properties].

Hidden attributes may be useful in the design, in a schematic program, or to an external program or script (such as Automation, ASCII, or ECO).

Exceptions: Automation ignores this setting and can change a hidden attribute. This setting is also ignored in the PCB Decal Editor and the library. This setting does not effect opening .pcb files, importing .asc files, importing .dxf files, importing .eco files, or interactive ECO operations (such as Add Part from Library, Update Part from Library, or Change Part).

See also: Attribute Hierarchy in the *Concepts Guide*

Using the Attribute Manager

Use the Attribute Manager to view all of the attributes on all objects in the design. The Attribute Manager provides a spreadsheet view of all the attributes in the design. You can use the Attribute Manager to add, edit, and delete attribute values on multiple object types. You can also create value summaries of an attribute that is based on every value of the attribute assigned to objects of the same type. In other words, summaries are applied per attribute and apply to all objects on the same tab.

Exceptions: Rows in the multi-column list are unavailable if the attribute is read-only. Rows are also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode. Hidden attributes do not appear in the list. All objects appear in the multi-column list, regardless of whether they have attributes assigned to them. Objects that do not have attributes assigned to them have <none> in the cell under the attribute name. Objects that have attributes assigned to them, but not values, have blank cells under the attribute name.

Recommendation: You can avoid DRC violations or shorts by placing attributes on Documentation layers if used in the design. If an attribute is displayed in the design, and placed on an Electrical layer it will show up as copper in the manufacturing documents. Place free text and attribute values on the Silkscreen Top layer or another documentation layer.

In this topic:

- Listing Design Objects
- Listing Object Attributes
- Interpreting the Hierarchy Level
- Sorting Column Cells
- Adding Attribute Values
- Modifying Attribute Values Using the Attribute Manager
- Deleting Attribute Values

- Applying an Attribute Value to All Other Objects
- Enabling Attribute Summaries
- Changing Summary Types

Listing Design Objects

The multi-column list of the Attribute Manager catalogs design objects and the attributes assigned to them. Objects appear in the left column and their attribute names appear in the column headers.

1. **Edit** menu > **Attribute Manager**.
2. In the View area, click a view option:

   Selected—Lists attributes from the objects selected in the design. If you select objects in the design and then click **Attribute Manager** from the Edit menu, selected objects appear on their appropriate tab. If you do not select objects of a certain type, the tab is unavailable. To view other objects, use the Filter.

   Filter—Lists attributes from all of the objects in your design. This view option allows you to view attributes assigned to non-selectable objects, such as Decals and Part Types. When using the filter, type the first characters of the objects to view, type an asterisk after the character, and then click **Apply Filter**. The objects appear on the appropriate tab in the multi-column list.

Listing Object Attributes

The multi-column list of the Attribute Manager catalogs design objects and the attributes assigned to them. Objects appear in the left column and their attribute names appear in the column headers. The rows show the attribute setting for each specific object. Use the Show and Hide buttons to control which attributes appear for design objects.

Showing Attributes

You can select which attributes you want to view in the multi-column list.

To show attribute columns:

1. **Edit** menu > **Attribute Manager** > click **Show**.
2. Use the Show Attributes dialog box to select which attributes you want to view in the list.
3. Click **OK**.

Hiding Attributes

You can remove a selected attribute from view in the multi-column list.

To hide attributes in the Attribute Manager dialog box:

1. **Edit** menu > **Attribute Manager**.
2. Select a cell in the column of the attribute you no longer want to view.
3. Click **Hide**. The attribute is no longer displayed.

Interpreting the Hierarchy Level

You can view the source of the attribute value in the Attribute Manager. For example, an attribute value can be assigned to a decal, part type, and to the component in the design. A component level value overrides a part type value which in turn, overrides a decal level value.

1. **Edit** menu > **Attribute Manager**.
2. Select a cell in the multi-column list with an attribute value and view the Hierarchy Level.

Tip: If you assign attributes to multiple levels and then delete an attribute, the attribute from the highest possible hierarchy level is assumed.

See also: Attribute Hierarchy in the *Concepts Guide*

Sorting Column Cells

You can sort the rows of the Attribute Manager.

1. **Edit** menu > **Attribute Manager**.
2. Click on the column header (name of the attribute) to sort the column in ascending order. Click on it again to sort in descending order.

Adding Attribute Values

In the Attribute Manager, you can add an attribute and value to an object that does not already have the attribute assigned to it. You can add values to cells with <none> in it, which means the attribute is available to the design but is not assigned to the object.

Restrictions: You cannot add a hidden attribute, read-only attribute, or ECO-registered attribute while not in ECO mode.

1. **Edit** menu > **Attribute Manager**.

2. Select a cell with <none> in it.
3. Click **Add**.
4. Type a value in the cell and press **Enter**.

Tips: You can only add attribute values to a design object for attributes that are available to the design. To create a new attribute for use with design objects, see Creating Attributes for the Design. You can include a unit with the value. PADS Layout also provides a default set of units (and unit prefixes) that are accepted as input and used as output. **See also:** Default Units in the *Concepts Guide*

Modifying Attribute Values Using the Attribute Manager

In the Attribute Manager, you can edit blank attribute value cells or cells with a value. If a cell is blank, it means that the attribute is assigned to the object, but has no value.

Restrictions: You cannot edit a read-only attribute, or an ECO-registered attribute while not in ECO mode.

1. **Edit** menu > **Attribute Manager**.
2. Select a blank cell or a cell with a value.
3. Click **Edit**.
4. Type a value in the cell and press **Enter**.

Tip: You can include a unit with the value. PADS Layout also provides a default set of units (and unit prefixes) that are accepted as input and used as output. **See also:** Default Units in the *Concepts Guide*

Deleting Attribute Values

In the Attribute Manager, you can delete an attribute and value from a design object.

Restrictions: You cannot delete a read-only attribute, or an ECO-registered attribute while not in ECO mode.

1. **Edit** menu > **Attribute Manager**.
2. Select a cell with a value.
3. Click **Delete**. The message “Are you sure you want to delete attribute value <attribute name>: <attribute value>?” appears.
4. Click **Yes** to delete the value.

Result: The attribute and value are removed from the object and the cell value is <none>.

Applying an Attribute Value to All Other Objects

In the Attribute Manager, you can apply an attribute value from one object to all other objects of the same type. In other words, you can set the same attribute value for all objects of the same type. Any existing values are changed.

1. **Edit** menu > **Attribute Manager**.
2. Click the cell whose attribute value you want to apply to the other objects on the tab.
3. Click **Fill Column**. The “Are you sure you want to fill <name of attribute> for all objects by <selected value>?” message appears.
4. Click **Yes** to apply the value. The attribute and its value are applied to the other cells in that column, except for Summary cells.

Enabling Attribute Summaries

In the Attribute Manager, you can create and modify summaries for attributes that are based on every value of the attribute assigned to objects of the same type. In other words, summaries are applied per attribute and apply to all objects in the same tab of the multi-column list. You can also change the summary type.

1. **Edit** menu > **Attribute Manager** > click **Show**.
2. Follow the procedures in Show Attributes to enable a summary.

Summaries appear in the last rows of the multicolumn list, and they are only available for **Number**, **Decimal Number**, and **Measure** attribute types. See Setting Attribute Properties for information on attribute properties.

Once you create one summary, it is easy to create others for other attributes without using the Show Attributes dialog box.

1. In another qualified attribute column, double-click in the cell of the first summary row (the summary title).
2. Click a summary from the list, and press **Enter**. The new summary information appears.

Changing Summary Types

In the Attribute Manager, you can change the type of summary listed at the bottom of an attribute column without using the Show Attributes dialog box.

1. **Edit** menu > **Attribute Manager**.
2. Double-click in the cell of the summary you want to change. You can only modify the first of the two summary rows.

3. Click a different summary type from the list, and press **Enter**. The new summary information appears.

Related Topics

Using Attribute Values in the *Concepts Guide*

Default Units in the *Concepts Guide*

Showing Attributes in the Attribute Manager

Use the Show Attributes dialog box to choose which attributes to view in the Attribute Manager dialog box and to choose which summaries to view, if any.

Restriction: Hidden attributes are not available to the Attribute Manager and are not listed in the Show Attributes dialog box.

See also: Using the Attribute Manager

In this topic:

- Selecting Attributes to List in the Attribute Manager
- Creating a Summary

Selecting Attributes to List in the Attribute Manager

Use the Show Attributes dialog box to select attributes to list in the multi-column list of the Attribute Manager.

1. **Edit** menu > **Attribute Manager** > click **Show**.
2. Select an attribute group in the Group list or <all> for all attributes.

   Tip: Use the Group list to filter the Attributes list. You can choose an attribute group to view.
3. In the Attributes list, select the check box next to the attribute name to view the attribute in the Attribute Manager dialog box.

Tip: You can use the **Select All** or **Unselect All** buttons to select all or clear all check boxes.

Creating a Summary

You can create summaries of every value of an attribute assigned to a particular objects type. In other words, summaries will appear at the bottom of attribute columns.

Restriction: Summaries are only available for Number, Decimal Number, and Measure attribute types. The type is an attribute property. See Setting Attribute Properties for information.

1. **Edit** menu > **Attribute Manager >** click **Show**.

   Requirement: An attribute must be selected for viewing in the Attribute Manager. See Selecting Attributes to List in the Attribute Manager for information.

2. Select an attribute in the Attributes list.

3. Click the check box for the summary type you want to create. You can select more than one summary. Choose from:

   Sum—Creates a summary that shows the total of the attribute values. The summary applies to the attribute you select in the Attributes list. The summary displays in the Attribute Manager dialog box.

   Max—Creates a summary that shows the maximum value used by the attribute. The summary applies to the attribute you select in the Attributes list box. The summary displays in the Attribute Manager dialog box. To create a summary that is the range of attribute values, click both the **Min** and **Max** check boxes.

   Min—Creates a summary that shows the minimum value used by the attribute. The summary applies to the attribute you select in the Attributes list box. The summary displays in the Attribute Manager dialog box. To create a summary that is the range of attribute values, click both the **Min** and **Max** check boxes.

   Average—Creates a summary that shows the average of the attribute values. The average is the sum of all attribute values divided by the number of values assigned. The summary applies to the attribute you select in the Attributes list. The summary displays in the Attribute Manager dialog box.

4. Click **OK** to return to the Attribute Manager dialog box. The summary you created appears in the last two rows of the multi-column list. If you enabled more than one summary, the second summary appears in the two rows following the first summary. You can modify the summary row containing the summary name; you cannot modify the summary row containing the summary calculation.

Tip: You can change summary types and copy summaries to other attributes without returning to the Show Attributes dialog box. See Enabling Attribute Summaries and Changing Summary Types.

Working with Object Attributes

Use the resizable Object Attributes dialog box to add, modify, or remove attributes of single objects or multiple objects of the same type.

Restrictions: Rows in the multicolumn list are unavailable if the attribute is read-only. Rows are also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.

Recommendation: You can avoid DRC violations or shorts by placing attributes on Documentation layers if used in the design. If an attribute is displayed in the design, and placed on an Electrical layer it will show up as copper in the manufacturing documents. Place free text and attribute values on the Silkscreen Top layer or another documentation layer.

In this topic:

- Assigning Attributes
- Modifying Attribute Values
- Removing Attributes
- Removing Attribute Values

Assigning Attributes

You can assign attributes to an object or objects of the same type using the Object Attributes dialog box. For example, this means that you can select multiple parts and assign attributes, but you cannot select parts and vias and assign attributes. The Object Attribute dialog box only shows attributes that are applicable to the objects you select.

1. Select the object(s) to which you want to assign an attribute. You can only select objects of the same type.
2. Right-click and click **Attribute**. The Object Attributes dialog box opens.

   See also: Assigning Attributes in the *Concepts Guide*
3. From the Groups list click an attribute group to view. If you assign multiple attributes and some of those are structured attributes, this list acts as a filter and allows you to choose the attribute group to view.
4. From the **Attributes For** list, select the attribute hierarchy level for which you want to assign an attribute. The hierarchy levels change, depending on the object you selected in step 1.

   See also: Attribute Hierarchy in the *Concepts Guide*

   Restriction: You cannot select a hierarchy level if you have multiple objects selected. The attribute is assigned at the current level; for example, if you select multiple parts, attributes are assigned at the Component level.
5. Click **Add**. A new, blank attribute line appears in the **Attribute** list. The pointer appears in the blank cell in the Attribute column. This cell is also a list that contains the attributes that apply to the object type you selected in step 1. This list is based on entries in the Attribute Dictionary.

See also: Using the Attribute Dictionary

6. Click an attribute from the **Attribute** list, or type the name of a new attribute in the blank cell. You cannot add a hidden or read-only attribute. Lists the name of the attribute. Attributes names can be 255 characters long. You can use any printable character, including spaces, in an attribute name. A space, however, can't be the first or last character in the attribute name. Attribute names are not case sensitive, and they are defined for the entire design, not per object.

   Tip: When you add a new attribute to the design, it is also added to the Attribute Dictionary.

7. Double-click in the blank cell in the **Value** column (next to the attribute you just added). Assign the value for the attribute. You can include a unit with the value. PADS Layout also provides a default set of units (and unit prefixes) that are accepted as input and used as output. The Level column lists the hierarchy level where the attribute is assigned. In other words, the attribute is inherited from the level that appears in this column. It takes its value from the Attributes For list.

   See also: Default Units in the *Concepts Guide*

   Tip: If you use the design unit **Mil**, assign the attribute at the board level. Otherwise the unit will not change when you change units for the design (using the Global tab of the Options dialog box).

8. Click **Close** to close the Object Attributes dialog box.

Tip: You can add the same attribute multiple times, as long as you add it to different levels of the attribute hierarchy.

See also: Creating Attributes for the Design, Control of Solder Mask and Paste Mask

Modifying Attribute Values

You can modify the attribute values of an object or objects of the same type using the Object Attributes dialog box. When you select multiple objects to modify attributes, the Attributes list displays the union of all attribute names. In other words, it lists all attributes that belong to all of the selected objects. When you add an attribute, it is added to all selected objects, and when you delete a value, it is deleted from all objects with that attribute. You can edit the value of a selected attribute if the attribute is defined on the current level. In other words, if the text in the Level column matches the text in the Attribute For list, you can edit the value. If the attribute is not defined on the current level (the text in the Level column does not match the text in the Attributes For list), then a new attribute is added for the current level that matches the attribute you want to edit. You can then edit the value in the new attribute. You cannot edit the name of an attribute. You can either delete the attribute entirely, using the Attribute Dictionary, or add a new attribute with the correct attribute name.

Procedure

1. If the attribute is ECO-registered, enter ECO mode. To enter ECO mode, click the **ECO Toolbar** button. If you don't enter ECO mode first, a message appears, informing you that you need to enter ECO mode.

   Tip: You cannot modify read-only attributes.

   See also: ECO Options Dialog Box

2. Select the object(s) to modify.

3. Right-click and click **Attribute**. Attribute information about the selected object(s) appears in spreadsheet form in the list.

4. From the Groups list click an attribute group to view. If you assign multiple attributes and some of those are structured attributes, this list acts as a filter and allows you to choose the attribute group to view.

5. From the **Attributes For** list, click the attribute hierarchy level for which you want to assign an attribute. The hierarchy levels change, depending on the object you selected in step 1.

   See also: Attribute Hierarchy in the *Concepts Guide*

   Restriction: You cannot click a hierarchy level if you have multiple objects selected. The attribute is assigned at the current level; for example, if you select multiple parts, attributes are assigned at the Component level.

6. Click the cell (in the **Value** column) of the attribute value to modify. If a cell is blank, it means that the attribute is assigned to the object, but has no value. It can also mean that the values differ for the selected objects.

7. Click **Edit**.

   Type or click the new attribute value and press **Enter**. The new value is added to the objects.

   See also: Using Attribute Values in the *Concepts Guide*

   You can include a unit with the value. PADS Layout provides a default set of units (and unit prefixes) that are accepted as input and used as output.

   See also: Default Units in the *Concepts Guide*

Tip: If attribute values are not the same, the value for that attribute is blank. If you modify that value, the new value applies to all objects with that attribute. You can also remove values from attributes (assign the attribute with no value).

See also: Removing Attribute Values

Removing Attributes

You can remove an attribute of an object or objects of the same type using the Object Attributes dialog box. You can only delete attributes on the current level. If you delete an attribute and a value is defined for attribute at a higher level in the hierarchy, it is applied to the current level. To delete the attribute entirely from the design, use the Attribute Dictionary.

1. If the attribute is ECO-registered, enter ECO mode. To enter ECO mode, click the **ECO Toolbar** button. The ECO Options dialog box appears. If you don't enter ECO mode first, a message appears, informing you that you must enter ECO mode.

   Tip: You cannot edit read-only attributes.

2. Select the object(s) to edit.
3. Right-click and click **Attribute**. The Object Attributes dialog box appears. Attribute information about the selected objects appears in spreadsheet form in the list.
4. From the Groups list click an attribute group to view. If you assign multiple attributes and some of those are structured attributes, this list acts as a filter and allows you to choose the attribute group to view.
5. Click the cell whose attribute value you want to delete.
6. Click **Delete**.

Result: The attribute is removed from the objects.

Removing Attribute Values

You can remove an attribute value of an object or objects of the same type using the Object Attributes dialog box.

1. If the attribute is ECO-registered, enter ECO mode. To enter ECO mode, click the **ECO Toolbar** button. The ECO Options dialog box appears. If you don't enter ECO mode first, a message appears, informing you that you must enter ECO mode.

   Tip: You cannot edit read-only attributes.

2. Select the object(s) to edit.
3. Right-click and click **Attribute**. The Object Attributes dialog box appears. Attribute information about the selected objects appears in spreadsheet form in the list.
4. From the Groups list click an attribute group to view. If you assign multiple attributes and some of those are structured attributes, this list acts as a filter and allows you to choose the attribute group to view.
5. Click the cell from which you want to remove values.
6. Click **Edit**.

7. Press the **Spacebar** and press **Enter**. The attribute is still assigned to the objects, but it has no value.

Modifying Default Attributes

You can edit the default attribute dictionary. You may want to change the default dictionary so it matches your library attributes. The list of default attributes is stored in two ASCII files, both stored in the *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Settings* folder.

Default.asc	Used for new designs.
DefaultAttributeDictionary.asc	Used for older (pre-version 3.0) designs. If this file is not found, an attribute dictionary is not loaded with older designs.

The appropriate ASCII file automatically imports when you create a new file or import an older file.

See also: Using the Attribute Dictionary, File Open Conversions in the *Concepts Guide*

Procedure

1. **File** menu > **New**.
2. If you want to edit the default attributes to use with older files, click **Import** from the File menu and import the file **DefaultAttributeDictionary.asc**.
3. Click **Attribute Dictionary** from the Edit menu.
4. Modify the existing attributes or add attributes as needed.

   See also: Modifying Design Attribute Properties
5. Click **OK** to close the dialog box.
6. To overwrite the existing file, click **Export** from the File menu. Do this to change the default.asc or the DefaultAttributeDictionary.asc files. If you want to create a new start-up file to use with only new files, go to Step 10.

   Tip: Back up the existing default.asc file before overwriting it.
7. Click **ASCII** as the file type and click **Save**. The ASCII Output dialog box appears.
8. Click **Attributes** in the **Sections** list.
9. Click **OK**. The default attribute dictionary is replaced.
10. To create a new start-up file, follow the steps described in Creating Start-up Files. Make sure you select the **Attributes** check box in the Start-up File Output dialog box. A new start-up file is created. You can use this start-up file with all new designs.

Customizing Units for Attributes

You can customize (enable or disable) the supported units by modifying the powerpcb.ini file in the *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs* folder.

To customize the .ini file:

1. Open the .ini file in a text editor, such as Notepad.
2. Add a new attribute unit section by typing the header **[SI Units]**.
3. Make modifications as described in the "Enabling Units" and "Disabling Units" sections below.
4. Save the .ini file.

Enabling Units

To enable units, delete the ignore; variable from the line. **Example:** The line for the Gram unit reads:

```
Gram=ignore;u,m,,k
```

Modify the line so it reads:

```
Gram=u,m,,k
```

See also: .ini File Format for Units in the *Concepts Guide*

Disabling Units

To disable units, add the ignore; variable to the line. For example, the line for the Farad unit reads:

```
Farad=p,n,u,m
```

Modify the line so it reads:

```
Farad=ignore;p,n,u,m
```

It is recommended that you leave the unit prefixes intact even when disabling the unit. This makes it easier to enable the unit later because you will not have to specify prefixes again.

See also: Customizing Units for Attributes in the *Concepts Guide*

Adding Attributes to Design Objects

Restrictions: You cannot add a hidden attribute, read-only attribute, or ECO-registered attribute while not in ECO mode.

Procedure

1. Select the object in the design.
2. **Right-click** > **Attribute**
3. In the Object Attributes dialog box, click the **Add** button.
4. Type a new attribute name, or select one from the list.
5. Enter a value.
6. Click the **Close** button.

Adding Attribute Values to Multiple Design Objects

Instead of selecting multiple objects one-at-a-time to add attribute values, you can add them more quickly using the spreadsheet in the Attribute Manager. You can add values to cells with <none> in them, which means the attribute is available to the design but is not assigned to the object.

Restrictions: You cannot add a hidden attribute, read-only attribute, or ECO-registered attribute while not in ECO mode.

Procedure

1. **Edit** menu > **Attribute Manager**.
2. In the Attribute Manager dialog box**,** select a cell with <none> in it.
3. Click **Add**.
4. Type a value in the cell and press **Enter**.

Tips: You can only add attribute values to a design object for attributes that are available to the design. To create a new attribute for use with design objects, see Creating Attributes for the Design. You can include a unit with the value. PADS Layout also provides a default set of units (and unit prefixes) that are accepted as input and used as output. **See also:** Default Units in the *Concepts Guide*

Adding Height Information to Design Components and Jumpers

Height information is used to prevent components being placed in height-constrained areas. It is also used when a design is exported to a 3-dimensional modeling application.

Tip: To quickly add height information to multiple objects, use the procedure in Adding Attribute Values to Multiple Design Objects.

Procedure

1. Select the object in the design.
2. **Right-click** > **Attribute**
3. In the Object Attributes dialog box, click the Add button.

   Restriction: Some attributes require you to be in ECO Mode.
4. Type Geometry.Height, or select it from the list.
5. Enter a value.
6. Click the Close button.

Chapter 21
Setting Rules and Using Keepouts

In this chapter:

Transferring Design Rules

Design rules are stored in schematic and design files. Design rules you set up in PADS Logic and DxDesigner are automatically transferred when you import the netlist into PADS Layout. Design rules you set up in PADS Layout are automatically available when you open the design in PADS Router.

Tip: PADS Logic supports only default, class, and net rules.

Importing and Exporting Design Rules

You can export and import design rules in order to reuse them in a future design that requires the same constraints.

- To export design rules, follow the instructions to export an ASCII file, but in the ASCII Output dialog box, select only the Rules check box.
- To import the design rules, follow the instructions to import an ASCII file.

Related Topics

Design Checking in the *Concepts Guide*

Checking Design Rules

Creating Rules for Your Design

There are many different levels of rules in the rules hierarchy. Follow the links below to procedures for creating rules at each level of the hierarchy.

Tip: The links below are in hierarchical order from lowest to highest priority. For more information, see Design Rule Hierarchy in the *Concepts Guide*.

Default Rules

- Creating Default Rules
- Creating Default Clearance-Rules for a Specific Layer (Conditional Rule)

Class Rules

- Creating Class Design Rules
- Creating Class Clearance-Rules for a Specific Layer (Conditional Rule)

Net Rules

- Creating Net Design Rules
- Creating Net Clearance-Rules for a Specific Layer (Conditional Rule)

Group Rules

- Creating Group Design Rules
- Creating Group Clearance-Rules for a Specific Layer (Conditional Rule)

Pin Pair Rules

- Creating Pin Pair Design Rules
- Creating Pin Pair Clearance-Rules for a Specific Layer (Conditional Rule)

Class Against... Rules

- Creating a Class Against Class Design Rule (Conditional Rule)

- Creating a Class Against Class Design Rule for a Specific Layer (Conditional Rule)

Net Against... Rules

- Creating a Net Against Class Design Rule (Conditional Rule)
- Creating a Net Against Class Design Rule for a Specific Layer (Conditional Rule)
- Creating a Net Against Net Design Rule (Conditional Rule)
- Creating a Net Against Net Design Rule for a Specific Layer (Conditional Rule)

Group Against... Rules

- Creating a Group Against Class Design Rule (Conditional Rule)
- Creating a Group Against Class Design Rule for a Specific Layer (Conditional Rule)
- Creating a Group Against Net Design Rule (Conditional Rule)
- Creating a Group Against Net Design Rule for a Specific Layer (Conditional Rule)
- Creating a Group Against Group Design Rule (Conditional Rule)
- Creating a Group Against Group Design Rule for a Specific Layer (Conditional Rule)

Pin Pair Against... Rules

- Creating a Pin Pair Against Class Design Rule (Conditional Rule)
- Creating a Pin Pair Against Class Design Rule for a Specific Layer (Conditional Rule)
- Creating a Pin Pair Against Net Design Rule (Conditional Rule)
- Creating a Pin Pair Against Net Design Rule for a Specific Layer (Conditional Rule)
- Creating a Pin Pair Against Group Design Rule (Conditional Rule)
- Creating a Pin Pair Against Group Design Rule for a Specific Layer (Conditional Rule)
- Creating a Pin Pair Against Pin Pair Design Rule (Conditional Rule)
- Creating a Pin Pair Against Pin Pair Design Rule for a Specific Layer (Conditional Rule)

Decal Rules

- Creating Decal Design Rules

Component Rules

- Creating Component Design Rules

Differential Pair Rules

- Creating Differential Pair Design Rules

Creating Default Rules

Each new design already has a default set of rules or constraints from the PADS Layout default.asc template file, or they are imported from the schematic design. Customize the default rules to the requirements of your design. You can also create default clearance rules for a specific layer.

Procedure

1. **Setup** menu > **Design Rules**.
2. In the Rules dialog box, click **Default**.
3. In the Default Rules dialog box, click any of the five rule category buttons (Clearance, Routing, HiSpeed, Fanout, Pad Entry) to customize the rules from their default values and settings.
4. After you've customized the rule categories, close all Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating Default Clearance-Rules for a Specific Layer

You can create a unique set of default rules on a specific layer that take precedence over the default rules which apply to all layers.

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **All**.
3. In the Against rule object area, with Layer type already selected, select the layer from the list.
4. In the Current rule set area, click **Clearance**.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.

7. In the Object to object box, type a value to apply to all objects or click the Matrix button to enter the Clearance Rules dialog box to apply different values between objects. You must click OK in the Clearance Rules dialog box to accept the changes.

8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating Class Design Rules

A class is a collection of nets to which you can assign a common set of design rules. You must first create a class and assign nets to the class before you can assign rules to the class. You create the class either by selecting nets in the design, or by selecting them from a list in the Class Rules dialog box. But you can only use the Class Rules dialog box to define its design rules.

Procedure

1. Create a new class using one of the following two methods:

   - Select nets in the design:

     i. In the design area, select the nets you want to add to the class.

        Tip: You can also use the Find dialog box to find nets and use wildcards in the Value field to filter the selection.

     ii. Right-click and click **Make Class**.

     iii. In the Add Net to Class dialog box, click **Create New Class**, type the class name in the Add to Class box, and then click **OK**.

   - Select nets from the list in the Class Rules dialog box:

     iv. On the **Setup** menu, click **Design Rules, then click the Class** button.

     v. In the Class name box, type the class name, and then click **Add**.

     vi. In the Nets area, in the Available list, double-click to quickly add the net to the class, or select multiple nets and click **Add>>**.

   Tips:

   - Nets cannot exist in more than one class. The Available list in the Class Rules dialog box displays only nets that have not been assigned to a class.

- The maximum class name length is 15 characters. You can use any alphanumeric characters except brackets {}, asterisks *, or spaces.

2. In the Class Rules dialog box (Setup menu > Design Rules > Class button), select the class in the Class box.

3. Click any of the three rule category buttons (Clearance, Routing, HiSpeed) to customize the rules from their default values and settings.

4. After you've customized the rule categories, click **OK** to accept the changes and close the Class Rules dialog box.

Result

After you customize a rule category for the class, non-default rules indicators appear in the dialog box.

Related Topics

Creating Net Design Rules

Adding Nets to an Existing Design Rule Class

Design Rule Hierarchy in the *Concepts Guide*

Deleting a Design Rule Class

Use the Class Rules dialog box to delete classes.

Procedure

1. **Setup** menu > **Design Rules** > **Class** button.

2. In the Class Rules dialog box, in the Class box, select one or more classes, and then click **Delete**.

   Tip: To display all classes in the Class box, clear the Show Classes with Rules check box.

3. Click **OK** to accept the changes and close the Class Rules dialog box.

4. Click **Close** in the Rules dialog box.

Adding Nets to an Existing Design Rule Class

You can add nets to an existing class design rule.

Procedure

1. Add nets to an existing class using one of the following two methods:
   - Select nets in the design:
     i. In the design area, select the nets you want to add to the class.

        Tip: You can also use the Find dialog box to find nets and use wildcards in the Value field to filter the selection.

     ii. Right-click and click **Make Class**.
     iii. In the Add Net to Class dialog box, click **Add to Existing Class**, and select the class in the Existing Classes list.
     iv. Click **OK**.
   - Select nets from the list in the Class Rules dialog box:
     v. **Setup** menu > **Design Rules > Class** button.
     vi. In the Class Rules dialog box, in the Class box, select the class to which you want to add a net.

        Tip: To display all classes in the Class box, clear the Show Classes with Rules check box.

     vii. In the Nets area, in the Available list, double-click to quickly add the net to the class, or **s**elect multiple nets and click **Add>>**.

   Tip: Nets cannot exist in more than one class. The Available list in the Class Rules dialog box displays only nets that have not been assigned to a class.
2. Click **OK** to accept the changes and close the Class Rules dialog box.

Removing Nets from a Design Rule Class

Use the Class Rules dialog box to remove nets from a class.

Procedure

1. **Setup** menu > **Design Rules** > **Class** button.
2. In the Class Rules dialog box, in the Class box, select the class.

   Tip: To display all classes in the Class list, clear the Show Classes with Rules check box.
3. Select the nets for removal in the Selected list, and then click **<<Remove**.

4. Click **OK** to accept the changes and close the Class Rules dialog box.
5. Click **Close** in the Rules dialog box.

Modifying Class Design Rules

You can modify the rules of a class of nets.

Procedure

1. **Setup** menu > **Design Rules** > **Class** button.
2. In the Class Rules dialog box, in the Class box, select the class.
3. Click any of the three rule category buttons (Clearance, Routing, HiSpeed) to modify the values and settings.
4. After you've customized the rule categories, click **OK** to accept the changes and close the Class Rules dialog box.
5. Click **Close** in the Rules dialog box.

Renaming a Design Rule Class

Use the Class Rules dialog box to rename a class.

Procedure

1. **Setup** menu > **Design Rules** > **Class** button.
2. In the Class Rules dialog box, in the Class box, select a class.

   Tip: To display all classes in the Class list, clear the Show Classes with Rules check box.
3. In the Class name box, type the new class name.
4. Click **Rename**.
5. Click **OK** to accept the changes and close the Class Rules dialog box.
6. Click **Close** in the Rules dialog box.

Resetting Class Rules to Default Rules

With the click of a button, you can reset the class rules to the same as the default rules.

Procedure

1. **Setup** menu > **Design Rules** > **Class** button.
2. In the Class Rules dialog box, select one or more classes in the Class list, click **Default**, and then click **Yes**.

   Tip: The Default button is unavailable if the class already has only default rules assigned to it.

   Result: You know you've successfully reset the class rules if all the icons below the rule categories (Clearance, Routing, HiSpeed) return to the Default icon, and the Default button is no longer available.
3. Click **OK** to accept the changes and close the Class Rules dialog box.
4. Click **Close** in the Rules dialog box.

Displaying the Nets of a Class Design Rule

If you want to know what nets are assigned to a class you can view them in the Class Rules dialog box.

Procedure

1. **Setup** menu > **Design Rules** > **Class** button.
2. In the Class Rules dialog box, in the Class list, click the name of the class.

   Result: Nets in the class appear in the Selected list.
3. After you've viewed the nets, click **Cancel** to close the Class Rules dialog box without saving any changes.
4. Click **Close** in the Rules dialog box.

Creating Class Clearance-Rules for a Specific Layer

You can create a unique set of class rules on a specific layer that take precedence over the class rules which apply to all layers.

Requirement

The Class of nets must already exist. **See also**: Creating Class Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Classes and then select** a class in the list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Layer** and then select a layer from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. In the Object to object box, type a value to apply to all objects or click the Matrix button to enter the Clearance Rules dialog box to apply different values between objects. (You must click OK in the Clearance Rules dialog box to accept the changes.)
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating Net Design Rules

Use the Net Rules dialog box to customize the design rules that apply to nets. Unless you customize the design rules for a net, it assumes the Default design rule values and settings.

Procedure

1. You can assign rules to one or more nets at a time. There are two ways to select the net(s) before accessing the rule categories.

   - Select nets in the design:

     i. In the design area, select the net or nets to which you want to apply rules.

        Tip: You can also use the Find dialog box to find nets and use wildcards in the Value field to filter the selection.

     ii. Right-click and click **Show Rules**.

   - Select nets from the list in the Net Rules dialog box:

     iii. **Setup** menu > **Design Rules** > **Net** button.

     iv. In the Net Rules dialog box, in the Nets box, click one or more nets.

   Tip: To display all nets, clear the Show Nets with Rules check box.

2. Click any of the three rule category buttons (Clearance, Routing, HiSpeed) to customize the rules from their default values and settings.

3. After you've customized the rule categories, close all rules dialog boxes to return to the design.

Result

After you customize a rule category for the net(s), non-default rules indicators appear in the dialog box.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Modifying Net Design Rules

Follow the procedure in the topic Creating Net Design Rules.

Resetting Net Rules to Default Rules

With the click of a button, you can reset the net rules to the same as the default rules.

Procedure

1. **Setup** menu > **Design Rules** > **Net** button.

2. In the Net Rules dialog box, select one or more nets in the Nets list, click **Default**, and then click **Yes**.

   Tip: The Default button is unavailable if the net has only default rules assigned to it.

   Result: You know you've successfully reset the net rules if all the icons below the rule categories (Clearance, Routing, HiSpeed) return to the Default icon, and the Default button is no longer available.

3. Close all rules dialog boxes to return to the design.

Creating Net Clearance-Rules for a Specific Layer

You can create a unique set of net rules on a specific layer that take precedence over net rules which apply to all layers.

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Nets and then select** a net in the list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Layer** and then select a layer from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. In the Object to object box, type a value to apply to all objects or click the Matrix button to enter the Clearance Rules dialog box to apply different values between objects. (You must click OK in the Clearance Rules dialog box to accept the changes.)
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating Group Design Rules

A group is a collection of pin pairs to which you can assign a common set of design rules. You must first create a group and assign pin pairs to the group before you can assign rules to the group. You create the group either by selecting pin pairs in the design, or by selecting them from a list in the Group Rules dialog box. But you can only use the Group Rules dialog box to define its design rules.

Procedure

1. Create a new group using one of the following two methods:
   - Select pin pairs in the design:
     i. In the design area, select the pin pairs you want to add to the group.

        Tip: You can also use the Find dialog box to find pin pairs and use wildcards in the Value field to filter the selection.

     ii. Right-click and click **Make Group**.
     iii. In the Add Pin Pairs to Group dialog box, click **Create New Group**, type the group name in the Add to Group box, and then click **OK**.
   - Select pin pairs from the list in the Group Rules dialog box:

iv. **Setup** menu > **Design Rules > Group** button.

v. In the Group Rules dialog box, in the Group name box, type the group name, and then click **Add**.

vi. In the Connections area, in the Available list, double-click to quickly add the pin pair to the group, or select multiple pin pairs and click **Add>>**. You can also filter the display of Available pin pairs to a single net by selecting the net in the From net list.

Tips:

- Pin pairs cannot exist in more than one group. The Available list in the Group Rules dialog box displays only pin pairs that have not been assigned to a group.
- The maximum group name length is 15 characters. You can use any alphanumeric characters except brackets {}, asterisks *, or spaces. These characters are automatically replaced with an underscore.

2. In the Group Rules dialog box (Setup menu > Design Rules > Group button), select the group in the Group box.
3. Click any of the three rule category buttons (Clearance, Routing, HiSpeed) to customize the rules from their default values and settings.
4. After you've customized the rule categories, click **OK** to accept the changes and close the Group Rules dialog box.

Result

After you customize a rule category for the group, non-default rules indicators appear in the dialog box.

Related Topics

Creating Pin Pair Design Rules

Adding Pin Pairs to an Existing Design Rule Group

Design Rule Hierarchy in the *Concepts Guide*

Deleting a Design Rule Group

Use the Group Rules dialog box to delete a group of pin pairs.

Procedure

1. **Setup** menu > **Design Rules** > **Group** button.

2. In the Group Rules dialog box, in the Group box, select one or more groups, and then click **Delete**. The group name is deleted from the Group box.

   Tip: To display all groups in the Group box, clear the *Show groups with rules* check box.

3. Click **OK** to accept the changes and close the Group Rules dialog box.

4. Click **Close** in the Rules dialog box.

Adding Pin Pairs to an Existing Design Rule Group

You can add pin pairs to an existing group design rule.

Procedure

1. Add pin pairs to an existing group using one of the following two methods:

   - Select pin pairs in the design:

     i. In the design area, select the pin pairs you want to add to the group.

        Tip: You can also use the Find dialog box to find pin pairs and use wildcards in the Value field to filter the selection.

     ii. Right-click and click **Make Group**.

     iii. In the Add Pin Pairs to Group dialog box, click **Add to Existing Group**, and select the group in the Existing Groups list.

     iv. Click **OK**.

   - Select pin pairs from the list in the Group Rules dialog box:

     v. **Setup** menu > **Design Rules > Group** button.

     vi. In the Group Rules dialog box, in the Group box, select the group to which you want to add a pin pair.

        Tip: To display all groups in the Group box, clear the *Show groups with rules* check box.

     vii. In the Connections area, in the Available list, double-click to quickly add the pin pair to the group, or **s**elect multiple pin pairs and click **Add>>**. You can also filter the display of Available pin pairs to a single net by selecting the net in the From net list.

   Tip: Pin Pairs cannot exist in more than one group. The Available list in the Group Rules dialog box displays only pin pairs that have not been assigned to a group.

2. Click **OK** to accept the changes and close the Group Rules dialog box.

Removing Pin Pairs from a Design Rule Group

Use the Group Rules dialog box to remove pin pairs from a group.

Procedure

1. **Setup** menu > **Design Rules** > **Group** button.
2. In the Group Rules dialog box, in the Group box, select the group.

   Tip: To display all groups in the Group list, clear the *Show groups with rules* check box.
3. Select the pin pairs for removal in the Selected list, and then click **<<Remove**.
4. Click **OK** to accept the changes and close the Group Rules dialog box.
5. Click **Close** in the Rules dialog box.

Modifying Group Design Rules

You can modify the rules of a group of pin pairs.

Procedure

1. **Setup** menu > **Design Rules** > **Group** button.
2. In the Group Rules dialog box, in the Group box, select the group.
3. Click any of the three rule category buttons (Clearance, Routing, HiSpeed) to modify the values and settings.
4. After you've customized the rule categories, click **OK** to accept the changes and close the Group Rules dialog box.
5. Click **Close** in the Rules dialog box.

Renaming a Design Rule Group

Use the Group Rules dialog box to rename a group.

Procedure

1. **Setup** menu > **Design Rules** > **Group** button.
2. In the Group Rules dialog box, in the Group box, select a group.

   Tip: To display all groups in the Group list, clear the *Show groups with rules* check box.
3. In the Group name box, type the new group name.
4. Click **Rename**.
5. Click **OK** to accept the changes and close the Group Rules dialog box.
6. Click **Close** in the Rules dialog box.

Resetting Group Rules to Default Rules

With the click of a button, you can reset the group rules to the same as the default rules.

Procedure

1. **Setup** menu > **Design Rules** > **Group** button.
2. In the Group Rules dialog box, select one or more groups in the Group list, click **Default**, and then click **Yes**.

   Tip: The Default button is unavailable if the class already has only default rules assigned to it.

   Result: You know you've successfully reset the group rules if all the icons below the rule categories (Clearance, Routing, HiSpeed) return to the Default icon, and the Default button is no longer available.
3. Click **OK** to accept the changes and close the Group Rules dialog box.
4. Click **Close** in the Rules dialog box.

Displaying the Pin Pairs of a Design Rule Group

If you want to know what pin pairs are assigned to a group you can view them in the Group Rules dialog box.

Procedure

1. **Setup** menu > **Design Rules** > **Group** button.

2. In the Group Rules dialog box, in the Group list, click the name of the group.

   Result: Pin pairs in the group appear in the Selected list.

3. After you've viewed the pin pairs, click **Cancel** to close the Group Rules dialog box without saving any changes.

4. Click **Close** in the Rules dialog box.

Creating Group Clearance-Rules for a Specific Layer

You can create a unique set of group rules on a specific layer that take precedence over the group rules which apply to all layers.

Requirement

The Group of pin pairs must already exist. **See also**: Creating Group Design Rules

Procedure

1. **Setup** menu > **Design Rules > Conditional Rules** button.

2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Groups and then select** a group in the list.

3. In the Current rule set area, click **Clearance**.

4. In the Against rule object area, click **Layer** and then select a layer from the list.

5. Click **Create**.

6. In the Existing rule sets list, if not already selected, select the newly created rule set.

7. In the Object to object box, type a value to apply to all objects or click the Matrix button to enter the Clearance Rules dialog box to apply different values between objects. (You must click OK in the Clearance Rules dialog box to accept the changes.)

8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating Pin Pair Design Rules

Use the Pin Pair Rules dialog box to create design rules for one or more pin pairs.

Procedure

1. You can assign rules to one or more pin pairs at a time. There are two ways to select the pin pair(s) before accessing the rule categories.
   - Select pin pairs in the design:
     i. In the design area, select the pin pair(s) to which you want to apply rules.

        Tip: You can also use the Find dialog box to find pin pairs and use wildcards in the Value field to filter the selection.

     ii. Right-click and click **Show Rules**.
   - Select nets from the list in the Pin Pair Rules dialog box:
     iii. On the **Setup** menu, click **Design Rules, then click the Pin Pairs** button.

     iv. In the Pin Pair Rules dialog box, in the Connections box, click one or more pin pairs.

   Tips:

   - To display pin pairs for a specific net, select the net in the *From Net* list.
   - If you want to display all pin pairs, clear the *Show pin pairs with rules* check box.
2. Click any of the three rule category buttons (Clearance, Routing, HiSpeed) to customize the rules from their default values and settings.
3. After you've customized the rule categories, close any rules dialog boxes to return to the design.

Result

After you customize a rule category for the pin pair(s), non-default rules indicators appear in the dialog box.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Modifying Pin Pair Design Rules

Follow the procedure in the topic Creating Pin Pair Design Rules.

Resetting Pin Pair Rules to Default Rules

With the click of a button, you can reset the pin pair rules to the same as the default rules.

Procedure

1. **Setup** menu > **Design Rules** > **Pin Pairs** button.
2. In the Pin Pair Rules dialog box, select one or more pin pairs in the Connections list, click **Default**, and then click **Yes**.

   Tip: The Default button is unavailable if the pin pair has only default rules assigned to it.

   Result: You know you've successfully reset the pin pair rules if all the icons below the rule categories (Clearance, Routing, HiSpeed) return to the Default icon, and the Default button is no longer available.
3. Close any rules dialog boxes to return to the design.

Creating Pin Pair Clearance-Rules for a Specific Layer

You can create a unique set of pin pair rules on a specific layer that take precedence over pin pair rules which apply to all layers.

Procedure

1. **Setup** menu > **Design Rules > Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Pin pairs and then select** a pin pair in the list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Layer** and then select a layer from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. In the Object to object box, type a value to apply to all objects or click the Matrix button to enter the Clearance Rules dialog box to apply different values between objects. (You must click OK in the Clearance Rules dialog box to accept the changes.)
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Class Against Class Design Rule

You can create clearance or high speed rules that apply between the same class or two different classes of nets.

Requirement

Both Classes of nets must already exist. **See also**: Creating Class Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Classes and then select** a class in the list.
3. In the Current rule set area, click **Clearance or High Speed**.
4. In the Against rule object area, click **Classes** and then select a class from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. Set the required Clearance or High Speed values.
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Class Against Class Design Rule for a Specific Layer

You can create clearance rules that apply between the same class or two different classes of nets on a specific layer. This also takes precedence over class against class clearance rules that apply to all layers.

Requirement

Both Classes of nets must already exist. **See also**: Creating Class Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.

2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Classes and then select** a class in the list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Classes** and then select a class from the list.
5. In the Apply to layer list, select a specific layer.
6. Click **Create**.
7. In the Existing rule sets list, if not already selected, select the newly created rule set.
8. Set the required Clearance values.
9. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Net Against Class Design Rule

You can create clearance or high speed rules that apply between a net and a class of nets.

Requirement

The Class of nets must already exist. **See also**: Creating Class Design Rules

Procedure

1. **Setup** menu > **Design Rules > Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Nets and then select** a net in the list.
3. In the Current rule set area, click **Clearance or High Speed**.
4. In the Against rule object area, click **Classes** and then select a class from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. Set the required Clearance or High Speed values.
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Net Against Class Design Rule for a Specific Layer

You can create clearance rules that apply between a net and a class of nets on a specific layer. This also takes precedence over net against class clearance rules that apply to all layers.

Requirement

The Class of nets must already exist. **See also**: Creating Class Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Nets and then select** a net in the list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Classes** and then select a class from the list.
5. In the Apply to layer list, select a specific layer.
6. Click **Create**.
7. In the Existing rule sets list, if not already selected, select the newly created rule set.
8. Set the required Clearance values.
9. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Net Against Net Design Rule

You can create clearance or high speed rules that apply between the same net or two different nets.

Procedure

1. **Setup** menu > **Design Rules > Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Nets and then select** a net in the list.
3. In the Current rule set area, click **Clearance or High Speed**.
4. In the Against rule object area, click **Nets** and then select a net from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. Set the required Clearance or High Speed values.
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Net Against Net Design Rule for a Specific Layer

You can create clearance rules that apply between the same net or two different nets on a specific layer. This also takes precedence over net against net clearance rules that apply to all layers.

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Nets and then select** a net in the list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Nets** and then select a net from the list.
5. In the Apply to layer list, select a specific layer.
6. Click **Create**.
7. In the Existing rule sets list, if not already selected, select the newly created rule set.
8. Set the required Clearance values.
9. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Group Against Class Design Rule

You can create clearance or high speed rules that apply between a group of pin pairs and a class of nets.

Requirement

The Group of pin pairs and the Class of nets must already exist. **See also**: Creating Group Design Rules, Creating Class Design Rules

Procedure

1. **Setup** menu > **Design Rules > Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Groups and then select** a group in the list.
3. In the Current rule set area, click **Clearance or High Speed**.
4. In the Against rule object area, click **Classes** and then select a class from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. Set the required Clearance or High Speed values.
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Group Against Class Design Rule for a Specific Layer

You can create clearance rules that apply between a group of pin pairs and a class of nets on a specific layer. This also takes precedence over group against class clearance rules that apply to all layers.

Requirement

The Group of pin pairs and the Class of nets must already exist. **See also**: Creating Group Design Rules, Creating Class Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Groups and then select** a group in the list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Classes** and then select a class from the list.
5. In the Apply to layer list, select a specific layer.
6. Click **Create**.
7. In the Existing rule sets list, if not already selected, select the newly created rule set.
8. Set the required Clearance values.
9. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Group Against Net Design Rule

You can create clearance or high speed rules that apply between a group of pin pairs and a net.

Requirement

The Group of pin pairs must already exist. **See also**: Creating Group Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Groups and then select** a group in the list.
3. In the Current rule set area, click **Clearance or High Speed**.
4. In the Against rule object area, click **Nets** and then select a net from the list.
5. Click **Create**.

6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. Set the required Clearance or High Speed values.
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Group Against Net Design Rule for a Specific Layer

You can create clearance rules that apply between a group of pin pairs and a net on a specific layer. This also takes precedence over group against net clearance rules that apply to all layers.

Requirement

The Group of pin pairs must already exist. **See also**: Creating Group Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Groups and then select** a group in the list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Nets** and then select a net from the list.
5. In the Apply to layer list, select a specific layer.
6. Click **Create**.
7. In the Existing rule sets list, if not already selected, select the newly created rule set.
8. Set the required Clearance values.
9. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Group Against Group Design Rule

You can create clearance or high speed rules that apply between the same group of pin pairs or two different groups of pin pairs.

Requirement

The Group(s) of pin pairs must already exist. **See also**: Creating Group Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Groups and then select** a group in the list.
3. In the Current rule set area, click **Clearance or High Speed**.
4. In the Against rule object area, click **Groups** and then select a group from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. Set the required Clearance or High Speed values.
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Group Against Group Design Rule for a Specific Layer

You can create clearance rules that apply between the same group of pin pairs or two different groups of pin pairs on a specific layer. This also takes precedence over group against group clearance rules that apply to all layers.

Requirement

The Group(s) of pin pairs must already exist. **See also**: Creating Group Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.

2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Groups and then select** a group in the list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Groups** and then select a group from the list.
5. In the Apply to layer list, select a specific layer.
6. Click **Create**.
7. In the Existing rule sets list, if not already selected, select the newly created rule set.
8. Set the required Clearance values.
9. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Pin Pair Against Class Design Rule

You can create clearance or high speed rules that apply between a pin pair and a class of nets.

Requirement

The Class of nets must already exist. **See also**: Creating Class Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Pin Pairs and then select** a pin pair in the list.

   Tip: You can also filter the list of pin pairs to a single net by selecting the net in the From net list.
3. In the Current rule set area, click **Clearance or High Speed**.
4. In the Against rule object area, click **Classes** and then select a class from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. Set the required Clearance or High Speed values.
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Pin Pair Against Class Design Rule for a Specific Layer

You can create clearance rules that apply between a pin pair and a class of nets on a specific layer. This also takes precedence over pin pair against class clearance rules that apply to all layers.

Requirement

The Class of nets must already exist. **See also**: Creating Class Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Pin Pairs and then select** a pin pair in the list.

   Tip: You can also filter the list of pin pairs to a single net by selecting the net in the From net list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Classes** and then select a class from the list.
5. In the Apply to layer list, select a specific layer.
6. Click **Create**.
7. In the Existing rule sets list, if not already selected, select the newly created rule set.
8. Set the required Clearance.
9. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Pin Pair Against Net Design Rule

You can create clearance or high speed rules that apply between a pin pair and a net.

Procedure

1. **Setup** menu > **Design Rules > Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Pin Pairs and then select** a pin pair in the list.

   Tip: You can also filter the list of pin pairs to a single net by selecting the net in the From net list.
3. In the Current rule set area, click **Clearance or High Speed**.
4. In the Against rule object area, click **Nets** and then select a net from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. Set the required Clearance or High Speed values.
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Pin Pair Against Net Design Rule for a Specific Layer

You can create clearance rules that apply between a pin pair and a net on a specific layer. This also takes precedence over pin pair against net clearance rules that apply to all layers.

Procedure

1. **Setup** menu > **Design Rules > Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Pin Pairs and then select** a pin pair in the list.

   Tip: You can also filter the list of pin pairs to a single net by selecting the net in the From net list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Nets** and then select a net from the list.
5. In the Apply to layer list, select a specific layer.
6. Click **Create**.

7. In the Existing rule sets list, if not already selected, select the newly created rule set.
8. Set the required Clearance values.
9. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Pin Pair Against Group Design Rule

You can create clearance or high speed rules that apply between a pin pair and a group of pin pairs.

Requirement

The Group of pin pairs must already exist. **See also**: Creating Group Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Pin Pairs and then select** a pin pair in the list.

   Tip: You can also filter the list of pin pairs to a single net by selecting the net in the From net list.
3. In the Current rule set area, click **Clearance or High Speed**.
4. In the Against rule object area, click **Groups** and then select a group from the list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. Set the required Clearance or High Speed values.
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Pin Pair Against Group Design Rule for a Specific Layer

You can create clearance rules that apply between a pin pair and a group of pin pairs on a specific layer. This also takes precedence over pin pair against group clearance rules that apply to all layers.

Requirement

The Group of pin pairs must already exist. **See also**: Creating Group Design Rules

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Pin Pairs and then select** a pin pair in the list.

   Tip: You can also filter the list of pin pairs to a single net by selecting the net in the From net list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Groups** and then select a group from the list.
5. In the Apply to layer list, select a specific layer.
6. Click **Create**.
7. In the Existing rule sets list, if not already selected, select the newly created rule set.
8. Set the required Clearance values.
9. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Pin Pair Against Pin Pair Design Rule

You can create clearance or high speed rules that apply between the same pin pair or two different pin pairs.

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Pin Pairs and then select** a pin pair in the list.

   Tip: You can also filter the list of pin pairs to a single net by selecting the net in the From net list.
3. In the Current rule set area, click **Clearance or High Speed**.
4. In the Against rule object area, click **Pin Pairs** and then select a pin pair from the list.

   Tip: You can also filter the list of pin pairs to a single net by selecting the net in the From net list.
5. Click **Create**.
6. In the Existing rule sets list, if not already selected, select the newly created rule set.
7. Set the required Clearance or High Speed values.
8. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating a Pin Pair Against Pin Pair Design Rule for a Specific Layer

You can create clearance rules that apply between the same pin pair or two different pin pairs on a specific layer. This also takes precedence over pin pair against pin pair clearance rules that apply to all layers.

Procedure

1. **Setup** menu > **Design Rules Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Source rule object area, click **Pin Pairs and then select** a pin pair in the list.

   Tip: You can also filter the list of pin pairs to a single net by selecting the net in the From net list.
3. In the Current rule set area, click **Clearance**.
4. In the Against rule object area, click **Pin Pairs** and then select a pin pair from the list.

Tip: You can also filter the list of pin pairs to a single net by selecting the net in the From net list.

5. In the Apply to layer list, select a specific layer.
6. Click **Create**.
7. In the Existing rule sets list, if not already selected, select the newly created rule set.
8. Set the required Clearance values.
9. When finished, close any open Rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating Decal Design Rules

Use the Decal Rules dialog box to define design rules that apply to all instances of a decal within the design. You can assign rules to one or more decals at a time.

Restriction

You can define Decal Rules in PADS Layout; however, these rules are used in PADS Router only.

Procedure

1. **Setup** menu > **Design Rules** > **Decal** button.

2. In the Decal Rules dialog box, in the Decals box, click one or more decals.

   Tip: To display all decals, clear the *Show decals with rules* check box.

3. Click any of the four rule category buttons (Clearance, Routing, Fanout, Pad Entry) to customize the rules from their default values and settings.
4. After you've customized the rule categories, close any rules dialog boxes to return to the design.

Result

After you customize a rule category for the class, non-default rules indicators appear in the dialog box.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Modifying Decal Design Rules

Follow the procedure in the topic Creating Decal Design Rules.

Resetting Decal Rules to Default Rules

With the click of a button, you can reset the decal rules to the same as the default rules.

Procedure

1. **Setup** menu > **Design Rules** > **Decal** button.
2. In the Decal Rules dialog box, select one or more decals in the Decals list, click **Default**, and then click **Yes**.

   Tip: The Default button is unavailable if the decal has only default rules assigned to it.

   Result: You know you've successfully reset the decal rules if all the icons below the rule categories (Clearance, Routing, Fanout, Pad Entry) return to the Default icon, and the Default button is no longer available.
3. Close any rules dialog boxes to return to the design.

Creating Decal Design Rules in the PCB Decal Editor

Use the Decal Rules dialog box to define design rules for the decal. The design rules are saved with the decal in the library and become active whenever the decal is used in a design.

Restriction

You can define Decal Rules in PADS Layout; however, these rules are used in PADS Router only.

Procedure

1. In the PCB Decal Editor, on the **Setup** menu, click **Decal Rules**.
2. In the Decal Rules dialog box, click any of the four rule category buttons (Clearance, Routing, Fanout, Pad Entry) to customize the rules from their default values and settings.
3. Close any rules dialog boxes to return to the design.

Result

After you customize a rule category for the decal(s), non-default rules indicators appear in the dialog box.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating Component Design Rules

Use the Component Rules dialog box to create design rules for one or more components.

Restriction

- You can define Component Rules in PADS Layout; however, these rules are used in PADS Router only.
-

Procedure

1. You can assign rules to one or more components at a time. There are two ways to select the component(s) before accessing the rule categories.
    - Select components in the design:
        i. In the design area, select the component(s) to which you want to apply rules.

            Tip: You can also use the Find dialog box to find components and use wildcards in the Value field to filter the selection.

        ii. Right-click and click **Show Rules**.
    - Select components from the list in the Component Rules dialog box:
        iii. **Setup** menu > **Design Rules** > **Component** button.

        iv. In the Component Rules dialog box, in the Components box, select one or more components.

    Tips:

    - To display components using a specific decal, select the decal in the *Using decal* list.
    - To display all components in the list, clear the *Show components with rules* check box.
2. Click any of the four rule category buttons (Clearance, Routing, Fanout, Pad Entry) to customize the rules from their default values and settings.

3. Close any rules dialog boxes to return to the design.

Result

After you customize a rule category for the pin pair(s), non-default rules indicators appear in the dialog box.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Modifying Component Design Rules

Follow the procedure in the topic Creating Component Design Rules.

Resetting Component Rules to Default Rules

With the click of a button, you can reset the component rules to the same as the default rules.

Procedure

1. **Setup** menu > **Design Rules** > **Component** button.
2. Select one or more components in the Components list, click **Default**, and then click **Yes**.

   Tip: The Default button is unavailable if the component has only default rules assigned to it.

   Result: You know you've successfully reset the component rules if all the icons below the rule categories (Clearance, Routing, Fanout, Pad Entry) return to the Default icon, and the Default button is no longer available.
3. Close any rules dialog boxes to return to the design.

Creating Via Routing Rules in the Decal Editor

Use the Routing Rules dialog box to specify which vias may be used with the decal.

-
- When you open the Routing Rules dialog box from the Decal Editor, only options in the Vias area are available.

Procedure

1. **Decal Editor** > **Setup** menu > **Decal Rules** > **Routing** button.

2. In the Routing Rules dialog box, in the Vias area, click **Via definition**.
3. In the Setup Via dialog box, specify vias to use for routing rules in the Decal Editor:
   a. To add a via, type the via name into the box, and then click **Add**.
   b. To rename a via, select the via in the vias list, and then click **Rename**.
   c. To delete vias, select the vias in the Vias list, and then click **Delete**.
   d. Click **OK**.

   Tip: The Setup Via dialog box does not know about via padstacks in the design. Use this dialog box to set up only via names, not the internal structure of the pad stacks. These via names should reference the real vias in your design, where the internal padstack structure is defined.
4. In the Routing Rules dialog box, do any of the following:
   - To make vias available to the Decal Editor, select the vias and click **Add**.
   - To make vias unavailable to the Decal Editor, select the vias and click **Remove**.

Related Topics

Routing Rules Dialog Box

Design Rule Hierarchy in the *Concepts Guide*

Creating and Editing PCB Decals

Deleting a Conditional Rule

You can delete conditional rules that have been set up in the Conditional Rule Setup dialog box.

Procedure

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.
2. In the Conditional Rule Setup dialog box, in the Existing rule sets list, select one or more rules to delete.
3. Click the **Delete** button.

Modifying a Conditional Rule

You can make changes to an existing conditional rule.

1. **Setup** menu > **Design Rules** > **Conditional Rules** button.

2. In the Conditional Rule Setup dialog box, in the Existing rule sets list, select the rule to modify.

   Result: In the Current rule set area, the rule set type enables either the clearance or high-speed value options.

3. Modify the Clearance or High speed rule values.

   Tips:

   - For reporting purposes, nets and pin pairs in the Source rule object list are identified as aggressors. If a class is in the Source rule object list, all nets in the class are identified as aggressors.
   - Rules specified in this dialog box override high-speed rules specified in the HiSpeed Rules dialog box.

4. Close all open rules dialog boxes.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Creating Differential Pair Design Rules

You can define differential pair rules for nets or pin pairs that behave electrically as differential pairs.

-
- You can define differential pair rules in PADS Layout or PADS Router; however, these rules are used in PADS Router only. To verify differential pairs in the design, you must check them in latium checking.

Procedure

1. **Setup** menu > **Design Rules** > **Differential Pairs** button.
2. In the Differential Pairs dialog box, choose the Net or Pin pairs tab.
3. In the Available list, double-click the first net, and then double-click the second net.

   Tips:

   - Double-clicking is a shortcut instead of clicking the Select>> button.
   - Nets or pin pairs cannot exist in more than one differential pair. The Available list displays only items that have not been assigned to a differential pair.

4. Click **Add**.
5. Set the Properties of the pair.

6. Click **OK**.
7. Close the Rules dialog box.

Deleting a Differential Pair Design Rule

Use the Differential Rules dialog box to delete a differential pair rule.

Procedure

1. **Setup** menu > **Design Rules** > **Differential Pairs** button.
2. In the Differential Pairs dialog box, choose the Net or Pin pairs tab.
3. Select the differential pair in the Pairs list, and then click **<<Remove**.
4. Click **OK**.
5. Close the Rules dialog box.

Creating a Report of the Design Rules

Use the Rules Report dialog box to create a report of some, or all design rules. You can then print the report of the design rules.

Procedure

1. **Setup** menu > **Design Rules** > **Report button**.
2. In the Rules Report dialog box, in the Rule Types area, enable any of the rule types.
3. In the Pin pairs, Groups, Components, Nets, Classes, and Decals areas, do one of the following:
   - Select the check box to report all objects.
   - Select one or more items in the list to report only specific objects.

   Tip: In the Pin pairs area, you can filter the contents of the Pin pair list by selecting a net in the Net list.
4. If you want to report the default rules for each enabled rule type in the Rules Types area, select the **Default Rules** check box.
5. In the Output area, click one of the following options:
   - **Rule Sets**—Report all rules that are different from the default rules

- **Rule Values**—Report the values of all rules, even if they match the default rules values

6. Click **OK**.

Result

The report is written to C:\PADS Projects\rules.rep and displayed by the default text editor.

Turning on Design Rule Checking

You can enable live design rule checking as you work on your design. It prevents or reports design rule violations during interactive placement or routing. This is called On-line DRC. Unless you constrain your actions using DRC, you can make design errors. And although you can check and find errors at any time by running the Verify Design tool, you risk significant redesign when you ignore design rule checking and compound design errors.

Restriction

There are many design rules that are not checked by On-line DRC. Some are only checked by PADS Router (for example, Maximum number of vias) and others can only be checked as a post process by the Verify Design tool (for example, net Capacitance). Check the documentation and test the rule to ensure it works with On-line DRC.

Procedure

There are two methods to control On-line DRC.

- Using the Options dialog box:
  i. On the **Tools** menu, click **Options**.
  ii. In the Options dialog box, click the **Design** page.
  iii. In the On-line DRC area, select a setting.
- Using one of the Design Rule Checking modeless commands (drp, drw, dri, or dro).

Checking Design Rules

You can check design rules during interactive place and route operations, or after place and route operations are complete.

See also: Turning on Design Rule Checking

Use Verify Design to check for design rule violations after placement or routing, and report any violations in the work area and in a text file.

See also: Verify the Design

Restricting Heights on Component Layers

You can create a board height restriction to limit the height of components placed on the board.

Tips:

- You cannot use a height restriction of 0 (zero) to prevent placement on a layer. A height restriction of 0 specifies that there is no restriction. To prevent placement of all components on a layer, make the layer a Routing layer instead of a Component layer in the Layers Setup dialog box.
- If you want to have a visual cue that a board has a height restriction, use the procedure in Restricting Heights in Areas of Component Layers to create a keepout area over the entire layer.

Restriction

- A board height restriction won't prevent placement of a decal that has no geometry.height attribute.
- No warning is given if you create a restriction *after* a component that is taller than the height restriction has been placed. The violation will be caught, however, when you run a clearance check in Verify Design.

Procedure

1. **Tools** menu > **Options** > **Drafting tab**
2. In the Board component height restriction area, enter the height restriction for the Top and/or Bottom layer.
3. Click OK.

Results

You will be unable to place components whose geometry.height attribute value is greater than the specified height restriction.

Related Topics

Restricting Heights in Areas of Component Layers

Restricting Heights in Areas of Component Layers

You can create a board height restriction to limit the height of components placed in an area of the board.

Restrictions

- A board height restriction won't prevent placement of a decal that has no geometry.height attribute.
- No warning is given if you put a keepout over an already-placed component that is taller than the height restriction. The violation will be caught, however, when you run a clearance check in Verify Design.

Procedure

1. In the Layer list of the standard toolbar, select a component layer.
2. Create a keepout **in the area of the board where you want to restrict component height.**
3. In the Restrictions area of the Add Drafting dialog box that appears after you create the keepout, select the Placement and the Component height checkboxes.
4. Type the height restriction in the box.
5. Click OK.

Results

You will be unable to place components whose geometry.height attribute value is greater than the specified height restriction.

Related Topics

Restricting Heights on Component Layers

Using Keepouts

Keepouts prevent the placement of design items within a specified area. Keepouts appear as one pixel-width lines that are used during interactive placement, cluster placement, routing, and other operations. The outer edge of design objects and board outlines/cutouts are used for clearance calculations against keepouts. Online Design Rule Checking and Verify Design recognize keepouts as obstacles.

You can create keepouts in both the Layout Editor and the PCB Decal Editor. User interaction is the same for either; the only difference is the type of objects you can restrict.

Restriction: The three component type keepouts can only be created in the Layout Editor.

In this topic:

- Creating Keepout Areas
- Modifying a Keepout

Creating Keepout Areas

Create a keepout to define areas where design objects cannot be placed. You can create keepout areas using closed polygons (with or without arcs), circles, or rectangles. The current angle mode and design grid settings determine the placement of the lines.

Procedure

1. **Drafting Toolbar** button > **Keepout** button.
2. Right-click and click a draw mode for the type of shape to create.
3. Create a closed shape to define the keepout area.
4. In the Add Drafting dialog box that appears, select restrictions.
5. Click the layer on which to place the keepout.

   Tip: When you choose layer assignments, restrictions not available for that layer are unavailable. For example, if you choose a non-placement layer, the Placement check box will not be available.
6. Click **OK**. The keepout is created. If you create other keepouts, they use the restrictions you set here as the default.

Modifying a Keepout

You can change the size of a keepout just as you would any other drafting object: move an edge or corner, or change the diameter of a circle. You can also copy a keepout to another location and change its restrictions or layer assignments.

Procedure

1. Select the keepout by selecting an edge and right-clicking and clicking **Select Shape**.
2. Right-click and click **Properties**.
3. Turn restrictions on or off and modify the layer settings. See also: Modifying Drafting Object Properties
4. Click **OK**.

Tip: You can't modify a keepout that is part of a physical design reuse. If you try to, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the operation.

Related Topics

Modifying Drafting Object Properties

Restricting Heights on Component Layers

Attribute Dictionary in the *Concepts Guide*

Chapter 22
Part Placement

Component Placement Process

Parts are initially located overlaying at the origin on the top layer. Before placing components: create a board outline, set up the layers of the design, set the grid values, and set up the clearance design rules - specifically those that apply to component placement. Many commands are available to place your components. You can:

- Switch to PADS Router and use it to do all of your placements.

  See also: Switching to PADS Router

- On the Tools menu, click **Disperse Components** to disperse all the components around the board outline.

  Tip: On the View menu, click Extents after you disperse components to bring all design objects into view.

- Move components in the design.

  See also: To Move Components, To Move with Drag and Attach, To Move with Drag and Drop

- Orient parts radially on a polar grid.

  See also: To Set Up a Polar Grid

- Move multiple components sequentially.

  See also: To Use Move Sequential

- Flip components to the bottom layer.

  See also: Modifying Board-Side Location of Components

- Array components.

  See also: To Create a Component Array

- Align edges or centerlines of multiple components.

  See also: To Align Objects

- Rotate objects in 90 degree increments or at any angle.

  See also: To Rotate an Object, To Spin an Object

- Swap the locations of two parts.

  See also: To Swap Parts

- Automatically nudge overlapping components into positions that agree with the clearance rules. If you disable design rule checking, you can place parts into positions that disagree with your design rule clearances.

See also: To Nudge Overlapping Parts

Setting the Origin of an Object

Set the origin of an object to move it or place it. You can set the origin of a board outline, 2-D line, decal shape (in the PCB Decal Editor), copper shape, keepout shape, and dimension shape.

Procedure

To set the origin of an object using the Dimensioning Shape or Drafting Shape shortcut menus:

1. Select the whole shape.

   Tip: Use the right-click Select Shape, Select Documentation, or Select Board Outline selection shortcuts to select the whole shape. As an alternative, you can **Shift** and select the a segment - this selects the shape rather than a segment or corner.

2. Right-click and click **Set Origin**.
3. Click to set the new origin point. A message appears to confirm the origin point.
4. Click **Yes** to change the origin point or click **No** to use the previous origin point.

Tip: To exit Set Origin mode, press **Esc**.

To Minimize Length

Before you begin placement, consider setting the topology types you want to use. When you run a length minimization, it does not change the netlist, it just finds better places to make the same connections the netlist specifies, based on the topology types you set.

- Tools menu > Length Minimization

Related Topics

Placement and Length Minimization in the *Concepts Guide*

Topology type in the Routing Rules dialog box

Using the Find Dialog Box During Placement

If you imported parts using the .asc format, the placement process usually begins with design parts overlaying at the design origin. You can use the Find dialog box to break out and select certain parts, or groups of a package type. The dialog box can also attach parts to your pointer for individual placement.

To Move Components

In Verb Mode

To move a component in Verb Mode:

1. **Design Toolbar** button > **Move** button.
2. Select a component. The component attaches to your pointer.
3. Click to indicate a new location to complete the move.

In Object Mode

Use one of the following methods:

Move Command

1. Select the component to move.
2. Right-click and click **Move**.

   Tip: Right-click and click command view properties or edit the selected object. You can rotate, flip to the other outside layer, or spin the object while it is attached to your pointer.
3. Click to indicate a new location for the object.

Dynamic Drag Methods

Use Drag and Attach and Drag and Drop to automatically invoke the Move command for selected components.

See also: The Drag moves setting on the Global / General page of the Options dialog box.

Tips:

- If you move a component with a pin that is a locked test point, a Warning appears.

  See also: Troubleshooting

- When you move a group, an error may appear in one of the Trace Copy dialog boxes.

  See also: Trace Copy Dialog Box

- If you try to move a component that is part of a physical design reuse, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the move.

To Move with Drag and Attach

To move with Drag and Attach:

1. Select the component to move.
2. Place your pointer over the selected component and left-click holding the left mouse button down. Move the pointer to initiate a drag move.
3. Release the left mouse button. The component remains dynamically attached to the pointer.
4. Click to indicate a new location to complete the move.

Tips:

- If you move a component with a pin that is a locked test point, a Warning appears.

  See also: Troubleshooting

- When you move a group, an error may appear in one of the Trace Copy dialog boxes.

  See also: Trace Copy Dialog Box

- If you try to move a component that is part of a physical design reuse, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the move.

To Move with Drag and Drop

To move with Drag and Drop:

1. Select the component to move.
2. Place your pointer over the selected component and left-click holding the left button down. Move the pointer to initiate a drag move.
3. Move the pointer to the object's new location and release the left mouse to complete the move.

Tips:

- If you move a component with a pin that is a locked test point, a Warning appears.

  See also: Troubleshooting

- When you move a group, an error may appear in one of the Trace Copy dialog boxes.

  See also: Trace Copy Dialog Box

- If you try to move a component that is part of a physical design reuse, the message "Reuse elements cannot be modified. Break the reuse first" appears. Click OK to cancel the move.

To Set Up a Polar Grid

Before you can use Radial Move, you need to set up a polar grid and the move options.

Procedure

1. **Tools** menu > **Options** > **Grids** tab.
2. Click **Radial Move Setup**.
3. In the Radial Move Setup Dialog Box, set up options and click **OK**. You can now use Radial Move.

Tip: Use the modeless command GP to make the polar grid visible in the design. You can use Setup menu > Set Origin to change the location of the polar grid origin, but you must Redraw to see the change.

Related Topics

Defining Arrays in the *Concepts Guide*

To Use Radial Move

Use Radial Move to adjust the orientation of objects. This command supports creation of circular decals and circular boards. You can also use Radial Move with text and reference designators.

Requirement: You must set up a Polar Grid.

1. Select single or multiple components or unions. In the PCB Decal Editor, select single or multiple terminals.
2. Click the **Design toolbar** button on the standard toolbar. The Design toolbar appears.
3. Click the **Radial Move** button on the **Design** toolbar. The Radial Move Setup dialog box appears. The status bar displays the current polar radius and polar angle position in addition to the angular and radial increments.

See also: Radial Move Setup Dialog Box

4. If you did not previously set up the polar grid, the Radial Move Setup dialog box appears.

   Set up the polar grid and move options and click **OK**. PADS Layout saves these settings to apply to any future use of Radial Move in the current session.

   See also: Polar Grid and Radial Move Example in the *Concepts Guide*

   The polar grid appears and the selected objects attach to the pointer.

5. Move the selected objects to an eligible site.

6. Click to indicate a location on the polar grid for the selected objects.

Radial Move in Verb Mode

To use Radial Move in Verb Mode:

1. Click the **Design toolbar** button on the standard toolbar. The Design toolbar appears.

2. Click the **Radial Move** button on the **Design** toolbar. The Radial Move Setup dialog box appears.

   See also: Radial Move Setup Dialog Box

3. Set up the polar grid and click **OK**.

4. Select a single component or union. In the PCB Decal Editor, select a single terminal. The polar grid appears and the selected object attaches to the pointer.

5. Move the selected object to an eligible grid site.

6. Click to indicate a location on the polar grid for the selected object.

Tip: When you exit Radial Move the polar grid is turned off. Use the GP modeless command to turn the polar grid on and off independently of Radial Move.

In the PCB Decal Editor, you can only use Radial Move with terminals.

To Use Move Sequential

After selecting a predetermined set of components, you can use Move Sequential to sequentially place components on your pointer for placement in the design.

Procedure

1. **Edit** menu > **Find**.

2. Set **Find By** to **Reference Designator** or **Part Type**.

Alternative: Instead of using the Find dialog box, you can also select components in the Project Explorer window or the design area, then right-click in the design area and click Move Sequential.

3. Use the resulting list to identify and highlight all the parts you want in the selection.
4. Click **Move Sequential** in the **Action** list and click **Apply**.

When all parts are selected, a single part attaches to the pointer and "Proceed with next object?" appears. You can select:

- **Yes** to select part after part, but prompt every time after a placement.
- **Yes to All** to select part after part, but no prompt after a placement.
- **No** to skip this part, go to next.
- **Cancel** to cancels the command. You return to the Find dialog box.

When a part attaches to the pointer, you can use any **Move** command on the shortcut menu. You can also click **Move Sequential** from the following shortcut menus: component, union, cluster, or mixed component, union, and cluster. You can only use Move Sequential with Radial Move if you turn the polar grid on using the GP modeless command.

Tips:

- You can use Move by Midpoint with Move Sequential when moving components, unions, and clusters.
- If a part selected for Move Sequential has a pin that is a test point, a Warning dialog box appears informing you of changes you are making to the test point.

  See also: Troubleshooting

You can also use the S, search modeless command, to position the pointer exactly. Type S, the X,Y coordinates you want the pointer to move to, and press Enter. The pointer moves to the coordinates, taking the part, by the origin, with it.

The parts are all selected and can be queried or moved away from the origin. If you've separated a group of package types, use Find to select the reference designators individually. Once an individual part is selected, you can drag it.

Related Topics

Commands for Find By

To Create a Component Array

Using the Cluster Placement Dialog Box

Find Dialog Box

Using the Find Dialog Box During Placement

Modifying Board-Side Location of Components

Use Flip Side to move a part or parts to the opposite side of the board. The flipped part moves to the opposite side in a hinging motion, pivoting on a vertical axis through the part origin. Text is mirrored so it always appears in the proper orientation. Use Flip Group to flip a component or components around an origin point you specify.

When you flip a component to the opposite side, layer associations change with the component. To turn off moving layer associations with components, set the /NTL switch before using Flip Side or Flip Group.

Warning: Use associated copper rather than free copper in decals in the solder or paste mask. Results are unexpected when TrueLayer is on and you use free copper.

To Flip a Component

To move a component to the opposite side of the board using its origin as the mirror point:

1. Select the components to move to the opposite side.
2. Right-click and click **Flip Side**.

   Or

   Click **Properties**. Select a different layer in the **Layer** list in the Properties dialog box.

Tip: If DRC is on and the system encounters a clearance violation, the flip is canceled. Test points flip with the flipped component.

To Flip a Group

Move a group of components to the opposite side of the board by specifying the mirrored location:

- Select the components > right-click > **Flip Group**.
- Click to indicate the mirror location.

Using the /NTL Switch

Layer associations automatically switch with a component when you flip it to another side. Set the /NTL switch before starting PADS Layout to turn TrueLayer associations off.

See also: Start-up Options

To Create a Component Array

You can arrange parts by creating arrays. A component array is a union with members placed on sites of a user-defined matrix. You can create either planar arrays or circular arrays.

To create a component array:

1. Select the components you want > right-click > **Create Array**.
2. Click the tab that represents the array you want to create: Planar Array or Circular Array.
3. Set the options for the array and click **OK**. A prompt appears asking you for an array or union name.

   See also: Defining Arrays in the *Concepts Guide*.
4. Type a name for the array or accept the default name and click **OK**. By default, arrays are sequentially named with an ARR_ prefix; for example, the first array in your design is named ARR_1, the second array is named ARR_2, and so on.

   The array is created and attaches to your pointer.
5. Click to indicate a location for the array.

Tips:

- You can use Move shortcut menu commands; such as Flip, Spin, and Rotate; while placing the array.
- If you move a component with a pin that is a locked test point, a warning appears.

  See also: Troubleshooting
- If you try to move a component that is part of a physical design reuse, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the move.

To Modify a Component Array

Using Modify Array, you can rearrange the components in an array after placing the array, or you can modify a union.

Requirements:

- You must select Unions in the Selection Filter to select arrays and unions.
- You can only modify one array or union at a time.

To modify an array or union:

1. select an array or union > right-click > Modify Array

   Results:

   - If you are modifying an array, the Modify Array dialog box appears with the tab and the options used to create the array.
   - If you are creating an array from a union, a prompt appears asking whether to dissolve the union to create an array. Click **Yes** to create the array from the Union. The Create Array dialog box appears.

2. Modify array options as necessary.
3. Click **OK**. The array is modified or created from the union.
4. If **Online DRC** or **DRC Warn** is selected, click to indicate a new position for the modified array.

Tips:

- Modify Array does not enter Move mode as Create Array does.
- Modify Array does not warn against nor prevent placement violation unless Online DRC or DRC Warn is selected.
- If you modify the location of an array that contains a component with a pin that is a locked test point, a warning appears.

  See also: Troubleshooting

To Align Objects

Once objects are roughly in position, you can select several objects and automatically align them. The alignment command aligns all selected objects with the last one selected. You can use Align with pins (and align components by those pins), reference designators, unions in the Layout Editor, and terminals and terminal numbers in the PCB Decal Editor.

Procedure

1. Using Ctrl+click, select the objects you want to include in the alignment ensuring that you select the master object last.
2. Right-click and click **Align**. The Align Parts dialog box appears in the Layout Editor. The Align Pins dialog box appears in the PCB Decal Editor.
3. Click the alignment scheme you want to use. The objects are automatically aligned according to the position of the last one selected.

   Tip: Automatic alignment does not guarantee minimum spacing if online DRC is not selected, but you can use Nudge to accomplish this interactively. If DRC is set to

Prevent or Warn, Align does not perform the alignment if it causes a violation. Violations are reported in the status bar.

Restriction: You can't align elements in a physical design reuse. Align becomes unavailable when you select physical design reuse elements.

To Rotate an Object

1. Select the object to rotate. You can select more than one object at a time.
2. Right-click and click **Rotate 90**. The selected object rotates 90 degrees counterclockwise.
3. Repeat step 2 to continue rotating the objects.

To use Rotate 90 in Verb Mode, click **Rotate** on the **Design** toolbar and then select the part to rotate.

Tips:

- If you rotate a component with a pin that is a locked test point, a warning appears.

  See also: Troubleshooting

- When you rotate a group, an error may appear in one of the Trace Copy dialog boxes.

  See also: Trace Copy Dialog Box

- If you try to rotate a component that is part of a physical design reuse, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the operation.

To Spin an Object

1. Select the object to spin.
2. Right-click and click **Spin**. The part attaches to the pointer. As you move the pointer, the part follows, changing the rotation angle.

   You can multiple-select and rotate a group of parts in the same way; each turns on its own axis.
3. Click to indicate a new angle.

To use Spin in Verb Mode, click **Spin** on the **Design** toolbar, and then select the part to spin. click to indicate a new angle.

Tips:

- To view the angle of rotation as you spin the part, right-click and click Properties. The current angle appears in the Rotation box.
- If you spin a component with a pin that is a locked test point, a warning appears.

  See also: Troubleshooting
- If you try to spin a component that is part of a physical design reuse, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the operation.

To Swap Parts

Sometimes simply switching neighboring parts' positions can improve connection length. You can swap the position of two parts anywhere on the board using Swap Part. When online DRC is selected, the swap is examined for placement violations.

Using Object Mode

To swap parts:

1. Select a first part > **Design Toolbar** button > **Swap Part** button.
2. Select the second part to switch. The X + Y connection lengths before and after the swap appear so you can see whether the lengths improve.
3. Click **Yes** to complete the swap. Parts are swapped origin-to-origin.

Using Verb Mode

To swap parts in Verb Mode:

1. **Design Toolbar** button > **Swap Part** button.
2. Select the first part to switch.
3. Select the second part to switch. The X + Y connection lengths before and after the swap appear so you can see whether the lengths improve.
4. Click **Yes** to complete the swap. Parts are swapped origin-to-origin.

To Nudge Overlapping Parts

Use Nudge to move overlapping parts. The manner in which Nudge operates is dependent on the current DRC setting.

See also: Nudging Parts in the *Concepts Guide*

Nudging All Components

- On the Tools menu, click **Nudge Components**. All components are automatically nudged according to the design rules.

Nudging Single Parts

1. Select a part.
2. Right-click and click **Nudge**.

   See also: Nudge Parts and Unions Dialog Box
3. Click the nudge direction and the direction to move overlapping parts: **Automatic**, **Left**, **Right**, **Up**, or **Down**.
4. Click **Run** to perform the nudge. Click **Undo** in the Nudge dialog box to reverse the move.

Often, several parts are adjusted to accommodate the moved part. Nudge identifies these parts, and allows for further adjustments, by displaying them in a specific color:

- **Select color**—All parts affected by the Nudge routine.
- **Highlight color**—The part to move in the next pass of Nudge.

Tips:

- Nudge does not move glued parts or parts outside the board outline. Parts inside the board outline are not nudged outside the outline.
- Nudge considers test points to be glued objects.
- You can't nudge elements in a physical design reuse. All component elements are considered glued. Nudge is unavailable when you select physical design reuse elements.

To Change Part Outline Width

To change the part outline width:

1. Select a component > right-click > **Properties**.
2. Type a width (between 0 and 250) in the **Part Outline Width** box.
3. Click **OK**.
4. To update all parts that have the same decals as the selected parts with the new part outline width click **Continue**. Click **Cancel** to cancel changing the part outline width to all parts that have the same decals, including the selected parts.

Tips:

- This change is reflected in the design only; the library is not updated.
- Alternate decals in use are modified; alternate decals not in use are not modified.
- Online DRC Body to Body checking and Verify the Design look at the part outline when checking.

Modifying Component Properties

To edit component properties:

1. Select a component.
2. Right-click and click **Properties**. The Component Properties Dialog Box appears.

Tip: Several of the options on the dialog box are unavailable when the component is part of a physical design reuse.

Creating a Label

You can create labels using the Labels tab in the Component Properties dialog box.

1. Click <**new**> from the Label list and click the **Label** button.
   See also: Adding a New Part Label
2. From the **Attribute** list, click the attribute for which you want to create a label. When you click an attribute, the value for the attribute appears in the Value box.

   Tip: The only attribute available for jumpers is the reference designator.
3. Accept the current value or type a new value. If you click **Reference Designator** or **Part Type** from the Attribute list, you cannot change the value.

   This box is unavailable if the attribute is read-only. This box is also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.
4. Set the visibility status and the placement information for the label.
5. Set the justification, right-readability, height, and width of the label.
6. Click the layer on which to place the label, and click **OK** to create the label.

Tips:

- If you don't set visibility information (such as in steps 5, 6, and 7), default positions are used.

  See also: Label Defaults in the *Concepts Guide*
- Labels are not checked for clearance violations. If you want to output labels from CAM as metal or copper, such as on a Routing layer, check your label placement carefully.

Editing a Label

The Label List contains existing labels for reference designator, part type, and attributes. To edit an existing label:

- Click a label in the list and click the button in this tab. A label is selected instead of the component, and the corresponding Part Label Properties dialog box appears where you can modify the label.

Tips:

- When modifying the Properties of a jumper name, Reference Designator is the only available label.
- When the current color for labels is set to the background color, this option is unavailable. To activate Label, assign a non-background color to labels in the Display Colors Setup dialog box.

Related Topics

Component Properties Dialog Box

Unions

Managing Unions

Unions are user-created part associations that have a strict relationship with each other, such as distance, rotation angle, top, or bottom side. A common example is placing a filter capacitor to reside on top of an IC. When a selected union is moved or placed, the physical relationship between the parts, or union members, remains unchanged.

This topic discusses the following:

- Creating a New Union
- Creating Like Unions
- Selecting Unions
- Adding a Part to a Union
- Modifying Unions
- Deleting Unions and Members

Creating a New Union

1. Select and position the parts that you want to make a union in a specific pattern. Leave the parts selected.

2. Right-click and click **Create Union**. The Union Name Definition dialog box appears with a default name.

3. Use the default name or type a new name for the union, and click **OK**.

Creating Like Unions

You can use a union as an example to automatically create other unions of the same configuration.

To automatically create other unions which match an existing union, or base part:

1. Select a union > right-click > **Create Like Unions**.

2. Click **Yes** to continue. A message appears prompting whether to disperse the new unions.

3. Click **Yes** to disperse the newly created unions around the board outline.

If you click No to disperse parts, the message "Keep Base?" appears. This message asks whether to position the members of the new unions to match the union used for creation. Click Yes to reposition the new unions exactly like the base part; click No to leave the new unions in their current position.

Tip: Create Like Unions ignores parts with test points.

Selecting Unions

There are two ways to establish filter settings for selecting unions:

- Select a union from a union member:

  a. Select a member of a union.

  b. Right-click and click **Select Union**.

- Automatically select a union when one of its members is selected.

  a. **Edit** > **Filter**

  b. Click both the **Parts** and **Unions** options.

  See also: Selecting Objects Among Others

Adding a Part to a Union

1. Set the filter to select the entire union when you select a member.

   See also: Selecting Objects Among Others

2. Use multiple selection, Ctrl+click, to select the parts to add.

3. Right-click and click **Create Union**. The message “Break union and include its parts in new?” appears.
4. Click **Yes** to add the new parts.
5. Click **Yes** to use the same union name.

Modifying Unions

The Union Modification Flag setting which prevents you from modifying unions is automatically set when you start PADS Layout.

To disable this setting:

1. Select a member of any union.
2. Right-click and click **Modify Union Member**. This option remains active for the entire PADS Layout session. Click the option again to deactivate it.

Deleting Unions and Members

You can remove a part from the union or remove the entire union.

To remove a part from a union:

1. Select the part to remove.
2. Right-click and click **Break From Union**.

To remove an entire union:

1. Select the union.
2. Right-click and click **Break**.

Tip: Use the Break Like Unions and Break All Unions options to remove all like unions or remove all unions.

Related Topics

Union Properties Dialog Box

Cluster Placement

With Cluster Placement you can create associations, or groupings, of connected parts. Cluster Placement works with the following two object types:

- **Unions** — User-created part associations that have a strict relationship with each other.

- **Clusters** — Collections of individual parts, unions, and other clusters, based on connectivity.

In this topic:

- Creating New Clusters
- Modifying Existing Clusters
- Cluster View Mode
- Display Parts in Cluster View Mode
- Moving Clusters Interactively
- Deleting a Cluster
- Collapsing Clusters
- Collapsing All Clusters
- Collapsing Cluster Members

Creating New Clusters

To manually create new clusters:

1. Select the parts, unions, and other clusters to include in the cluster.
2. Right-click and click **Create Cluster**. The cluster parts are erased and replaced by a circle with all connections originating from its center. This circle represents the cluster.

Modifying Existing Clusters

You can add parts and clusters to another cluster manually, semi-automatically, or automatically. You can only remove parts from existing cluster manually.

1. Select the clusters to change.
2. Right-click and click:

   Edit Manual—Click Add to Cluster to add to the selected cluster. The item appears in the current Highlight color. Click **Remove from Cluster** to remove a highlighted item from the cluster. Click **Complete** from the shortcut menu to finish modifying the cluster.

   Grow Incremental—Opens the Cluster Grow Incremental dialog box which contains information on the highlighted cluster plus the following buttons:

Accept	Adds the highlighted item to the selected cluster.

Skip	Skips this item and highlights the next.
Stop	Adds the items identified through the Accept button and completes the operation.
Cancel	Cancels the operation without modifying the selected cluster.

Grow Automatic—Opens the Cluster Size Limit Definition dialog box which displays the current size of the cluster, or the number of cluster members, plus a text window containing the recommended number of cluster members. Click OK to accept the new value, or type a higher value to add additional parts to the cluster.

Tip: Use the Cluster Manager to modify existing clusters using a dialog box.

Cluster View Mode

PADS Layout automatically switches to Cluster View Mode any time you create or modify clusters, unless you are using Cluster Manager. Click Clusters from the View menu to switch between Cluster View and normal view.

In Cluster View Mode, several commands and options are unavailable or limited.

Display Parts in Cluster View Mode

To display parts belonging to a cluster while in Cluster View Mode:

1. **Edit** menu > **Filter**.
2. Click **Clusters**.
3. Select the cluster.
4. Right-click and click **Show Contents**. The parts appear in the current Highlight color.
5. To return the cluster to cluster view mode right-click and click **Cancel**.

Moving Clusters Interactively

To reposition a cluster:

1. Select a cluster > right-click > **Move**.
2. Move the cluster to a new location and click. The cluster moves to the new location and the members of the cluster collapse.

You can set PADS Layout so that clusters do not collapse after moving. For more information see “Collapsing Clusters.”

Tip: Autoplacement features ignore components that are part of a physical design reuse.

Deleting a Cluster

To remove a cluster:

1. Select the cluster. You can make multiple selections.
2. Right-click and click **Break**.

To remove all clusters:

1. Select any cluster.
2. Right-click and click **Break All Clusters**.

Collapsing Clusters

Collapse relocates the members of a cluster to the center of the cluster. The Place Parts process of Cluster Placement eliminates overlaps created when you collapse a cluster. You can also move each part manually. Collapse operations ignore glued objects or unions containing glued objects.

Tip: To switch between viewing parts and viewing clusters click **Clusters** on the **View** menu.

Collapsing All Clusters

Procedure

1. With nothing selected, right-click and click Select Clusters.
2. Right-click and click Select All.
3. Right-click and click Collapse Members.

Collapsing Cluster Members

After moving a cluster, PADS Layout collapses the cluster. This is the default setting. To turn this option off use Collapse Cluster Members from the shortcut menu. You must be moving a cluster to access this command from the shortcut.

Tip: Collapse Cluster Members is set to on each time you start PADS Layout.

Related Topics

Cluster and Union Placement in the *Concepts Guide*

Cluster Information Properties Dialog Box

Cluster Properties Dialog Box

Build Clusters Setup Dialog Box

Cluster Manager Dialog Box

Cluster Placement Dialog Box

Place Clusters Setup Dialog Box

Place Parts Setup Dialog Box

Using the Cluster Placement Dialog Box

Use the Cluster Placement dialog box to build new clusters, place clusters within the board outline, and place parts within the board outline.

In this topic:

- Preparing for Automatic Placement
- Placing Parts Automatically
- Cluster Placement Status Dialog Box

Preparing for Automatic Placement

This topic discusses basic automatic placement procedures that will accommodate the average digital design. Use the following procedures to prepare your design for automatic placement and begin the automatic placement process.

After creating a board outline and importing a netlist that places all parts at the origin follow these steps:

1. Set the **Design Grid** to accommodate the parts you want to place. In English units a common grid is 100 or 50 mils for ICs and 50 mils for discrete parts. Start with a grid that is the most common denominator for your ICs.
2. Place all parts that should be in a fixed location, such as connectors, mounting holes, and so on. Select these parts.
3. Click the **Properties** button.
4. Select the **Glued** check box in the Properties dialog box to prevent Automatic Placement from moving the parts.
5. Click **Disperse Components** on the **Tools** menu to move all unglued components outside and around the board outline. Disperse arranges parts around the board outline based on height and length.

   Tip: Disperse ignores components that are part of a physical design reuse.
6. Click the **Component Keepout** button in the **Drafting** toolbar to create areas where parts should not be placed during automatic placement.

7. Use **Rotate** and **Flip** to arrange parts that should be placed at an angle or reside on the bottom side of the design.

Placing Parts Automatically

To place parts:

1. **Tools** menu > **Cluster Placement**.
2. Click the **Place Parts** button. To modify the automatic placement options, click the **Setup** button below the button. For more information see Place Parts Setup Dialog Box.
3. Click **Run**. A status dialog box appears to show placement progress. For more information see "Using the Cluster Placement Dialog Box."

Using the Cluster Manager Dialog Box

Use Cluster Manager to display and manage cluster members and unions. You can move cluster members and unions from one cluster to another and break, or delete, clusters. Cluster Manager works similarly to the Microsoft Windows Explorer; with it you can view items at the top level or at any level of the hierarchy.

The main elements of the Cluster Manager dialog box are two list boxes showing all clusters, unions, and components that exist in the design.

In this topic:

- Adding a Component to a Cluster
- Making Members of One Cluster Part of Another

Adding a Component to a Cluster

To add components to an existing cluster:

1. **Tools** menu > **Cluster Manager**.
2. In one list box, view the hierarchy of the cluster you want to add to.
3. In the other list box, select the components you want to add to the cluster.
4. Click the **Move** arrow.

Making Members of One Cluster Part of Another

To make members of one cluster part of another:

1. **Tools** menu > **Cluster Manager**.

2. View the hierarchy of the cluster to add to in either list.
3. In the other list, view the hierarchy of the cluster containing the members to merge.
4. Highlight the members you want to move.
5. Click the **Move** arrow.

Removing all members from a cluster empties the cluster, but does not delete the cluster name until you click OK to end the session. Before clicking OK, you can add members to this empty cluster by double-clicking it and repeating the above steps for the members to add.

Chapter 23
Working With Labels

You can create attribute, reference designator, and part type labels for components and jumpers. You can control the visibility of, justification, right-readability, and alignment of labels. When you create labels, they may not be visible. Turn on the visibility of labels using the Display Colors Setup dialog box, where you can set the color for reference designators, part type, and attribute labels.

Unlike free text, when you add a label, there is *no* invisible bounding rectangle around the label itself.

In this section:

Adding a New Part Label

Use the Add New Part Label dialog box to create attribute labels, part type labels, and reference designator labels for components or jumpers.

Tips:

- Reference designator is the only label available for use when you are creating labels for jumpers.
- If you don't set visibility information, default positions are used.

  See also: Label Defaults in the *Concepts Guide*
- Unlike free text, when you add a label, there is *no* invisible bounding rectangle around the label itself.

Procedure

1. **Select a part** > right-click > **Add New Label**

2. In the Attribute list, select the attribute you want. If you are creating labels for jumpers, Reference Designator is the only available attribute.

   Tip: Hidden attributes do not appear in the Attribute list unless the attribute was selected for the label before it was set as a hidden attribute.

3. The Value box lists the value of the selected attribute. Accept this value, or type a new one. This box is unavailable if you clicked Reference Designator or Part Type from the Attribute list, if the attribute is read-only, or if you open the Properties dialog box for different attribute types. However, if the labels you select belong to attributes of the same type, you can edit this box.

   Tips:

   - If the attributes have different values, the box is blank. You can type a new value in the box to apply it to all of the selected attribute labels and their parent objects.
   - Value is also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.

4. In the Show list, click the value you want to control the visibility of the label. You can choose to turn the label off, to display only the label name, to display only the label value, to display the name and value, or to display the full name and value (when labeling a structured attribute).

   Tip: Labels are invisible regardless of this setting unless you use the Display Colors Setup dialog box to change the color of labels to a color different from that of the background.

5. In the Font list, select the font you want to use.

   Tips:

   - Select stroke font or a system font.
   - For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.

6. In the Layer list, select the layer on which you want the part label.
7. In the Position and sizes area, select the **Relative to Component** check box to place the label at the X and Y location relative to the component or the jumper. If you clear this check box, the label is placed at the X and Y location relative to the design origin.
8. In the X,Y location boxes, type new values to move the part label to a specified location.
9. The Rotation box shows the current rotation angle of the label. Type a new rotation angle if you want to change the rotation of the label.
10. In the Size box, type the size you want.
11. For stroke font, type the line width you want.

12. Select the **Mirrored** check box if you want to flip the label. When Mirrored is checked, text is considered readable from the bottom side of the board.
13. In the Justification area, set the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change.

    Tips:

    - For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**.
    - Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.
14. The Right reading area controls whether labels are readable (from left to right, or from bottom to top if the label is rotated). Click the **None**, **Orthogonal**, or **Angled** button to indicate the direction of reading you want.
15. Click **OK**.

Selecting a Label

To select labels using the Selection Filter:

1. **Edit** menu > **Filter**.

    Result: The Selection Filter dialog box appears.
2. On the object tab, select the **Labels** check box.
3. Click **Close**.
4. Select the label.

Deleting a Label

To delete a label:

1. Select the label to delete. Use the selection filter if you need help selecting the label.
2. Press **Delete** or click **Delete** from the **Edit** menu.

Justifying a Label

You can justify free text and attribute, reference designator, or part type labels. You can set justification options when you create the label.

To justify text and labels after you create them:

1. Select the label or free text.
2. Either:
   - Right-click and click **Justify Horiz.** Then click **Left**, **Center**, or **Right**, and right-click and click **Justify Vert.** Then click **Up**, **Center**, or **Down**.

   Or

   - Right-click and click **Properties** to open the Properties dialog box for the object and set the appropriate justification setting.

You can also set justification options when you spin or move labels and text. For more information, see To Move Text and Labels and To Rotate an Object

Related Topics

Adding a New Part Label

Component Properties Dialog Box

Display Colors Setup Dialog Box

Labels in the PCB Decal Editor in the *Concepts Guide*

Modifying Part Label Properties

Use the Part Label properties dialog box to modify a label and to change the attribute the label displays.

Tip: If you select multiple labels, settings in this dialog box apply to all selected labels.

Procedure

1. Select a part label > right-click > **Properties**.
2. In the Attribute list, select the attribute you want. If you are creating or modifying labels for jumpers, Reference Designator is the only available attribute.

   Tip: Hidden attributes do not appear in the Attribute list unless the attribute was selected for the label before it was set as a hidden attribute.
3. The Value box lists the value of the selected attribute. Accept this value, or type a new one. This box is unavailable if you clicked Reference Designator or Part Type from the Attribute list, if the attribute is read-only, or if you open the Properties dialog box for labels of different attribute types. However, if the labels you select belong to attributes of the same type, you can edit this box.

   Tips:

   - If the attributes have different values, the box is blank. You can type a new value in the box to apply it to all of the selected attribute labels and their parent objects.

- Value is also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.

4. In the Show list, click the value you want to control the visibility of the label. You can choose to turn the label off, to display only the label name, to display only the label value, to display the name and value, or to display the full name and value (when labeling a structured attribute).

   Tip: Labels are invisible regardless of this setting unless you use the Display Colors Setup dialog box to change the color of labels to a color different from that of the background.

5. In the Font list, select the font you want to use.

   Tips:

   - Select stroke font or a system font.
   - For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.

6. In the Layer list, select the layer on which you want the label.
7. In the Position and sizes area, select the **Relative to Component** check box to place the label at the X and Y location relative to the component or the jumper. If you clear this check box, the label is placed at the X and Y location relative to the design origin.
8. In the X,Y location boxes, type new values to move the part label to a specified location.
9. The Rotation box shows the current rotation angle of the label. Type a new rotation angle if you want to change the rotation of the label.
10. In the Size box, type the size you want.
11. For stroke font, type the line width you want.
12. Select the **Mirrored** check box if you want to flip the label. When Mirrored is checked, text is considered readable from the bottom side of the board.
13. In the Justification area, set the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change.

    Tips:

    - For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**.
    - Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.

14. Click the **Component** button if you want access to the component associated with the part label.

15. The Right reading area controls whether labels are readable (from left to right, or from bottom to top if the label is rotated). Click the **None**, **Orthogonal**, or **Angled** button to indicate the direction of reading you want.
16. Click **OK**.

Modifying Labels using the Component Properties Dialog Box

You can modify labels from the Component Properties dialog box.

1. Select a component > right-click > **Properties**.
2. Click the **Labels** tab.
3. Click the label to modify from the **Label** list and click the **Label** button. The Part Label Properties dialog box appears.

For information on deleting labels, see “Deleting a Label.”

Chapter 24 Reusing Designs or Parts of Designs

In this chapter:

To Create a Physical Design Reuse

To create a physical design reuse:

1. Select the objects you want to include in the physical design reuse. You must select the entire object. Partial selections are not included. If you intend to create a reuse for cloning (using Make Like Reuse), set your Selection Filter to Pin Pairs and Parts.

   See also: Using the Selection Filter

   You can select routing objects in different ways. If you want to select traces to include in the physical design reuse, you can select trace segments, tacks, and trace corners.

2. Right-click and click **Make Reuse**.

   The selection is parsed for eligible items. Selected tacks and trace corners are replaced with appropriate trace segments. Selected pin pairs are replaced with trace segments, vias, and jumpers (if pin pair is fully or partially routed). Unrouted pin pairs are removed from the selection. Other ineligible items are also removed from the selection. Then the Make Reuse dialog box appears.

If none of the items you selected are valid physical design reuse elements, the message “Selection does not include items valid for inclusion into a reuse. Retry your selection” appears. Click **Report** to open a deselection report, in the default text editor, that lists items not valid for inclusion in a reuse. When you close the report, you return to the error message. Click **OK** and repeat Step 1.

3. In the Make Reuse dialog box, type a reuse type up to 255 characters long, in the **Reuse Type** box.

   Restriction: Illegal characters are slashes (\ /), colon (:), asterisk (*), question mark (?), quotation marks ("), less than and greater than signs (< >), and pipe (|).

4. Type a reuse name, up to 15 characters long, in the Reuse Name box. The default reuse name is based on the reuse type.

   Restriction: Illegal characters are comma (,), brackets ({ }), asterisk (*), period (.), and space.

5. If you want to save the physical design reuse to a file, click **Save to File**. Save it as a file to use the physical design reuse in other designs.

6. To view the Selection or Deselection report, click the appropriate button.

7. Click **OK**. The OK button remains unavailable until you provide a reuse type and name.

   If the reuse type is already assigned to another physical design reuse in the design, the message “A reuse type in the design already exists with the name XXX. Reuse type names cannot be duplicated in the design” appears. Click **OK** and specify another reuse type.

   If the reuse name is already assigned to another physical design reuse in the design, the message “Reuse name <name> already exists. Reuse names cannot be duplicated in the design” appears. Click **OK** and specify another reuse name.

   When the reuse name and type are accepted, the physical design reuse is created. If you selected the Save to File check box, the Reuse Save As dialog box appears.

8. Type a physical design reuse file name and specify a location in which to save the file. The default physical design reuse file name is the reuse type name with a .reu file extension. The default folder is \PADS Projects\Reuse.

9. Click **Save**. If you specified a reuse file name that is different from the reuse name used in the design, the message “Do you want to change reuse type name to XXX?” appears.

10. Click **Yes** to change the reuse type in the design to match the file name you specified, click **No** to ignore the change, or click **Cancel** to cancel the Save to File process.

To Add a Physical Design Reuse

Once you create a physical design reuse, you can add it to other designs. You can also add a copy of a reuse that already exists in the design.

Requirement: To add or copy a reuse, you must be in ECO mode; therefore, all components, pin pairs, and nets added to the design are recorded in the .eco file using standard ECO commands.

Tip: You cannot add a reuse to a design if the design is in default layer mode and the reuse to add is in increased layer mode. Use the Layers Setup dialog box to change the design to increased layer mode.

To add the first instance of a physical design reuse to a design:

1. Click the **ECO Toolbar** button.
2. Turn Design Rules Checking off by typing **DRO** and pressing Enter. You must turn DRC off to add or copy a reuse. If DRC is not turned off, a prompt appears that allows you to turn DRC off.
3. Click the **Add Reuse** button on the **ECO** toolbar. The Add Reuse dialog box appears.
4. Select the reuse file to add and click **Open**. The Reuse Properties dialog box appears.

   If the reuse file you select to add already exists in the design, the message “Reuse type XXX already exists in the design. OK to make a copy of this reuse type?” appears. Click **OK** to add a copy of the selected physical design reuse from the design or click **Cancel**.
5. Set the reuse name and reference designator preferences.
6. Click **Net Properties**. The Net Properties dialog box appears.
7. Resolve net conflicts, if any, and click **OK**. You return to the Reuse Properties dialog box.
8. Click **OK**. The reuse file is compared against the current design to detect possible reference designator, layer, decal, and netname conflicts, and to detect other errors and warnings.

   See also: Adding a Physical Design Reuse in the *Concepts Guide*

   The results of this comparison are recorded in the Layout.err report file in the *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs* folder.
9. If errors are found, the message “Adding this reuse to the design resulted in xxx errors and xxx warnings. The Add Reuse command was aborted. Show report file?” appears. Click **Yes** to view the report file in the default text editor; otherwise click **No** and return to step 3.

If errors are not found, but warnings are, the message “Adding this reuse to the design resulted in 0 errors and xxx warnings. Show report file?” appears. Click **Yes** to view the report file in the default text editor; otherwise click **No** and proceed to step 10.

If no errors or warnings are found, the message “Reuse added without warnings or errors. Show report file?” appears. Click **Yes** to view the report file in the default text editor; otherwise click **No** and proceed to Step 10.

10. When all physical design reuse elements are successfully added to the design, the physical design reuse dynamically attaches to the pointer. Click to indicate the location of the physical design reuse.

Result

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

Related Topics

To Add an Existing Reuse in Verb Mode

Adding a Physical Design Reuse in the *Concepts Guide*

Adding an Existing Reuse

You can add a copy of an existing physical design reuse to your design in one of three ways: in Verb mode, in Object mode, or by using copy and paste.

To Add an Existing Reuse in Object Mode

Requirement: You must be in ECO mode to add or copy a reuse.

Procedure

1. Click the **ECO Toolbar** button.
2. Turn Design Rules Checking off by typing **DRO** and pressing Enter. You must turn DRC off to add or copy a reuse.
3. Select an existing physical design reuse in the design as described in “To Select a Physical Design Reuse.”
4. Click the **Add Reuse** button on the **ECO** toolbar. The Reuse Properties dialog box appears.
5. Modify the physical design reuse properties as described in tabularTo Add a Physical Design Reuse.”

6. Click **OK**. The physical design reuse dynamically attaches to the pointer. Click to indicate the location of the copied physical design reuse.

Result

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

To Add an Existing Reuse in Verb Mode

Requirement: You must be in ECO mode to add or copy a reuse.

Procedure

1. Click the **ECO Toolbar** button.
2. Turn Design Rules Checking off by typing **DRO** and pressing Enter. You must turn DRC off to add or copy a reuse.
3. Click the **Add Reuse** button from the **ECO** toolbar. The Add Reuse dialog box appears.
4. Click the reuse file you want to add and click **Open**. The Reuse Properties dialog box appears.

   If the reuse type already exists in the current design, the message “Reuse Type XXX already exists in the design. OK to make a copy of this reuse type?” appears.
5. Click **OK** to continue, or click **Cancel** to abort the Add Reuse process.
6. Modify the physical design reuse properties as described in “To Add a Physical Design Reuse.”
7. Click **OK**. The physical design reuse dynamically attaches to the pointer. Click to indicate the location of the copied physical design reuse.

Result

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

To Copy and Paste an Existing Reuse

Requirement: You must be in ECO mode to add or copy a reuse unless you are pasting a physical design reuse that contains only drafting items

Procedure

1. Click the **ECO Toolbar** button.

2. Turn Design Rules Checking off by typing **DRO** and pressing Enter. You must turn DRC off to add or copy a reuse.
3. Select an existing physical design reuse in the design as described in To Select a Physical Design Reuse.
4. Click **Copy** from the **Edit** menu.

   Tip: You can only copy one reuse at a time.
5. Click **Paste** from the **Edit** menu. The Reuse Properties dialog box appears.
6. Modify the physical design reuse properties as described in To Add a Physical Design Reuse
7. Click **OK**. A copy of the physical design reuse dynamically attaches to the pointer. Click to indicate the location of the copied physical design reuse.

Result

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

Tip: When you copy a physical design reuse and paste it into a different design, the reuse file is compared against the current design to detect possible reference designator, layer, decal, and netname conflicts.

See also: Adding a Physical Design Reuse in the *Concepts Guide*

Using the Make Like Reuse Command

Make Like Reuse clones a selected physical design reuse using existing components and their logical interconnects as the elements for the physical design reuse. All other physical design reuse elements; such as traces, vias, coppers, 2D lines, and text are created.

To Make a Like Reuse in Object Mode

You can add a physical design reuse in one of the three ways: in Verb mode, in Object mode, or by using the shortcut menu command.

Procedure

1. Turn Design Rules Checking off by typing **DRO** and pressing Enter. You must turn DRC off to make a like reuse.
2. Select an existing physical design reuse in the design as described in "To Select a Physical Design Reuse."
3. Click the **Design Toolbar** button. The Design toolbar appears.

4. Click the **Make Like Reuse** button from the **Design** toolbar. If a match is found, the new physical design reuse dynamically attaches to the pointer. If no match is found, the message “No match found in unrouted components for this reuse definition” appears.

   Tip: You cannot add a physical design reuse if the design is in default layer mode and the reuse to add is in increased layer mode. Use the Layers Setup dialog box to change the design to increased layer mode.

5. Click to indicate a location for the new physical design reuse.

Result

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

To Make a Like Reuse in Verb Mode

You can add a physical design reuse in one of the three ways: in Verb mode, in Object mode, or by using the shortcut menu command.

Procedure

1. Turn off Design Rules Checking by typing **DRO** and pressing Enter. You must turn DRC off to make a like reuse.

2. Select the group of components you want to include in the new reuse. If you do not select a specific group of components, all components are included in making the like reuse. The group of components you select can be larger than required for the reuse; any useless components are ignored. You cannot select both the reuse and the group of components. If you want to select a particular group of components, save your reuse definition before making a like reuse.

3. Click the **Design Toolbar** button. The Design toolbar appears.

4. Click the **Make Like Reuse** button on the **Design** toolbar. The Open Reuse File dialog box appears.

5. Click a reuse file and click **Open**. If a match is found, the new physical design reuse is attached to the pointer. If no match is found, the message “No match found in unrouted components for this reuse definition” appears.

   Tip: You cannot add a reuse to a design if the design is in default layer mode and the reuse to add is in increased layer mode. Use the Layers Setup dialog box to change the design to increased layer mode.

6. Click to indicate the location of the new physical design reuse.

Result

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

To Make a Like Reuse Using the Shortcut Menu Command

You can add a physical design reuse in one of the three ways: in Verb mode, in Object mode, or by using the shortcut menu command.

Procedure

1. Turn Design Rules Checking off by typing **DRO** and pressing Enter. You must turn DRC off to make a like reuse. If DRC is not turned off, a prompt appears that allows you to turn DRC off.
2. Select an existing physical design reuse in the design as described in "To Select a Physical Design Reuse."
3. Right-click and click **Make Like Reuse**. If a match is found, the new physical design reuse dynamically attaches to the pointer. If no match is found, the message "No match found in unrouted components for this reuse definition" appears.

   Tip: You cannot add a physical design reuse if the design is in default layer mode and the reuse to add is in increased layer mode. Use the Layers Setup dialog box to change the design to increased layer mode.
4. Click to indicate a location for the new physical design reuse.

Result

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

To Select a Physical Design Reuse

To select a physical design reuse:

1. Select any element in the physical design reuse.
2. Right-click and click **Select Reuse**. The physical design reuse is selected.

Tip: You can set the Selection Filter to *Reuse* or the Find dialog box to *Reuse type* to select physical design reuse blocks.

See also: Using the Selection Filter, Find Dialog Box

Modifying a Physical Design Reuse Block

To Edit a Physical Design Reuse Definition

After you open a physical design reuse, you can then edit its definition and update the reuse file.

Tip: The reuse definition is the master copy of the physical design reuse that is saved to a file. The saved version of the physical design reuse is the version you should use in other designs. All resulting instances of the physical design reuse are based on this file.

To edit a physical design reuse:

1. Open the physical design reuse file:
   a. **File** menu > **Open**.
   b. Click **PADS Layout Reuse Files** (*.reu) as the file type.
   c. Navigate to the appropriate folder and click the reuse file to open.
   d. Click **Open**.
2. If necessary, select the physical design reuse. The physical design reuse should be selected after you open it.
3. If necessary, break the physical design reuse by right-clicking and clicking **Break Reuse**. You can edit the design rules without breaking the physical design reuse.
4. Edit the physical design reuse, as necessary.
5. Recreate the physical design reuse by selecting all of the components and right-clicking and clicking **Make Reuse**. The Make Reuse dialog box appears.

   See also: Reusing Designs or Parts of Designs
6. Click **Save to File**.
7. In the Reuse Save As dialog box, click the name of the reuse file you just modified and click **Save**. The message “xxx.reu already exists. Do you want to replace it?” appears.
8. Click **Yes** to overwrite the existing file. The physical design reuse definition is updated.

Modifying Physical Design Reuse Properties

You can modify a physical design reuse, but you can't modify the elements within the physical design reuse. More modification options are available to you in ECO mode. When in ECO mode, any component or netnames you modify in the Reuse Properties dialog box are recorded in the .eco file using standard ECO commands.

To edit physical design reuse properties:

1. Select a physical design reuse > right-click > **Properties**.
2. Modify the location, rotation, glued status, and reuse name, if necessary.
3. Modify the reference designator preferences or net properties, if necessary.
4. Click **OK**.

   If you changed the reference designator preferences or net properties, the component elements and private nets are renamed.

To Reset the Origin of a Physical Design Reuse

Every physical design reuse has an origin. The default origin is the component pin with the lowest X and Y values. The origin marker is square, rather than round, to differentiate it from the design and component origin markers. The physical design reuse origin marker is visible only when you reset the origin.

To reset a physical design reuse origin:

1. Select the physical design reuse whose origin you want to reset as described in "To Select a Physical Design Reuse."
2. Right-click and click **Reset Origin**. The reuse origin marker appears.
3. Click to indicate the new position of the origin. A message appears indicating the coordinates of the new origin and asks if you want to use this as the origin.
4. Click **Yes** to use the new origin. Click **No** to choose another origin.

To Save a Physical Design Reuse

You can save a physical design reuse to a file at any time. The saved version of the physical design reuse is called the reuse definition.

See also: To Select a Physical Design Reuse

To save a physical design reuse:

1. Select a physical design reuse > right-click > **Save to File**.
2. Type a reuse type name and a path in which to save the selected physical design reuse. The default physical design reuse file name is the reuse type name with a .reu file extension. The default folder is \PADS Projects\Reuse.
3. Click **Save**. If you specified a reuse file name, the message "Do you want to change reuse type name to XXX?" appears.
4. Click **Yes** to change the reuse type file name, click **No** to ignore the change, or click **Cancel** to abort the Save to File process.

If the name in the Reuse Type box is already assigned to another physical design reuse in the reuse folder, the message “XXX.reu already exists. Do you want to replace it?” appears.

5. Click **Yes** to overwrite the reuse file or Click **No** to specify another type.

To Break a Physical Design Reuse

The Break Reuse command disassembles the physical design reuse and returns its elements to normal design data. When you break a physical design reuse, elements are returned to their pre-reuse state; therefore, route protection and other settings are maintained. You do not need to be in ECO mode to breaking a physical design reuse.

To break a physical design reuse:

1. Select the physical design reuse you want to break as described in “To Select a Physical Design Reuse.”
2. Right-click and click **Break Reuse**. The message “OK to Break Reuse(s)?” appears.
3. Click **Yes** to break the physical design reuse.

Tips:

- You must break a physical design reuse before you can edit the elements within it.
- Reuses are automatically broken when you import an .eco file.

To Delete a Physical Design Reuse

To delete a reuse, you must be in ECO mode; therefore, all components, pin pairs, and nets deleted to the design are recorded in the .eco file using standard ECO commands.

Requirement: You must be in ECO mode to delete a reuse.

To delete a physical design reuse:

1. Click the **ECO Toolbar** button.
2. Select the physical design reuse you want to delete.

   See also: “To Select a Physical Design Reuse“
3. Click **Delete** from the **Edit** menu, or press **Delete**. The confirmation message appears.
4. To maintain routes connected to the physical design reuse, click to clear **Delete Routes**.

Tip: It is not always possible to maintain traces connected to physical design reuse components. Delete Routes is clicked by default, meaning routes are deleted with the physical design reuse.

5. Click **OK** to delete the physical design reuse.

Tip: You can't delete physical design reuses that contain glued components or protected routes.

To Move a Physical Design Reuse

When you move a physical design reuse, traces are stretched, similarly to moving a group.

To move a physical design reuse:

1. Turn Design Rules Checking off by typing **DRO** and pressing Enter. You must turn DRC off to move a physical design reuse.
2. Select the physical design reuse as described in "To Select a Physical Design Reuse."
3. Right-click and click **Move**.
4. Click to indicate a new location for the physical design reuse. You can right-click and click **Cancel** to cancel the move.

Creating a Physical Design Reuse Report

You can create a report that contains the following information of the selected physical design reuse: name, type, and elements.

Procedure

1. Select the physical design reuse.

   See also: To Select a Physical Design Reuse

2. Right-click and click **Report Contents**.

Result

The report is written to C:\PADS Projects\rules.rep and is displayed by the default text editor.

Chapter 25
Drafting Operations

Creating a Drafting Object

All drafting objects are created in the same manner. These instructions apply to creating 2D lines, copper, copper cut outs, copper pour areas, copper pour cut outs, board outlines, board outline cut outs, keepouts, plane areas, and plane area cut outs.

Procedure

1. After you click a button on the Drafting toolbar, right-click to set any values before creating the drafting object.
2. Right-click and choose a shape to draw.
3. Draw the:
   - Circle
   - Polygon or Path
   - Rectangle
   - Chamfered path (copper only)

Setting Values Before Creating a Drafting Object

Use the Drafting toolbar to add drafting objects, such as copper shapes, keepouts, and non-electrical drawn items generally not associated with placement and routing. After you click the button for the type of item to add, but before you place the drafting object, you can set various values.

Procedure

1. Type **W<nn>**, where **<nn>** is the width value, or right-click and click **Width** to specify the width of the drafting object boundary.

   Tip: The default widths for each object type are defined using the Drafting / Text and Lines page of the Options dialog box.

2. Right-click and click **Layer**. Choose one of the following:
   - Type **L<n>**, where **<n>** is a layer number, or click **Layer** to specify the layer where you want to place the Drafting object.

- Specify the layer using the Layer box on the Toolbar.

3. Right-click and click **Auto Miter** to miter all added corners. Define the miter type and size in the Design page of the Options dialog box.
4. Right-click and click **Orthogonal** to add strictly horizontal or vertical segments.
5. Right-click and click **Diagonal** to add 45-degree segments.
6. Right-click and click **Any Angle** to add segments at any angle.

Related Topics

Creating a Circle Drafting Object

Creating a Polygon or Path Drafting Object

Creating a Rectangle Drafting Object

Deleting a Drafting Segment or Object

Creating a Polygon or Path Drafting Object

Use the Drafting toolbar to add drafting objects such as copper shapes, keepouts, and non-electrical drawn items generally not associated with placement and routing.

Procedure

1. Click the button for the type of item to add: 2D Line, Copper, Cut Out, Pour Keepout, or Board Outline. The add command for that object type attaches to your pointer.
2. Right-click and set your drafting values.
3. To start the drafting object and finish each segment or place each corner, use one of the following:
    - Locate each coordinate with the pointer, and click to start and indicate consecutive corners.
    - Type **S<x>, <y>,** where <x> and <y> are the coordinate values for the starting point and consecutive corners.
    - Type **SR<x>, <y>** if you want to specify relative coordinate values for each point and corner.

    Tips:

    - Right-click and click **Add Arc** to add an arc instead of a straight-line segment.
    - Press **BackSpace** to remove the last corner.
    - Press **Esc** to cancel the operation.

4. Double-click or right-click and click Complete to end the shape. Polygons are automatically closed; paths are terminated at the last-defined corner.

Related Topics

Creating a Circle Drafting Object

Creating a Rectangle Drafting Object

Deleting a Drafting Segment or Object

Setting Values Before Creating a Drafting Object

Creating a Circle Drafting Object

Use the Drafting toolbar to add drafting objects such as copper shapes, keepouts, and non-electrical drawn items generally not associated with placement and routing.

Procedure

1. Click the button for the type of item to add: 2D Line, Copper, Cut Out, Pour Keepout, or Board Outline. The add command for that object type attaches to your pointer.

2. Right-click and set your drafting values.

3. Click to indicate the location of the circle's center.

   Alternative: Instead of using the pointer, you can type the coordinates using the following methods.

   - Type **S<x>, <y>,** where <x> and <y> are the coordinate values for the starting and ending point.

   - Type **SR<x>, <y>** if you want to specify relative coordinate values for the starting and ending point.

   Tip: Press **Esc** to cancel the operation.

4. Click to indicate the circle's radius to complete the circle definition.

Related Topics

Creating a Polygon or Path Drafting Object

Creating a Rectangle Drafting Object

Setting Values Before Creating a Drafting Object

Creating a Rectangle Drafting Object

Use the Drafting toolbar to add drafting objects such as copper shapes, keepouts, and non-electrical drawn items generally not associated with placement and routing.

Procedure

1. Click the button for the type of item to add: 2D Line, Copper, Cut Out, Pour Keepout, or Board Outline. The add command for that object type attaches to your pointer.
2. Right-click and set your drafting values.
3. Click to indicate one corner of the rectangle.

   Alternative: Instead of using the pointer, you can type the coordinates using the following methods.

   - Type **S<x>, <y>,** where <x> and <y> are the coordinate values for the starting and ending point.
   - Type **SR<x>, <y>** if you want to specify relative coordinate values for the starting and ending point.

   Tip: Press **Esc** to cancel the operation.
4. Click to indicate the location of the diagonally opposite corner.

Related Topics

Creating a Circle Drafting Object

Creating a Polygon or Path Drafting Object

Deleting a Drafting Segment or Object

Setting Values Before Creating a Drafting Object

Text

In this section:

- Adding Free Text
- Modifying Text Properties
- To Mirror Text
- To Move Text and Labels

Adding Free Text

You can add single lines of free text (text not belonging to an object) to a design. When you add text, there is an invisible bounding rectangle or outline box around the text itself. Type the X modeless command to toggle the outline box on or off.

Tip: To add multiple lines of text to a design, see Embedding a Text Document.

Procedure

1. **Drafting Toolbar** button > **Text** button.
2. In the Add Free Text dialog box, type the text string you want to use in the **Text** box.

   Tip: There is a maximum of 128 characters per text string.
3. In the Font list, select the font you want to use.

   Tips:

   - The default font name and style appear in the Font box when the dialog box opens.
   - Select a stroke font or a system font.
   - For system fonts, you can also click a font style button or any combination of buttons: **B** for bold, **I** for italic, or **U** for underlined.
4. In the Layer list, select the layer on which you want the text.
5. In the X,Y location boxes, type values to move the text string to a specified location. If not specified, the text string attaches to your pointer for placement.
6. The Rotation box shows the current orientation of the text string. Type a new value to change the degree of rotation.
7. The Size box shows the current size used for display or CAM output of the text string. Type a new value to change the size.
8. For stroke font, type the line width you want.
9. Select the **Mirrored** check box if you want to flip the text. When Mirrored is checked, text is considered readable from the bottom side of the board.
10. In the Justification area, set the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change.

    Tips:

    - For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**.
    - Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically,** and then clicking **Up**, **Center**, or **Down**.
11. Click **OK** to close the dialog box.

    Tip: If you did not type coordinates to place the text, the text dynamically attaches to the pointer. Click to indicate the location for the text. After you place the text, the Add Free Text dialog box reappears so you can create additional strings.

Recommendation: Place free text and attribute values on the Silkscreen Top layer to avoid DRC violations or shorts.

Related Topics

Modifying Text Properties

Combining Line and Text Objects

To Mirror Text

To Move Text and Labels

Modifying Text Properties

Use the Text Properties dialog box to modify free text. You can change the font, font style, layer assignment, orientation, rotation, size, line width, if it is mirrored, and justification. You can also access the parent object if the text string is combined with a drafting object.

To modify text properties:

1. Select a text string > right-click > **Properties**.
2. The Text box displays the selected text string. Modify the existing text string, or type a new text string.
3. In the Font area, select the font you want to use.

   Tips:

   - Select stroke font or a system font.
   - For system fonts, you can also click a font style button or any combination of buttons: **B** for bold, **I** for italic, or **U** for underlined.
4. In the Layer area, select the layer on which you want the text.
5. In the X,Y location boxes, type new values to move the text string to a specified location.
6. The Rotation box shows the current orientation of the text string. Type a new value to change the degree of rotation.
7. The Size box shows the current size used for display or CAM output of the text string is shown. Type a new value to change the size.
8. For stroke font, type the line width you want.
9. Select the **Mirrored** check box if you want to flip the text. When Mirrored is checked, text is considered readable from the bottom side of the board.
10. In the Justification area, set the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change.

Tips:

- For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**.
- Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.

11. Click the **Parent** button if you want to access the parent object for a text string that is combined with a drafting object.
12. Click **OK**.

Tips:

- If you select another text object before closing the dialog box, the text information is updated.
- Several options in this dialog box are unavailable if the text is part of a physical design reuse.

Related Topics

Text

To Mirror Text

To Move Text and Labels

To Mirror Text

To mirror text objects:

1. Select the object.
2. Right-click and click **Mirror**.

Related Topics

Text

Modifying Text Properties

Combining Line and Text Objects

To Move Text and Labels

To Move Text and Labels

To move text and labels:

1. Select a text or label > right-click > **Move**.

Result: The object attaches to the pointer.

2. You can open the Properties dialog box, rotate, spin, mirror, or justify the selected text or label while it is attached to your pointer.
3. Click to complete the move.

You can also move text and labels with drag and drop operations or using the Move button in the Design toolbar. You still have the same editing abilities while moving the text or label. You can use the arrow keys while in Move Verb mode, but you do not have the same editing abilities.

Tips:

- If you move a text object that is combined with other drafting objects, all the combined objects move.
- If you click Drag and Drop in the Global / General page of the Options dialog box, the text object does not attach to your pointer when you release the left button.
- You can't modify text that is part of a physical design reuse. If you try to, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the operation.

Related Topics

Text

Modifying Text Properties

Combining Line and Text Objects

To Mirror Text

Modifying Copper Chamfered Paths Properties

You can select the copper chamfered paths of a net to change its properties.

1. Select a net in the design.
2. Right-click and click Select Chamfered Paths.
3. Right-click and click Properties.

Related Topics

Modifying Drafting Object Properties

Modifying Drafting Objects

In this section:

To Move a Drafting Object

Procedure

1. Select the drafting object to move.
2. Move the object by positioning the pointer over your selection, clicking, moving the mouse to drag, and releasing. The drafting object remains attached to the pointer.
3. You can open the Properties dialog box, rotate, or spin the selected drafting object while it is attached to your pointer. You can also temporarily disable On-line DRC for this move by right-clicking.
4. Click to complete the move.

Tips:

- If you move a drafting object that is combined with other drafting objects, all the combined objects move.
- If you click the Move button in the Design toolbar, a click attaches the drafting object to your pointer.
- If you click Drag and Drop in the Global / General page of the Options dialog box, the drafting object is not attached to your pointer when you release the left button.
- If you click Move By Origin in the Design page of the Options dialog box, the drafting object attaches to the pointer at its origin.
- You can't modify, move, or delete a polygon of any type (2D line, copper, or pour) that is part of a physical design reuse. You can, however, flood pour outlines for copper pour and split planes. If you try to otherwise modify a polygon, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the operation.

Modifying Drafting Edge Properties

1. To select an edge, click on it once.
2. Right-click and click **Properties**. The Drafting Edge Properties dialog box opens; showing you what type of drafting object the item is and rules information.

Tip: Several of the options in this dialog box are unavailable if the edge is part of a physical design reuse.

Modifying Drafting Corner Properties

To modify drafting corner properties:

1. Select a drafting corner > right-click > **Properties**.
2. In the X and Y boxes, type the X and Y location of the drafting corner.
3. Click **Net** to modify settings for the net to which the drafting corner belongs.
4. Click **Parent** to modify drafting object to which the corner belongs.

Tip: Several of the options in this dialog box are unavailable if the corner is part of a physical design reuse.

Related Topics

Modifying Net Properties

Modifying Drafting Object Properties

To Delete a Corner

To Move a Corner

Modifying Drafting Object Properties

You can select and edit drafting shapes in pieces or as whole items. The properties you can modify depend on whether you select a corner of the drafting object, an edge of the drafting object, or the whole, or parent object of a 2D line, copper, or text.

Exception: Several of the options in this dialog box are unavailable if the object is part of a physical design reuse.

In this section:

- Converting Shapes
- Changing Line Widths
- Scaling

- Setting the Arc Approximation Error
- Rotating
- Setting Object Options
- Changing Net Properties of Assigned Objects
- Changing Layers
- Assigning Nets
- Assigning Nets by Clicking Design Objects
- Setting Keepout Restrictions

Converting Shapes

You can convert a drafting object to another shape. In the Layout Editor, you can convert any drafting object to a 2D line, board outline, board cut out, copper, copper cut out, keepout, copper pour, plane area, or plane area cut out. In the PCB Decal Editor, you can convert any drafting object to a 2D line, copper, copper cut out, or keepout.

Tip: Convert shapes without scaling, using a Scale Factor of 1 to maintain the shape size.

Restrictions:

- If there is no board outline in the design, the Board Cut Out type does not appear in the Drafting Properties dialog box Type list.
- If there is a board outline in the design, the Board Outline option does not appear in the dialog box.

To Convert to a new shape type:

1. Select a drafting item > right-click > **Properties**.
2. In the Type list, select a new shape type.

Changing Line Widths

You can change the line width of drafting object outlines and fill lines if it applies.

1. Select a drafting item > right-click > **Properties**.
2. In the Width box, type a value.

   Exceptions:

   - Some drafting object line widths can't be changed.

- Like any other line width, the chamfered path does not appear as the correct width if the Minimum display width option for the design is larger than the Polygon outline width of the path. The path appears more narrow by the value of the Polygon outline width.

Scaling

Create a scale model of the objects on one or more nonelectrical layers that won't be plotted in CAM. We recommend using a 1:10 scale for the model. The scale model lets you work with arcs in a coordinate range that PADS Layout allows.

To scale:

1. Select a drafting item.
2. If necessary, right-click and click Set Origin to set the origin of the scale model.
3. Right-click, and click **Properties**.
4. In the Properties dialog box, set scale options and click **OK**.

   Tip: Enter a floating-point number greater than 0. Fractions are not supported.

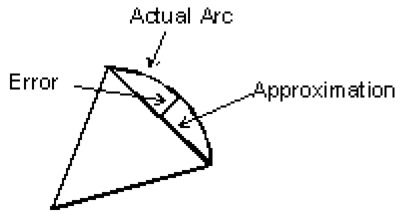
   Exceptions:

   - If the new shape type is copper pour and the layer is a split/mixed plane layer, then the new shape is a plane area shape and the Assign Net to Selected Polygon dialog box appears. Click the net to assign to the scaled shape and click OK. The shape is scaled.
   - If scaling changes a dimension, select the dimension, right-click, and click **Reset Measurement** to update the dimension measurement.

Setting the Arc Approximation Error

- Select a drafting item > right-click > **Properties**.

In the Arc Approximation Error box, type the allowable approximation error for scaling arcs. Arcs are converted to a set of straight-line segments that approximate the arc. The approximation error is the perpendicular distance from the approximation segment to the actual arc.



Rotating

You can change the rotation of drafting objects.

1. Select a drafting item > right-click > **Properties**.
2. In the Rotation box, type the rotation angle of the drafting shape.

   Exception: Some drafting object line widths can't be changed.

Filling a Shape with Solid Copper

Copper is filled using the settings of the Copper Hatch Grid and the Drafting Default width. You can ignore these settings and fill a shape with solid copper. When you create chamfered paths, they are created with solid copper.

- Select the **Solid Copper** check box.

Tip: The Solid Copper check box forces copper to use trace clearance rules.

Setting Object Options

You can access additional options for certain drafting objects.

1. Select a drafting item > right-click > **Properties**.
2. Click the **Options** button.

Changing Net Properties of Assigned Objects

You can access the Net Properties of an assigned object.

1. Select a drafting item > right-click > **Properties**.
2. Click the **Net** button to modify settings for the net to which the drafting object belongs.

Exception: You can assign clearance rules for copper on the net level only: group and pin pair assignments are invalid.

Changing Layers

You can change the layer on which the drafting object exists.

1. Select a drafting item > right-click > **Properties**.
2. In the Layer list, select the new layer.

Tip: Moving assigned copper to a nonelectrical layer removes the connection from the copper shape. Place the copper back on an electrical layer to reassign the connections.

Assigning Nets

You can assign nets to electrical drafting objects.

1. Select a drafting item > right-click > **Properties**.
2. In the Net Assignment area, select a netname from the Net list and then click **Apply** to assign an existing netname to a copper shape.

Assigning Nets by Clicking Design Objects

You can assign nets to electrical drafting objects by clicking objects in the design. You can click objects such as a pin, via, trace, net, copper, or unroute to assign the netname of the object to the copper shape.

1. Select a drafting item > right-click > **Properties**.
2. Click the **Assign Net by Click** button and then select a design object.
3. Click the **Assign Net by Click** button again to cancel the object selection and return to the dialog box.

Alternative: If a drafting object does not yet have a net assigned, in the design, right-click and click Assign Net by Click. You do not have to open the Properties dialog box.

Setting Keepout Restrictions

You can select restrictions for keepout areas.

Tip: When defining keepouts in the PCB Decal Editor, you can also assign keepouts to an <Opposite Side>. You can’t do this in the Layout Editor.

- Select a drafting item > right-click > **Properties**.
    - Component Height restricts components with heights greater than the specified height. This check box is unavailable unless you turn on the Placement check box.
    - Component Drill restricts components that contain drilled through holes.

- Click **Select All** to select all restrictions except Component Height.
- Copper Pour and Plane Area restricts copper pours from flooding and split/mixed planes from plane connection.

Restrictions:

- If you set a layer before assigning restrictions, there are restrictions that are not available.
- Placement, Component Height, and Component Drill check boxes are unavailable in the PCB Decal Editor.

Related Topics

Modifying Net Properties

Using Keepouts

Selecting Drafting Objects

To Move a Miter

You can expand or contract a miter or arc on a drafting segment with the Move Miter command.

1. Select a miter segment or arc > right-click > **Move Miter**.

   Result: The object dynamically attaches to your pointer.

2. Move the pointer to expand or contract the object.
3. Click to indicate the new miter position to finish.

Indicate a location outside the point of intersection of the two lines to convert the arc or miter back to a corner.

Tips:

- Use the Stretch command to modify an arc or miter on a route.
- You can't modify, move, or delete a polygon of any type (2D line, copper, or pour) that is part of a physical design reuse. You can, however, flood pour outlines for copper pour and split planes. If you try to otherwise modify a polygon, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the operation.

Related Topics

Pulling an Arc from a Drafting Segment/Corner

Pulling an Arc from a Drafting Segment/Corner

Use the Pull Arc command to convert a drafting segment or corner into an arc. The starting points of the drafting segment become the start and stop angle for the new arc.

Procedure

1. Select a segment or corner > right-click > **Pull Arc**.
2. The segment or corner attaches to your pointer and changes to an arc.
3. Click to indicate the new position for the arc.

Use the Move Miter command to modify the radius of a created arc or miter.

Tip: You can't modify, move, or delete a polygon of any type (2D line, copper, or pour) that is part of a physical design reuse. You can, however, flood pour outlines for copper pour and split planes. If you try to otherwise modify a polygon, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the operation.

See also: To Create Miters, To Create Arcs

Deleting a Drafting Segment or Object

To delete drafting segments or objects:

1. Select the segment or object. See Selecting Drafting Objects
2. Click the **Delete** button or on the **Edit** menu click Delete.

Tip: You can't modify, move, or delete a polygon of any type (2D line, copper, or pour) that is part of a physical design reuse. You can, however, flood pour outlines for copper pour and split planes. If you try to otherwise modify a polygon, the message “Reuse elements cannot be modified. Break the reuse first” appears. Click OK to cancel the operation.

Related Topics

Creating a Circle Drafting Object

Creating a Polygon or Path Drafting Object

Creating a Rectangle Drafting Object

Setting Values Before Creating a Drafting Object

Deleting an Item

Use Delete to remove a selected item from the design.

1. Select the item you want to delete.

2. Click **Delete** from the **Edit** menu.

The item is removed from the design.

Combining Drafting Objects

In this section:

- Combining Line and Text Objects
- Exploding Combined Objects
- Uncombining Drafting Objects

Combining Line and Text Objects

You can combine text objects and line objects to form one group. When you combine objects, you can select and move them as one item and save it to the library. When you move an object that is combined with other objects, all the combined objects move. You can use Combine in the Layout Editor and in the Decal Editor.

Procedure

1. Use Shift-click to select the entire drafting shape that you want to combine.
2. When the shape is highlighted, use Ctrl+click to select the text to combine.
3. Right-click and click **Combine**. You can't combine text objects without including a drafting object.

Result

Once you combine objects, selection characteristics change for each object. When you Shift-click a shape, the whole shape and the text are selected for action. You can reposition the combined objects or save them to the library as one unit. If you select the text, however, only text is highlighted for editing or repositioning.

You can remove an object from the combination using **Uncombine** from the shortcut menu. You can uncombine the entire combination using **Explode** from the shortcut menu.

Related Topics

Exploding Combined Objectss

Uncombining Drafting Objects

Exploding Combined Objects

Use Explode to remove all items from a combined object. Combined objects are made of multiple drafting and text objects that were combined using the Combine command. You can use Explode in the Layout Editor and in the PCB Decal Editor.

To remove all objects from a combination:

1. Select the combination.
2. Right-click and click **Explode**.

Related Topics

Combining Line and Text Objects

Uncombining Drafting Objects

Uncombining Drafting Objects

Use Uncombine to remove a single item from a combined object. Combined objects are made of multiple drafting and text objects that have been combined with the Combine command. You can use Uncombine in the Decal Editor.

To remove any object from a combination:

1. Select the combination.

   Tip: To select a combination, select any part of the combination, right-click, and click **Select Shape**.
2. Right-click and click **Uncombine**.
3. Select the object.

A quicker way to remove a *text* object from a combination is:

1. Select the text object.
2. Right-click and click **Uncombine**.

Related Topics

Combining Line and Text Objects

Exploding Combined Objects

Joining and Closing 2D Lines and Copper Shapes

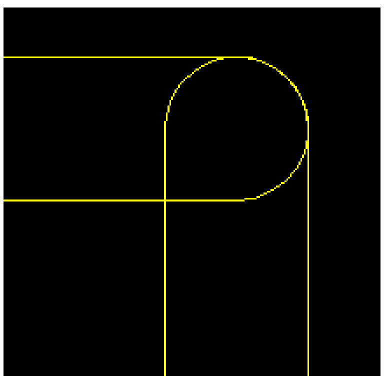
Use Join and Close to connect 2D lines and copper shapes to create single objects that you can then change the properties of; for example, you can create a closed 2D line and then change its properties to copper pour.

Tip: These commands are especially useful for after you've imported a dxf file.

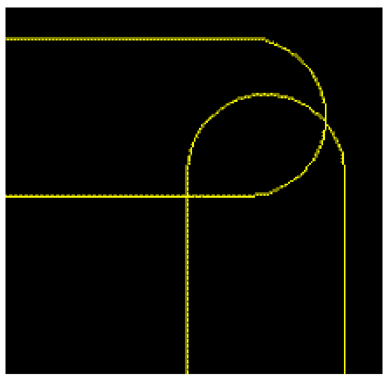
You can use Join and Close when the line ends are "near enough": meaning: touching, crossing slightly, or perfectly matched. If the lines are not touching or crossed too far, Join and Close will not work.

Examples of Near Enough

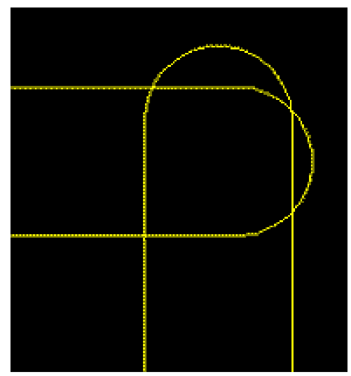
Join and Close will work on any of the following examples:



Perfect match



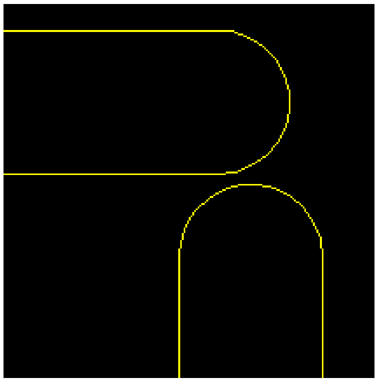
Lines touch



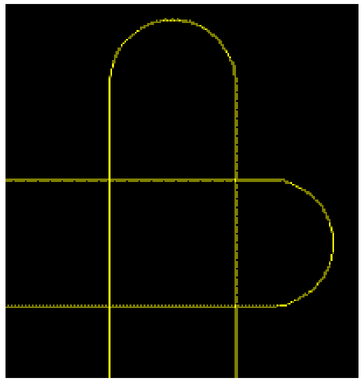
Lines cross slightly

Examples of Not Near Enough

Join and Close will NOT work on either of the following examples:



Lines do not touch



Lines cross too far

Limitations

You cannot join or close when the selection includes:

- Objects that are not 2D lines (for example, a keepout)
- 2D Line shapes on different layers
- 2D Lines that are part of a reuse
- 2D Lines that are closed or a circle
- More than one connection available with the same distance
  Exception: The connection is available if both ends aren't already connected with a smaller distance.
- Shapes that cannot be joined or closed into single shape. For example, you cannot do a multiple Join or Close

In this section:

- Joining 2D Lines and Copper Shapes
- Closing 2D Lines and Copper Shapes
- Breaking an Object

Joining 2D Lines and Copper Shapes

Use Join to connect any 2D lines or copper shapes that are “near enough.”
See: Joining and Closing 2D Lines and Copper Shapes

Joining two ends is completed in one of two ways:

- If two segments cross near enough to their ends, the two segments are shortened and the cross point is the point where they are joined
- If the segments don't cross, an additional segment is added

Procedure

1. Select the two 2D lines or copper shapes you want to join.
   Requirement: The objects must be near enough.
2. Right-click and click **Join**.

Result

If the command worked, you'll see "2D Line" in the Status bar. You can now select the entire joined object.

If the command did not work, you'll see the error message: "Join command aborted" in the Output Window.

Closing 2D Lines and Copper Shapes

Use Close to create a closed (and joined) shape that has contiguous lines. The objects must be "near enough."
See: Joining and Closing 2D Lines and Copper Shapes

Procedure

1. Select the 2D lines or copper shapes you want to close.
   Requirement: The objects must be near enough.
2. Right-click and click **Close**.

Result

If the command worked, you'll see "2D Line" in the Stats bar. You can now select the entire closed object.

If the command did not work, you'll see the error message: "Close command aborted" in the Output Window.

Breaking an Object

Any 2D Line or copper shape that is not a single straight line or a single arc can be broken into multiple 2D Lines. The Break command creates a "break point" at every corner of the shape.

Exception: You cannot break a circle or elements that belong to a reuse.

Procedure

1. Select the object you want to break.
2. Right-click and click **Break**.

Result

If the command worked, you'll see "Multiple Selection" in the Stats bar. You can now select each segment separately.

Saving a Drafting Item to a Library

Add drafting items to your library so you can reuse them in this design or any other design. All of these objects are stored with the Lines filter in the Library.

Procedure

1. Press **Shift** and select the item to select the whole shape rather than a segment or corner of the shape.
   Tip: You can select a board outline, 2-D lines, copper objects, keepout objects, and dimension objects.

   Restriction: You cannot save text alone to the library. Text must be combined with a 2D line in order to save it to the library.
2. Right-click and click **Save to Library**.
3. In the Save to Library dialog box, select the library you want, and type the name of the item.
4. Click **OK**.

Result

The drafting item is in the Library. Use the Lines filter in the Library Manager to find the item for future use.

Related Topics

Adding Drafting Items from a Library

Adding Drafting Items from a Library

You can add a drafting item into your design from the library.

Procedure

1. **Drafting Toolbar** button > **From Library** button.
2. In the Get Drafting Item from Library dialog box, select an item from a library.
   Tip: Use the Library list to view drafting items in all libraries or in a specific library. Use wildcards in the Filter to limit the objects that appear in the Drafting Items list.
3. Click **OK**. The item attaches to your pointer.
4. Move the item to where you want and click to place the item.

Tips:

- You cannot add a drafting object to a design if the design is in default layer mode and the object is in increased layer mode. Use the Layers Setup dialog box to change the design to increased layer mode.
- You can place a board outline, copper object, or keepout object from a library only while on-line DRC is off.

Related Topics

Saving a Drafting Item to a Library

Changing Trace or Drafting Object Width

There are two methods to change the current width of traces or drafting objects.

1. To change the width of a segment, pin pair, or net once it is routed:
   a. Select the item.
   b. Right-click and select **Properties**.
2. To find all traces of a similar width:
   a. On the **Edit** menu click Find.
   b. Select the traces, and change them to a new width.

Drawn Line Width

This is the width applied when you begin to draw a shape.

You set the default width of drawn lines differently from that of route widths. Use the Drafting / Text and Lines page in the Options dialog box to set the default width for drawn lines. To change the width of an existing shape, select the shape and right-click and click Properties. Use the Find dialog box to search for and change all lines of a similar width.

Setting the Pointer Location for Items in the Paste Buffer

The default origin for the paste buffer contents is the lower left corner of the area that encompasses all information in the buffer. You can set the origin at a specific location.

Procedure

1. **Tools** menu > **Options** > **Design page**.
2. In the Move preference area, click **Move by Cursor Location** and then click OK.
3. In the design, place the pointer at the new origin location.
4. Select the information to cut or copy and press Ctrl+X (for Cut) or Ctrl+C (for Copy).
5. Press Ctrl+P. The image appears on the pointer with the origin as the point where the image was selected.
6. Find the new location and click to place the object.

Selecting Drafting Objects

Drafting objects are made up of the line segment outlines and sometimes fill. At times you want to select a segment and at other times you need to select the whole outline with the fill included.

Selecting Drafting Object Outlines

You select the outlines of a drafting object in order to move the outline to change the shape of the object.

1. Right-click and click **Select Anything**.
2. Click the outline of the Shape.
3. Right-click and click a command to alter the segment as needed.

Selecting Whole Drafting Objects

You select the outlines and fill of a drafting object in order to move the object or change the properties of the whole object. There are several ways to select the whole drafting object.

- With nothing selected, right-click and click Select Shapes. Click the object.

While designing, you may find yourself frequently using the Select Anything filter mode and only entering a more specific filter because of design density. Here are some methods to select drafting objects in Select Anything mode.

- Shift-click along the outline of the object.
- Drag a selection box over an outline segment.

Chapter 26
Cut, Copy, and Paste

In this chapter:

To Cut Objects

Use Cut to remove a selected item from the design and place it into the Windows paste buffer.

1. Select the item you want to cut.
2. Click **Cut** from the **Edit** menu.

The item is removed from the design and placed in the Windows paste buffer.

To Copy Objects

Use Copy to place a copy of a selected item from the design into the Windows paste buffer.

1. Select the item you want to copy.
2. Click **Copy** from the **Edit** menu.

The item is placed in the Windows paste buffer.

To Copy a Bitmap

You can use Capture Area to define a rectangular area to copy graphics information as a bitmap image. Then you can switch to another application, Microsoft Word for example, and use the Paste command to insert the bitmap into a document.

To create a bitmap:

1. **View** menu > **Copy as Bitmap**.

2. Area select the area to copy. The area is automatically saved to the Windows copy buffer. All items in the rectangle including the background, dot grid, and color, are copied.

3. Open the application in which you want to place the bitmap and use the Paste command to place the image.

Tip: In addition to generating a bitmap of the defined area, the entire work area is copied as a picture that you can insert with the Paste Special command in products that support this feature.

To Paste Objects

Use Paste to place the contents of the Windows paste buffer (what you copies or moved there) anywhere on the design. You can also open a different design to paste into. You can paste the same copy repeatedly: the copy remains in the buffer until you overwrite it with a new cut or copy action.

1. Cut or copy an item to place it in the Windows paste buffer.

2. Click **Paste** from the **Edit** menu.

The item is pasted into your design.

Chapter 27
Clearances and Keepouts

In this chapter:

- To View the Clearance Between Nets
- To View the Clearance Between Items
- To View the Clearance Between a Net and an Item

To View the Clearance Between Nets

Use Net-to-Net Clearance to detect the minimum clearance for specified nets during PCB layout design. You can use minimum clearance to spread busses of traces or differential pairs.

To show the minimum clearance between two nets:

1. **View** menu > **Clearance** > **Net to Net** button.
2. To center the minimum clearance marker in the workspace, select **Min Clearance**.
3. Select two nets. A minimum clearance marker displays the minimum clearance location.

Tip: If the minimum clearance marker is not visible in the workspace, the minimum clearance box remains blank.

To View the Clearance Between Items

Use Item-to-Item clearance to annotate a PCB layout design with dimensioning that indicates critical or problem clearances. Annotate your design after you complete layout.

To show the minimum clearance between two items:

1. Go to the layer on which the items are placed. If you are not on the correct layer, you will not get a true clearance.
2. Click **Clearance** on the **View** menu. The View Clearance dialog box appears.
3. Click the **Item to Item** button.
4. Select the two items. Supported item types are pads, vias, jumpers, traces, 2D lines, copper, and component outlines.

Information about the items appears in the Selected Items area of the View Clearance dialog box. Line extensions and arrows appear on the design showing the location and size of the minimum clearance.

5. To place the dimensions in the design move the pointer to where you want the dimensions and right-click. If you do not want the dimensions in the design, press Cancel.

Clearances are measured on the current layer. When one or both selected items are not on the current layer, the clearance is measured to the centerline of the item that is not on the current layer.

To View the Clearance Between a Net and an Item

Use Net-to-Item Clearance to detect the minimum clearance for a specified net during PCB layout design. You can use minimum clearance to spread busses of traces or differential pairs.

To show the minimum clearance between a net and its surrounding items:

1. **View** menu > **Clearance** > **Net to Item** button
2. To center the minimum clearance marker in the workspace, select **Min Clearance**.
3. Select a net. A minimum clearance marker displays the minimum clearance location.

Tip: If the minimum clearance marker is not visible in the workspace, Minimum Clearance remains blank.

Chapter 28
Copper Operations

In this chapter:

Creating a Copper Shape

Copper is used, for example, as heat sinks, shielding, or bridging nets. Nets can be assigned to copper shapes.

Requirements

- Ensure you select a layer for placement of the copper. You cannot create a copper shape on all layers (layer number zero). If you need the same copper shape on many layers, copy the shape to other layers.
- Copper must be created with On-Line Design Rule Checking in the Off mode since copper objects cover any and all other electrical objects. If you need a copper shape that avoids objects other than the net to which it is assigned, use the copper pour feature. See also: Creating a Copper Pour Area.

Procedure

1. Click the **Copper** button from the Drafting toolbar.
2. Right-click and click a command to change the values of the drafting object. See also: Setting Values Before Creating a Drafting Object.
3. Create the shape using one of the following:
   - Creating a Circle Drafting Object
   - Creating a Polygon or Path Drafting Object
   - Creating a Rectangle Drafting Object

4. Once you complete the shape, it becomes a filled shape, and the Add Drafting Properties dialog box appears.

5. In the Add Drafting properties dialog box, you can make changes to the copper properties.

    - For example, in the Net Assignment area, you can assign a net by selecting the net name in the Net list or you can click the Assign Net by Click button and click a design object attached to the net. See also: Modifying Drafting Object Properties.

Result

Is the resulting shape what you expected?

- If the fill is not correct, see The Fill of Copper, Copper Pour and Plane Shapes.
- If the shape edges are not correct, see Edge Precision of Drafting Shapes.
- If the shape needs to be modified, see Modifying Drafting Object Properties.
- If you want to start over, see Deleting a Drafting Segment or Object.

Related Topics

Creating a Copper Cut Out

Assigning Existing Copper to a Net

Routing To and From a Copper Shape

Creating Copper in the Decal

Bridging Nets with Copper

Creating Copper Chamfered Paths

Because some designs, for example RF designs, require specialized trace corner geometries, you can add chamfered copper to your design as an alternative for regular traces. Using Chamfered Path, orthogonal corners are square or chamfered, wide angle corners can be chamfered, and acute corners are chamfered. And since copper is created with an outline and a fill, you can specify a very narrow outline width to achieve very sharp corners. All Chamfered Paths are created with a Solid Copper fill.

Restriction: This feature requires the Advanced Editing/RF license option.

1. Drafting Toolbar > **Copper** button.

    - If you have Online Design Rule Checking enabled, in the prompt that appears, click **OK** to switch off Design Rule Checking.

2. Right-click and click **Chamfered Path**.

3. In the Add Chamfered Path dialog box, set the Chamfered Path Parameters and then click OK.
4. Click to start the copper.
5. Move the pointer, then do one of the following:
    - Click to place a corner.
    - Type **S<x>, <y>,** where <x> and <y> are the coordinate values for the next corner.
    - Type **SR<x>, <y>** if you want to specify relative coordinate values.

    Tip: Right-click and click **Add Arc** to add an arc instead of a straight-line segment.
6. Continue adding copper and corners as necessary.
7. To back up and remove the last corner, press **BackSpace**.
8. To cancel the operation, press **Esc**.
9. To end the copper, either **double-click** or right-click and click **Complete**.
10. In the Properties dialog box, you can assign a net and modify other settings.

    Tip: If you create a chamfered path pin pair, unroutes will update when you add the copper in the same way as a trace is placed between pads - from and to the center (or Pin Position) of the pads and when you assign the net to the copper. You can also convert traces to chamfered copper paths which automatically assigns the net and unroutes would also update with the original trace connection. See Converting a Trace to a Copper Chamfered Path.

Exception: The chamfered path does not appear as the correct width if the Minimum display width option for the design is larger than the Polygon outline width of the path. The path appears more narrow by the value of the Polygon outline width.

Related Topics

Setting Chamfered Path Parameters

Restoring Traces After Conversion to Copper Chamfered Paths

Setting Chamfered Path Parameters

The Add Chamfered Path dialog box opens upon selection of the Chamfered Path - copper shape. You set Chamfered Path parameters before you add the copper to the design.

1. In the Polygon outline width box, type a width value for the copper outline.

    Tip: Since copper is created with an outline and a fill, you can specify a very narrow outline width to achieve very sharp corners. Decrease the value for sharper corners and increase the value for more blunt corners. All corners are rounded with a radius equal to one half of the outline width.

2. In the Chamfered path width box, type a width value for the overall width of the copper path.
3. In the Corner chamfer width ratio box, type a value specifying the ratio of the chamfered corner width to the chamfered path width. If the ratio is 1.0, the width of the chamfered corner is the same as the chamfered path. Reduce the ratio for a more narrow chamfered corner.
4. Clear the **Chamfer corners with angles less than or equal to** check box to only chamfer angles less than 90 degrees.
   or
   Select the **Chamfer corners with angles less than or equal to** check box to specify an angle between 90 and 180 degrees as the upper limit beneath which all angles are chamfered. Outside corners less than 90 degrees are always chamfered.
5. Click **OK**.

Bridging Nets with Copper

You can create a physical connection between objects of two different nets with design rule checking enabled.

Restrictions

- You can not create bridge copper in the decal.
- Bridge copper saved to the library loses its bridge status.

Procedure

1. Use one the following methods.
   - If the copper already exists, select the copper shape, and then right-click and click **Properties**.
   - If the copper doesn't already exist, Create the copper shape.
2. In the Add Drafting or Drafting Properties dialog box, select the **Bridge** check box.
3. Click the **Nets to bridge** button.
4. In the Net Association dialog box, add two or more nets to bridge from the Available nets list.
5. Click **OK** in the Net Associations dialog box.
6. Click **OK** in the Add Drafting dialog box.

Results

You can use this copper to bridge objects of the assigned nets without getting an online design rule checking error.

When you run a connectivity check, it will only report an error if no physical connection exists between the bridge copper and a net associated to it.

When you run a clearance check, it will only report an error if you use any non bridge-copper object to bridge the nets. This is also prevented by online DRC.

Tip: You can use the Find dialog box to search for *Nets with bridges*. Results display the nets and list the bridge coppers associated with them.

Restriction: You can copy bridge coppers and they retain their bridge status, but their net associations are removed. See also: Copying Bridge Copper in ECO Mode

Warning: Bridged nets are combined when you export the design into the .hyp (HyperLynx), .ipc (IPC356), and .tgz (ODB++) file formats.

Assigning a Unique Netname to a Copper, Copper Pour, or Plane Area

You can assign a unique netname to a copper, copper pour or plane area., which lets you assign rules and properties to the copper shape independently of the netlist.

Procedure

1. Select a shape > right-click > **Add Net**.
2. To specify a custom netname, click **Typed New Name** and type the netname in the Name of New Net box.

   Alternative: Click Autogenerated New Name to use the default netname.

   Result: The new name is added to the netlist.

Tips:

- The maximum netname length is 47 characters. You can use any alphanumeric characters except braces { }, asterisks *, spaces, question marks, or commas.
- Because this action does not affect the schematic logic, the new netname is not recorded as an ECO operation.
- To assign an existing netname to a selected copper shape, use the Drafting Properties dialog box.

Related Topics

Creating a Copper Pour Area

To Find By Pour Area and Isolated Pour

Flooding a Copper Pour Area

Assigning Copper to a Net

Creating a Copper Cut Out

Copper cut outs are used for creating cut outs or voids inside copper shapes. Creating a cut out involves combining the fixed copper shape with the copper cut out.

Restriction: A cut out does not create a void in the outline of the copper shape. Create the cut out as a polygon shape to create features in the outline.

Requirement

- You must put the cut out on the same layer as the copper.

Procedure

You can create the cut out before the copper or the copper before the cut out.

1. **Drafting Toolbar** > **Copper Cut Out** button.
2. Create one or more cut out areas for the copper. See Creating a Drafting Object for more information.
3. Set the selection filter to select shapes by right-clicking and clicking **Select Shapes**.

   Tip: If you cannot see a cut out within the copper shape, use the Drafting tab in the Options dialog box to set the hatch view to No Hatch. See the Hatch options on the Drafting / Hatch and Flood page of the Options dialog box.
4. Drag a box to group-select the copper and cut out(s).

   Alternative: Some circumstances don't allow for dragging a group selection box. Instead, click to select the cut out. Typically, the copper is selected first. Click the Cycle button to cycle to selecting the cut out. If you have multiple cut outs, use Ctrl+click to select the copper then click the Cycle button to select another cut out. As you select additional cut outs, previous cut outs do not retain the selection color but remain selected. Finally, Ctrl+click to select the copper.
5. Right-click and click **Combine**. Copper inside the cut out area is automatically removed.

Result

Is the result what you expected?

- Do you need to move or edit a cut out? You'll need to uncombine it first. See Uncombining Drafting Objects.
- Do you need to move or edit the copper? Any modifications you make to the copper shape (move, rotate, miter, etc.,) also affect the combined cut out(s).

Creating Nested Copper

You can create a copper shape within another, "outer", copper shape. To do this, you create a copper cut out inside the outer shape, combine it with the outer shape, and then create the new nested copper shape inside the copper cutout.

Procedure

1. In the outer copper shape, Create a copper cut out of appropriate size and shape, and combine it with the outer shape.
2. Create the new nested copper shape within the copper cut out.

Results

You should see the new nested copper shape inside the cut out in the outer shape.

Chapter 29
Copper Pour Operations

In this chapter:

Customizing Design Rule Thermals

You can customize the default thermals that are created on pins and vias within copper pour areas and plane areas. Default thermals are created based upon the Thermals Options for spoke settings and the Copper-to-Pad or Copper-to-Via clearance rule value for the clearance between the pin pad or via pad and the copper.

Restriction

Default thermals are the only thermals available to copper pour areas. Plane areas can also use custom thermals created in pin or via pad stacks.

Procedure

1. Open the Thermals Options. There are separate settings for Drilled (Through-hole) and Non-Drilled (surface mount) pads.
2. Set the width of the spokes in the Width box.
3. Set the minimum allowed number of spokes in the Min. spoke box.
4. For each pad shape, set the orientation of the spokes.

   Tip: If you only want to flood over vias, you can enable an option in the properties of the copper pour or plane area. See Flood and Hatch Options Dialog Box for the *Flood over vias* check box.

5. If you want to allow routed pads to also have thermal connections, select the *Routed pad thermals* check box.

   Restriction: This option applies only to copper pour areas and not to plane areas.

6. Open the Design Rules. In any applicable level of the hierarchy of the design rules, in the Clearance rules, set the Copper-to-Pad and Copper-to-Via value according to the needs of your design.

Results

- Once you fill the copper pour or plane area, a therm.err report is generated if any pads receive fewer spokes than the Min. spoke value. For example, if the pad intersects the boundary of a copper pour, it may not be possible to create all four spokes.
- Did a pad not receive a thermal even though the *Routed pad thermals* option wasn't enabled? Use the Selection Filter to check for trace stubs at the pad. Small left-over trace segments attached to a pad prevent a pad from receiving thermals if the Routed Pad Thermals option is not enabled. Set the Selection Filter to Traces, Corners, and Tacks to select and remove trace segments attached to the pad.

Related Topics

Design Rule Hierarchy

Customizing Design Rule Antipads of Copper Pours

Customizing Design Rule Antipads of Copper Pours

You can customize the default antipads that are created around pins, vias, or drill holes within copper pour areas. Default antipads are created based upon the Copper-to-Pad, Copper-to-Via, or Copper-to-Drill clearance rule value. You can apply the clearance rule value to either the net of the copper or the net of the pad/via or both - unless the pin is a no-connect. You can use the clearance rules at any level of the rules hierarchy.

Restriction

The antipads created in copper pour areas do not remove the unused pads. Plane areas are capable of automatically removing the unused pads.

Procedure

- Open the Design Rules. In any applicable level of the hierarchy of the design rules, in the Clearance rules, set the Copper-to-Pad, Copper-to-Via, and Copper-to-Drill values according to the needs of your design.

Example

By applying the clearance to the net of the copper rather than the pads, you can affect all pads vias and drills within the copper pour area. But by applying the clearance to the net of pads and vias rather than the copper, you can achieve different antipad spacings. Or you can use a combination - apply a general minimum clearance to the net of the copper pour and apply a larger clearance to nets of objects that need larger antipads.

Results

- When you're dealing with antipads, there are two nets involved. If you're not getting the results you expect, ensure that the clearance values of the other net involved aren't conflicting with the values you've already changed. Larger clearance values always have precedence.
- With through hole pins and vias, the copper-to-drill value also applies. Although this clearance value will typically be less than the value of copper-to-pad or copper-to-via, it can still take precedence if it is larger.

Related Topics

Customizing Design Rule Thermals

Design Rule Hierarchy

Creating a Copper Pour Area

Use copper pour to create a copper area that avoids objects not connected to net assigned to the pour, but makes thermal connections to objects that are connected to that net.

Poured copper provides a plane of connectivity that is insulated from traces or pins that do not share the netname. You can flood a copper pour outline to create isolation areas around intersecting pins and traces automatically. Fixed copper is used for electrical items not found in traces or pads. PADS Layout does not automatically create isolation areas around intersecting pins and traces.

See Also: Differences Between Copper, Copper Pour, and Plane Areas.

Procedure

1. Click the **Copper Pour** button from the Drafting toolbar.
2. Right-click and click a command to change the values of the drafting object. See also: Setting Values Before Creating a Drafting Object.
3. Create the shape using one of the following:
    - Creating a Circle Drafting Object

    - Creating a Polygon or Path Drafting Object
    - Creating a Rectangle Drafting Object

4. In the Add Drafting properties dialog box that appears:

    a. Assign a net by selecting the net name in the Net list; or you can click the Assign Net by Click button and click a design object attached to the net.

    b. Make any other appropriate changes to the copper pour properties.

5. Click OK.

6. Flood the copper pour.

Result

Is the resulting shape what you expected?

- If the gap between the pad and the copper for thermals or antipads is incorrect, see Customizing Design Rule Thermals or Customizing Design Rule Antipads of Copper Pours
- If the fill is not correct, see The Fill of Copper, Copper Pour and Plane Shapes.
- If the shape edges are not correct, see Edge Precision of Drafting Shapes.
- If the shape needs to be modified, see Modifying Drafting Object Properties.
- If you want to start over, see Deleting a Drafting Segment or Object.

Related Topics

Creating a Copper Pour Area Cut Out

To Find By Pour Area and Isolated Pour

Assigning Copper to a Net

Creating a Copper Pour Area Cut Out

Copper pour cut outs are used for creating cut outs or voids inside copper pour areas. Creating a cut out involves combining the copper pour shape with the copper pour cut out shape.

Restriction: A cut out does not create a void in the outline of the copper pour area. Create the outline as a polygon shape to create features in the outline.

Requirement

- You must put the cut out on the same layer as the copper pour area.

Procedure

You can create the cut out before the copper pour or the copper pour before the cut out.

1. **Drafting Toolbar** > **Copper Pour Cut Out** button.
2. Create one or more cut out areas for the copper pour. See Creating a Drafting Object for more information.
3. Set the selection filter to select shapes by right-clicking and clicking **Select Shapes**.

   Tip: If you cannot see a cut out within the copper pour shape because the shape is flooded, type the modeless command PO to return the copper pour area to pour outline display mode. You will need to re-flood the plane once you've combined the cut out with the copper pour area.
4. Drag a box to group-select the copper pour and cut out(s).

   Alternative: Some circumstances don't allow for dragging a group selection box. Instead, use Ctrl+click to select the copper pour and the cut out.
5. Right-click and click **Combine**.
6. Flood the copper pour. Copper pour inside the cut out area is automatically removed.

Result

Is the result what you expected?

- Do you need to move or edit the copper pour or the cut out? Any modifications you make to the copper shape (move, rotate, miter, etc.,) also affect the combined cut out(s). You'll need to uncombine it first. See Uncombining Drafting Objects.

Creating Nested Copper Pour Areas

You can create a copper pour area within another, "outer", copper pour area. To do this, you create a copper pour cut out inside the outer area, combine it with the outer copper pour, and then create the new nested copper pour are inside the copper pour cut out.

Procedure

1. In the outer copper pour area, Create a copper pour area cut out of appropriate size and shape, and combine it with the outer pour area.
2. Create the new nested copper pour area within the copper pour cut out.

Results

You should see the new nested copper shape inside the cut out in the outer shape.

Assigning Copper to a Net

You can assign copper to a net by using any of the following methods:

- Assigning existing copper to a net
- Assigning existing copper to a different net
- Assigning new copper to a net, before starting drafting
- Assigning new copper to a net, before completing drafting

Warning: Moving a copper assigned to a net to a non-electrical layer removes the net connection from the copper. Place the copper on an electrical layer to reassign the net connection.

Assigning Existing Copper to a Net

To assign existing copper to a net:

1. Create the copper shape or create the pour on a specific routing layer.
2. Select one or more coppers, right-click, and then click **Assign Net By Click**.

   Tip: If any of the coppers has already been assigned to a net, Assign Net By Click is unavailable.
3. Click a design object, such as a pin, via, trace, net, copper, or unroute, with the netname you want to assign to the copper.

   Result: The status bar displays the netname assigned to the copper.

Assigning Existing Copper to a Different Net

To assign existing copper to a different net:

1. Select one or more coppers, right-click, and then click **Properties**.
2. Click the **Assign Net By Click** button, and then click a pin, via, trace, net, copper, or unroute, with the netname you want to assign to the copper.

   Alternative: In the Net list, select a net.
3. Click **OK**.

Assigning New Copper to a Net Before Starting Drafting

To assign new copper to a net, before starting drafting:

1. Select a single design object, such as a pin, via, trace, net, copper, or unroute, on the net you want to assign to the copper.
2. Right-click, and then click **Add Copper Area** or **Add Copper Pour**.

Tip: If you select multiple design objects, Add Copper Area and Add Copper Pour are unavailable.

3. Create the copper shape or create the pour on a specific routing layer.

   Result: The copper inherits the netname from the selected object.

Assigning New Copper to a Net Before Completing Drafting

To assign new copper to a net, before completing drafting:

1. Begin creating the copper shape or creating the pour on a specific routing layer.
2. Right-click and click **Complete**. The Add Drafting dialog box appears.

   Tip: If you do not want the Add Drafting dialog box to appear when completing the copper, clear the Prompt for Net Name at Completion of Copper check box on the Drafting / Hatch and Flood page on the Options dialog box.
3. Click the **Assign Net By Click** button, and then click a pin, via, trace, net, copper, or unroute, with the netname you want to assign to the copper.

   Alternative: In the Net list, select a net.
4. Click **OK**.

Chapter 30
Plane Operations

In this chapter:

Creating a Plane Area

A plane area is a closed, non-self-intersecting polygon that provides access to universally necessary nets, like power and ground. One design may use several plane areas. Planes are usually located on inner layers that are dedicated to the plane only, although you can place them on outer layers. The plane area can occupy all or part of the layer it is on. Two or more partial planes can occupy one layer, each servicing a different net. These are called split planes.

Plane areas defined with Copper Pour may have insulated traces and vias passing across the plane area, as long as the traces do not divide the plane to break connectivity.

You can create plane areas manually using the drafting tools or automatically using Create Plane Area, using the board outline as a guide. After identifying a layer as a split/mixed plane layer and assigning nets to the plane, you can add plane areas.

See also: Differences Between Copper, Copper Pour, and Plane Areas, and Split Planes in the *Concepts Guide.*

Restriction

You can only create a plane area on a layer set as a split/mixed plane layer and the net you want to assign to the shape must also be assigned to the layer.in the Layers Setup dialog box.

To Manually Create a Plane Area

You create plane areas the same way you create 2D paths with the drafting tools: define each corner of the area. To manually create a plane area using the Plane Area button from the Drafting toolbar:

1. Set the active layer to a split/mixed plane layer in the Layer list on the toolbar.
2. Click the **Plane Area** button from the **Drafting** toolbar.
3. Click to indicate a starting location for the plane area.
4. Continue indicating corners to define the plane.
5. Double-click the starting location again to create a closed area.

To Automatically Create a Plane Area

You can create a plane area by copying the shape of the board outline. A shape is generated proportionally smaller than the outline and includes miters or small notches existing in the board outline. To automatically create a plane area using the board outline as a guide:

1. Set the active layer to a split/mixed plane layer in the Layer list on the toolbar.
2. Select the board outline.
3. Right-click and click **Create Plane Area**.
4. Click a net to associate with the plane area in the Assigned Net to Selected Polygon dialog box. You can also click **None** and associate a net later.

   You can create plane area polygons for assigned nets when assigning or you can wait until later. If you create them when assigning, pads associated with these nets will get thermal indicators if they are inside the appropriate polygons.

Tip: When you use Create Plane Area, you can receive an error message indicating a self-intersecting polygon. The generated plane area is proportionally smaller than the outline. Small miters and/or notches in the board outline can be impossible to create smaller using the specified line width. This error can be corrected most often by reducing the distance between the board outline and the plane. Type a larger gap for the Auto Separate Gap setting in the Split/Mixed Plane tab of Options. Alternatively, you can use a smaller line width for the plane area. This is not preferred since you also need to reduce the copper hatch grid to match in order to maintain a solid fill for the plane. Generating more fill lines consumes more memory and could slow operations if the line width is greatly reduced.

Customizing Design Rule Antipads of Plane Areas

You can customize the default antipads that are created around pins, vias, or drill holes within plane areas. Default antipads are created based upon the Copper-to-Pad, Copper-to-Via, or Copper-to-Drill clearance rule value for the clearance between the pin pad, via pad, or drill hole and the copper. You can apply the clearance value to either the net of the plane or the net of the pad/via.

Procedure

- Open the Design Rules. In any applicable level of the hierarchy of the design rules, in the Clearance rules, set the Copper-to-Pad, Copper-to-Via, and Copper-to-Drill values according to the needs of your design.

Example

By applying the clearance to the net of the plane rather than the pads, you can affect all pads, vias, and drills within the plane area. But by applying the clearance to the net of pads and vias rather than the plane, you can achieve different antipad spacings. Or you can use a combination - apply a general minimum clearance to the net of the plane and apply a larger clearance to nets of objects that need larger antipads.

Results

- When you’re dealing with antipads, there are two nets involved. If you’re not getting the results you expect, ensure that the clearance values of the other net involved aren’t conflicting with the values you’ve already changed. Larger clearance values always have precedence.
- With through hole pins and vias, the copper-to-drill value also applies. Although this clearance value will typically be less than the value of copper-to-pad or copper-to-via, it could still take precedence if it was larger.
- The Split/Mixed Plane options *Remove unused pads* and *Use design rules for thermals and antipad*s both affect the clearance of the antipad.

Related Topics

Customizing Design Rule Thermals

To Associate a Net to a Plane Area

Use the Net Properties dialog box to associate a net to a plane if:

- You created a plane area before you assigned nets to the plane in the Layer Setup dialog box.
- You chose None from the Assign Net to Selected Polygon dialog box when you split a plane area.
- You want to associate a different net to a plane area.

To Associate a net to a plane area:

1. Select the plane area outline.
2. Right-click and click **Properties**.
3. Click a net from the **Net** list.
4. Click **OK**.

Assigning Nets to Split/Mixed Plane Layers

Use Assign Nets in the Layers Setup dialog box to assign one or more nets to the split/mixed plane.

1. **Setup** menu > **Layer Definition** > select a split/mixed plane layer > **Assign Nets** button.
2. Select a net from the **All Nets** list.
3. Click **Add** to move the net to the Assigned Nets list box.
4. Repeat Steps 2 and 3 to associate additional nets.
5. Click **OK** to close Plane Layer Nets dialog box.
6. Click **OK** to close the Layer Setup dialog box.

You can create plane area polygons for assigned nets when assigning or you can wait until later. If you create them when assigning, pads associated with these nets will get thermal indicators if they are inside the appropriate polygons.

You can assign a net to as many layers as required.

Related Topics

Creating a Plane Area

Creating Split Planes

Controlling the Display of Thermals in Plane Areas

Use the Split/Mixed Plane page in the Options dialog box to control the display of thermal information. You can display all thermal information to view the plane in a flooded state or just display the thermal indicators.

See also: Assigning Plane Thermal Attributes

Use the Active Layer Comes to Front check box in the Global / General page to view more detail for the thermal and antipad clearances. When this check box is on, a small outline in the shape of the pad or via is visible when the current layer is a split/mixed plane layer. If the thermal display properties are correctly set, thermal graphics appear at all pins and vias belonging to the plane net.

System-prompted Plane Area Filling

There are times when the system will prompt you to fill a plane area.

Verify Design

When you use Tools menu > Verify Design and you perform a Plane Check and select the Check clearance and connectivity option in the Setup, it checks for proper setup, creates the plane area (if only one net is assigned), if necessary; and prompts to fill the plane automatically.

See also: Verify the Design

Run CAM

When you run CAM on a split/mixed plane layer, it checks for proper setup; creates the plane area if necessary; and prompts to fill the plane automatically.

See also: Creating the Outputs

Troubleshooting Plane Area Fills

The following situations can prevent a plane area from filling.

1. No pads of the assigned net are located within the plane area. Ensure that at least one pad of the net assigned to the plane area is within the shape.

2. No drilled pads are located within the plane area. Typically, the default for non-drilled pads is to not have the Plane Thermal property enabled. Enable the Plane Thermal property for non-drilled pads to allow them to connect to the plane with thermals. Select the pad(s), right-click and click Properties. In the Pin Properties dialog box, select the Plane Thermal check box. This causes the system to re-check which pins should get plane thermals based on the layers and nets.

3. No pads or vias located within the plane area have the Plane Thermal property enabled. Enable the Plane Thermal property for pads and vias to allow them to connect to the plane with thermals. Select the pad(s), right-click and click Properties. In the Pin Properties dialog box, select the Plane Thermal check box. This causes the system to re-check which pins and vias should get plane thermals based on the layers and nets. Repeat the process for the via(s).

4. The layer on which you created the plane is not defined as a Split/Mixed plane layer and/or the net you assigned to the plane area is not a net assigned to that layer. Ensure the Layer is defined as a Split/Mixed plane layer, and assign the net to the layer.

5. No plane area is defined. Ensure you can see the outline of the plane.

6. Multiple plane areas are intersecting or defined on top of each other. One of the planes must take priority and you must create a cutout of the other plane. You might benefit from enabling the *Create cutouts around embedded planes option* found on the Tools menu > Options > Split/Mixed Plane page. For information about giving a plane a flood priority, see the Flood Priority setting in the Flood and Hatch Options dialog box. The Flood Priority numbers should be lowest for inner nested planes and greatest for the outer planes.

7. You've modified the plane outline while the plane area is filled. Return the plane area to the basic plane outline using the modeless command SPO before modifying the shape of the outline.

8. You have embedded/nested planes. You must create the largest plane first, then the smaller inner plane(s) next. (As long as you select the Tools menu > Options > Split/Mixed Plane tab > Create cutouts around embedded planes check box, this automatically assigns the correct flood priority and creates the required plane area cutout within the larger split plane.)
   If you neglected to select the Create cutouts around embedded planes option or you simply created the inner plane area first, the plane areas have conflicting flood priority values and are unable to fill. You must decide the order in which the plane areas should be filled and adjust their flood priorities accordingly. For information about giving a plane a flood priority, see the Flood Priority setting the Flood and Hatch Options dialog box. If you have only one inner plane, there is a quicker way to give it the higher flood priority. Use the SPO modeless command to return the plane to outline mode. Select the plane area shape, right-click and click Bring to Front. The system recognizes the new flood priority and fills the plane areas.

9. You created the plane area with the copper pour tool. You must create plane areas with the Plane Area tool.

10. Traces or pins block the fill from connecting with any pads of the same net. You must free up space for the fill at the current width and grid setting to flow and contact at least one pad of the same net with thermals.

11. Your minimum hatch area setting is too large. In the Drafting Options, check the 'Min. Hatch Area' and ensure the value entered is not larger than the plane area size.

12. Your keepout is restricting the plane area. Assign a color to Keepouts for all layers and ensure they do not restrict copper pour and plane areas for the location and layer.

13. You've hidden the fill. Ensure the fill is visible. In the Drafting Options, in the Hatch View area, click Normal instead of No Hatch.

14. You've set Thermal options to *no connect*. Ensure that drilled and non-drilled pads have orthogonal, diagonal, or flood over settings in the Thermals Options.

15. You've already routed to all available pads of the same net located within the plane area. If nets of the same name as the pour or plane are already routed, ensure that the *Routed Pad Thermals* check box is selected in the Thermals Options.

16. The plane area is defined on a non split/mixed layer. Ensure the plane area is located on a split/mixed layer.

Tip: The preferred method for flooding Split Planes is to use the Plane Connect tab of the Pour Manager.

Troubleshooting Thermal Results

When you fill (flood/pour) a Plane area, a Thermal Relief Errors Report (therm.err) might appear in your default text editor. This file appears when pads within the plane area did not receive the minimum number of spokes according to your Tools menu > Options > Thermals tab settings.

The Thermal Relief Errors Report

Report sections state whether there is less than 100% or even 50% thermal extensions. Below each section there is a listing of the reference designator with pin number (if a component pad), pad coordinate location and the actual number of thermal spokes that were generated. For example:

```
Drilled pads with
less than 50% thermal extensions

Report of Thermal Spokes Generator

On Primary Component Side:

J1.5 (455, 650) # = 2
```

This indicates that pin 5 of reference designator J1, at coordinates 455, 650 has only 2 thermal spokes that were generated. The required thermal spokes for this would be 4 or more. There could be a variety of reasons why the number of thermal spokes is reduced such as a clearance issue where as a trace is too close to a pad, or the copper pour or plane area outline width and hatch grid are too large for the copper to flood between the component pins, etc.

Interpreting Thermal Results

- If a large number of pads and/or vias have zero thermal spokes being generated, there may be a plane area that is not filling. See Troubleshooting Plane Area Fills.
- If there are only a few pads or vias that have zero thermal spokes and yet there is no obvious obstruction to their creation, you might have a situation where those pads or vias do not have the Plane Thermal property enabled. Enable the Plane Thermal property for those pads and vias to allow them to connect to the plane with thermals. Select the pad(s), right-click and click Properties. In the Pin Properties dialog box, select the Plane Thermal check box. This causes the system to re-check which pins and vias should get plane thermals based on the layers and nets. Repeat the process for the via(s).

Creating Split Planes

You can create more than one plane area on a split/mixed plane layer. There are three recommended methods to split a plane:

- Create multiple plane areas.
  See also: Creating a Plane Area
- Use the Auto Plane Separate tool to split a plane that covers the whole layer or a large area into two or more plane areas.
  See also: Automatically Separating Plane Areas
- Embed a plane area within another plane area.
  See also: To Create an Embedded Plane

Automatically Separating Plane Areas

Use the Auto Separate button to divide a plane area into two new plane areas. Using Auto Separate, you draw a path from one side of a plane area to another side of the same area. This path defines the center of the separation between the two new plane areas and divides the plane area into two distinct shapes.

Use the Auto Separate Gap option in the Split/Mixed Plane page to define the distance between the two planes.

Requirements

- You must start and end the separation within the same plane area polygon.
- You must have multiple nets assigned to the Split/Mixed Plane layer.

Procedure

1. On the Standard toolbar in the Layer list, click a split/mixed plane layer.
2. Use one of the following:
    - For object mode, right-click and click Select Anything. Select the starting point of the split on a segment of the plane area. Right-click and click **Auto Separate**.
    - For verb mode, on the Standard toolbar, click the **Drafting toolbar** button. On the Drafting toolbar, click the **Auto Separate** button. Click to indicate a starting location somewhere on the perimeter of the plane area polygon.
3. Click to indicate corners, if any, to define the line of separation.
4. Double-click at the ending point on the plane area outline to complete the Auto Separate command. The Assign Net to Selected Polygon dialog box appears and automatically selects one of the new plane areas.
5. Click a net to assign to the selected plane from the **Choose Existing Net** area. This assigns the net, deselects the plane area, and selects the other area. Click a net for the other area. This assigns the net, deselects the plane area and closes the dialog box.

Tips:

- You must have at least two corners each lying on an edge of the polygon or the message “Too few corners” appears.
- No corners of the auto-separate path may be outside of the polygon or the message “All corners must be within the plane area polygon” appears.
- No segments of the auto-separate path may be outside of the polygon or the message “All segments must be within the plane area polygon” appears.
- If the message “Can't shrink polygon #1” or “Can't shrink polygon #2” appears, reduce each of the new polygons by one-half of the line width. An error occurred during this operation (probably self-intersection).

To Create an Embedded Plane

You can embed planes (or create a plane area within another plane area) using one of the following methods:

- Work from the outermost area to the innermost area. This ensures that all the areas will flood as expected.

  a. Create the outermost plane area by drawing a plane area as described in "Creating a Plane Area."

  b. Create the inner plane area using the **Plane Area** button in the Drafting toolbar.

  c. Continue nesting plane areas until you define all the embedded planes for the layer.

  Tip: The embedded planes are automatically brought to the front so they are flooded correctly. To assign different flood priorities, see the Flood and Hatch Options dialog box.

- If the innermost planes were drawn before the outmost plane areas, correct flooding is not ensured. You must assign flood priorities and then create plane area cut outs around the perimeter of all embedded planes. You must also combine these cut outs with the outer plane area.

  The following steps ensure correct plane flooding:

  a. Select the embedded plane, right-click and click **Bring to front**.

  b. Create a plane area cut out on the outside perimeter of all embedded plane areas.

     See also: Creating a Plane Area Cut Out

  c. Select the **Cut out** polygon shape.

  d. Press **Ctrl** and select the next outer plane area.

  e. Right-click and click **Combine**.

Creating a Plane Area Cut Out

A plane cut out is a closed, non-self-intersecting polygon that defines a void within a plane area. Creating a cut out involves combining the plane area shape with the plane area cut out shape.

Restriction: A cut out does not create a void in the outline of the plane area. Create the outline as a polygon shape to create features in the outline.

Requirement

- You must put the cut out on the same layer as the plane area.

Procedure

You can create the cut out before the plane area or the plane area before the cut out.

1. **Drafting Toolbar** > **Plane Area Cut Out** button.

2. Create one or more cut out areas for the plane. See Creating a Drafting Object for more information.

3. Set the selection filter to select shapes by right-clicking and clicking **Select Shapes**.

   Tip: If you cannot see a cut out within the plane shape because the shape is flooded, type the modeless command SPO to return the plane area to plane polygon display mode. You will need to re-flood the plane once you've combined the cut out with the plane area.

4. Drag a box to group-select the plane and cut out(s).

   Alternative: Some circumstances don't allow for dragging a group selection box. Instead, use Ctrl+click to select the plane area and the cut out.

5. Right-click and click **Combine**.

6. Flood the plane area. The plane area inside the cut out area is automatically removed.

Result

Is the result what you expected?

- Do you need to move or edit the plane area or the cut out? Any modifications you make to the plane shape (move, rotate, miter, etc.,) also affect the combined cut out(s). You'll need to uncombine it first. See Uncombining Drafting Objects.

Assigning Plane Thermal Attributes

When you specify a layer as a split/mixed plane layer, all pad stacks in the design are automatically assigned a thermal setting. Use the Plane Thermal check box in the Pin Properties, Via Properties, and Jumper Pin Properties dialog boxes to determine whether the pad stack is an eligible thermal candidate. Turn the Plane Thermal check box off if you don't want any type of thermal relief on any layer of the pad stack.

Tips:

- The plane thermal attribute for pads determines whether or not a pad on a plane is a candidate for a thermal. Unroutes are not used for this purpose for split/mixed planes. Each pad has a separate thermal attribute for each layer.

- Copper pour shapes continue to use unroutes to determine if a pad is a thermal candidate. Use Plane Area polygons on split/mixed planes to control thermals using the plane thermal attribute.

See Also: Thermal Generation in the *Concepts Guide*

Related Topics

Controlling the Display of Thermals in Plane Areas

To Create Flood-over Pads

To create flood-over pads for component pads on split/mixed plane layers:

1. **Setup** menu > **Pad Stacks**.
2. Select a decal in the **Decal Name** list, and click **Thermal** from the **Pad Style** list.
3. Select the inner layer identified as a split/mixed plane layer from the **Shape, Size, Layer** list.
4. Click a shape from the buttons, either **round** or **square**.
5. Set equal **Inner Diameter** and **Outer Diameter** values (for example, 50 mils).
6. Click **Yes** to apply this update to the selected component or all components with this decal.
7. Repeat the above steps for other decals for which you want flood-over pads.

For via and jumpers pads, flood-over settings are stored per type. Create a new via type if required for flood-over definition, then follow the directions above.

Tip: You can flood-over vias in copper pour or split/mixed plane areas using the Flood and Hatch Options dialog box.

Discarding Plane Data on Save

To discard plane data:

1. **Tools** menu > **Options** > **Split/Mixed Plane** tab.
2. Click **Plane Polygon Outlines** in the **Save to PCB File** area and click **OK**.
3. Click **Pour Manager** from the **Tools** menu to flood or connect the plane areas.
4. Save the design. The Discard Plane Data dialog box appears.
5. Click **Proceed** to save only the plane polygons. Click **Save All** to save all plane data and change the option for future saves.

Displaying Connections for Pads Connected to a Plane

You can hide the connection lines (ratsnest or unroute connections) of nets assigned to a plane. Plane nets are typically power or ground and hiding them can help to clear the design space and reduce the connections to ones that require more difficult routing. Plane net connections can be made by through-hole pins connecting directly to the plane, or through fanout (short trace ending with a via) connections. These nets don't usually require long traces and are often not important in the strategic placement of the part - unless the plane to which they are attached is only in a certain area of the board. You can use the View Nets dialog box to hide the unroute connections of any or all nets in the design.

See also: View Nets Dialog Box

Converting Old Designs to Split Plane Designs

If you defined split planes using old methods, you can easily convert them to PowerPCB 2.0 or later split/mixed planes. The method of conversion is slightly different depending on your original method of creation. Choose the method you used from the following list to find the appropriate conversion technique:

- Converting Planes Separated by 2D Line or Copper Line
- Converting Planes Defined Using Copper Polygons on Plane Layers
- Converting Planes Defined With Copper Pour Polygons

Converting Planes Separated by 2D Line or Copper Line

If you defined your planes by adding line objects to the layer to represent the separation between plane areas, create split plane polygons as follows:

1. Assign the layer as split/mixed plane layer type.
2. Verify your plane net assignments.
3. Define plane polygons.
4. Delete the existing 2D or copper lines if they are no longer required.

To Assign the Layer as Split/Mixed Plane Layer Type

Change the layer type for the split plane layer to a Split/Mixed plane layer.

1. Click **Layer Definition** from the **Setup** menu. The Layers Setup dialog box appears.
2. Click the layer in the **Layer** list, and click **Split/Mixed** from the **Plane Type** area.

To Verify Your Plane Nets Assignments

Check your plane net assignments for each split/mixed plane layer.

1. In the Layers Setup dialog box, click the **split/mixed** layer in the list and click **Assign Nets**.
2. Click **nets** from the **All Nets** list and click **Add** to associate them as required.
3. Click **OK** to close the Plane Layer Nets dialog box and apply the changes.

4. Click **OK** to close the Layer Definition dialog box and apply the changes.

To Define Plane Polygons

Create new split/mixed plane polygons equivalent to the areas defined by the original separation using the commands described in Creating a Plane Area and Automatically Separating Plane Areas.

Converting Planes Defined Using Copper Polygons on Plane Layers

If you defined your planes by creating copper polygons associated with the plane nets, use the following steps to convert them to plane polygons:

1. Assign the layer as split/mixed plane layer type.
2. Verify your plane net assignments.
3. Define plane polygons.

To Assign the Layer as Split/Mixed Plane Layer Type

Change the layer type for the split plane layer to a Split/Mixed plane layer.

1. Click **Layer Definition** from the **Setup** menu. The Layers Setup dialog box appears.
2. Click the layer in the **Layer** list, and click **Split/Mixed** from the **Plane Type** area.

To Verify your Plane Nets Assignments

Check your plane net assignments for each split/mixed plane layer.

1. In the Layers Setup dialog box, click the **split/mixed** layer in the list and click **Assign Nets**.
2. Click **nets** from the **All Nets** list and click **Add** to associate them as required.
3. Click **OK** to close the Plane Layer Nets dialog box and apply the changes.
4. Click **OK** to close the Layer Definition dialog box and apply the changes.

To Define Plane Polygons

Create new split/mixed plane polygons by tracing the outline with a new plane polygon using the commands described in Creating a Plane Area and Automatically Separating Plane Areas.

Converting Planes Defined With Copper Pour Polygons

If you defined your planes using multiple polygons of copper pour, you can easily convert them to split/mixed plane polygons using the following steps:

1. Assign the layer as split/mixed plane layer type.
2. Verify your plane net assignments.

Assign the Layer as Split/Mixed Plane Layer Type

Change the layer type for the split plane layer to a Split/Mixed plane layer.

1. Click **Layer Definition** from the **Setup** menu. The Layers Setup dialog box appears.
2. Click the layer in the **Layer** list, and click **Split/Mixed** from the **Plane Type** area.

Verify your Plane Net Assignments

Check your plane net assignments for each split/mixed plane layer.

1. In the Layers Setup dialog box, click the **split/mixed** layer in the list and click **Assign Nets**.
2. Click **nets** from the **All Nets** list and click **Add** to associate them as required.
3. Click **OK** to close the Plane Layer Nets dialog box and apply the changes.
4. Click **OK** to close the Layer Definition dialog box and apply the changes.

Chapter 31
Routing The Design

There are many ways to route your design in PADS Layout. You can also send your design to PADS Router for autorouting and advanced interactive routing.

See also: Switching to PADS Router

Requirements

- Set the routing direction for all layers.
- Set your grids: design, via, and display.
- Set your design rules and enable DRC.

PADS Layout Routing Tools

Use one of the following tools to route your design:

- Add Route—Click to indicate each corner in the trace. You can route manually or in Verb mode. This type of routing works in any DRC mode.
- Dynamic Route—An interactive autorouter that follows the direction of your pointer as you move it, seeking optimal paths and installing corners as the trace progresses. Dynamic Routing is available only in Prevent Errors DRC mode.
- Auto Route—A pin-to-pin autorouter that automatically adds traces between pin pairs. Other routes are moved as required. Auto Routing is available only in Prevent Errors DRC mode.
- Bus Route—A dynamic router that creates data lines, memory arrays, or similar connections where several routes need to flow together from one set of parts to another. Bus Routing is available only in Prevent Errors DRC mode.
- Autoroute using PADS Router—Set up a routing strategy and options; indicate whether to autoroute with PADS Router in the background or foreground.

Related Topics

During Routing

After Routing

Routing Manually

You can manually route using the Add Route button on the Design toolbar. Click to indicate each corner in the trace. You can route manually or in Verb mode. This type of routing works in any DRC mode.

Tips:

- You can set Route to start automatically when you double-click an unrouted path (connection): select **Add Route** in the **Routing** tab of the Options dialog box.
- Set your grids: design, via, and display.

Procedure

1. **Design** toolbar > **Add Route** button.
2. Select an object to route.
   Tips:
   - The trace begins on the layer on which the object resides. If the object is on multiple layers, for example, a through-hole pin or via, the trace begins on the active layer.
   - When you start routing from a trace, a tack appears at the selection point.
3. Use the pointer to move the end of the trace you are drawing from grid point to grid point.
   Tip: The trace can move in one of the three Angle Modes: Orthogonal, Diagonal, or Any Angle.
   See also: Options Dialog Box, Design Page
4. Click to indicate a corner.
5. To change layers, right-click and click **Add via**.
   See also: Adding a Via While Routing
6. Proceed to your destination until you see the target shape over a pad, trace, or via.
7. Click to complete the trace.

Routing Dynamically

The Dynamic Route tool is an interactive autorouter that follows the direction of your pointer as you move it, seeking optimal paths and installing corners as the trace progresses. Dynamic Routing is available only in Prevent Errors DRC mode.

Tips:

- You can set Dynamic Route to start automatically when you double-click an unrouted path (connection): select **Dynamic Route** in the **Routing** tab of the Options dialog box.
- Set your grids: design, via, and display.
- Each layer has a preferred routing direction, be sure to route on a layer that is set in the direction you need.
  See also: Layers Setup Dialog Box

Requirement

- You must be in Design Rule **Prevent mode (drp** modeless command**)** to route dynamically.

  See also: Turning on Design Rule Checking**,** Design Rule Checking in the *Concepts Guide*

Procedure

1. **Design** toolbar > **Dynamic Route** button.
2. Select an object to route.
   Tips:
   - The trace begins on the layer on which the object resides. If the object is on multiple layers, for example, a through-hole pin or via, the trace begins on the active layer.
   - When you start routing from a trace, a tack appears at the selection point.
3. Guide the pointer through the items you want to bypass without creating corners.
   Tip: The trace can move in one of the three Angle Modes: Orthogonal, Diagonal, or Any Angle.
   See also: Options Dialog Box, Design Page
4. Click to indicate a corner; these act as tacked corners to prevent preceding segments from moving.
5. To change layers, right-click and click **Add via**.
   See also: Adding a Via While Routing
6. Proceed to your destination until you see the target shape over a pad, trace, or via.
7. Click to complete the trace.

Tips:

- To remove unwanted corners, slowly move the pointer back over the unwanted traces.
- As you route, the guard band appears whenever the head of the trace meets a clearance obstacle that it cannot shove. The dynamic route tool won't complete the trace without changing the clearance rules or removing the obstacle.

- Use Transparent Mode, **T** modeless command, to view obstacles that may lie under traces on the active layer.

Related Topics

To Reroute with Sketch Route

Auto Routing

The auto router activates a single-layer, pin-to-pin autorouter for a selected connection or pin pair.

Restrictions

- The auto router operates on one layer only; it will not add vias.
- The auto router follows the current design grid.
  Tip: Set your grids: design and display.
- Each layer has a preferred routing direction, the auto router works only in the preferred routing direction.
  See also: Layers Setup Dialog Box

Requirement

- You must be in Design Rule **Prevent mode (drp** modeless command**)**.

  See also: Turning on Design Rule Checking**,** Design Rule Checking in the *Concepts Guide*

Procedure

1. **Design** toolbar > **Auto Route** button.
2. Select an object to route.

Result

The router makes three attempts to find a path, shoving existing traces in the process to create new channels for the route. If the router can't complete the route, the routine aborts.

Bus Routing

Use Bus Route to dynamically route similar connections where several routes need to flow together from one set of parts to another.

Tips:

- Set your grids: design, via, and display.
- Each layer has a preferred routing direction, be sure to route on a layer that is set in the direction you need.
  See also: Layers Setup Dialog Box

Requirement

- You must be in Design Rule **Prevent mode (drp** modeless command**)** to route dynamically.

  See also: Turning on Design Rule Checking**,** Design Rule Checking in the *Concepts Guide*

Warning: If the Smooth bus route traces option is enabled, the Smooth command is applied to all Bus Route traces once they are complete.

Procedure

1. **Design Toolbar** button > **Bus Route** button.
2. Select multiple objects to route by holding the mouse button down and dragging a selection rectangle around them.
   Result: A route segment for the first connection, or guide route, attaches to the pointer (and follows pointer movement).

   Tip: To use a different connection for the guide route, right-click and click **Next Guide** (or **Previous Guide**) to cycle through all of the connections.
3. As you move the pointer, you route in a dynamic route mode where corners are added automatically.

   Tips:

   - The trace can move in one of the three Angle Modes: Orthogonal, Diagonal, or Any Angle.
     See also: Options Dialog Box, Design Page
   - For best results, click to lock down a corner and use the inside connection as the guide since the guide does not take into account space required by the follow routes to turn around obstacles. After you indicate a corner, the guide route is reattached to the pointer.
   - To add a corner to the guide route only, right-click and click **Add Corner to Guide** (or Alt+Click). The follow routes follow the guide route after you indicate the next corner.
   - The spacing of the routes is constrained by trace to trace clearances and snaps to the Design grid regardless of the snap to grid setting. You can vary the results of the bus

routes by changing the grid spacing. For example, set the x and y coordinates of the grid to zero and then try 10.

- The follow routes pattern is based on the direction of the last segment added for the guide route. For example, if the last guide route segment is horizontal, then all follow routes are added as horizontal with a vertical set of end points.

Result: The other bus connections, or follow routes, are routed following the guide route's path only after you add a corner or a via, end a trace, or complete the connection. If the bus router cannot create follow routes automatically, it switches to a manual bus route mode where you create each corner interactively. The bus router analyzes the new route and routes the other connections if possible, or it returns to manual mode. In manual mode, entering corners is the same as when you route dynamically. When you complete a connection in manual mode, the next connection becomes the new guide route.

4. To change layers for all connections at once:

   a. Right-click and select **Via Pattern Mode**, then click the pattern you want.

   b. Move the pointer to where you want to add the via pattern.

   c. Right-click and click **Add Vias**.

   Troubleshooting: If DRC violations are detected, the message “Bus Router failed. Add vias for the current connection manually” appears on the status bar. Try Add Vias again allowing more room for the via placement using the current pattern. Otherwise, proceed to the next step to add the vias one at a time.

   Tips:

   - The spacing between vias is based on the via size, the via grid, and the widths and clearances of the traces.

   - If the via pattern you just added isn’t quite right, you can right-click and click Cycle Via Pattern to see if a different pattern will improve placement of the vias. Patterns that do not fit into the current area are skipped and next pattern is used.
     Restriction: The Cycle Via Pattern menu item is only available if Add Vias or Add Via to Guide was the last command used.

   See also: Via Pattern Mode in the *Concepts Guide*

5. If Add Vias failed, change the layers one connection at a time:

   a. Move the pointer to where you want to add the via.

   b. Right-click and click **Add Via to Guide**. A via is added to the current guide route and the next connection becomes the new guide route.

   Tips: The spacing between vias is based on the via size, the via grid, and the widths and clearances of the traces.

See also: Adding a Via While Routing

6. To end the trace short of the target (leaving a dangling connection, or ending at a via or testpoint):

   a. Right-click, point to **End Via Mode**, and then click an End mode.
   See also: Using End Via Mode

   b. Move the pointer to where you want to add the tack or via.

   c. Right-click and click **End**.

   Troubleshooting: If DRC violations are detected, the message "Bus Router failed. End bus manually" appears on the status bar. Try ending the bus again allowing more room. Otherwise, end the routes individually clicking End Guide.

   Tip: The spacing between vias is based on the via size, the via grid, and the widths and clearances of the traces.

7. To complete the bus route, proceed to the final destination and do one of the following:

   - Right-click and click **Complete**.
   - To incrementally complete each connection in the bus, move the guide route to its final connection point and click. The pointer changes to a target shape at the correct connection point. The next connection in the bus becomes the new guide route. Repeat this process until you route all connections.

Shortcuts

You can become very proficient with the bus router when you make use of the keyboard shortcuts shown in Table 31-1.

Table 31-1. Bus Router Shortcuts

Command	Shortcut
Next Guide	Tab
Previous Guide	Shift+Tab
Add Corners	Click
Add Corners to Guide	Alt+Click
Add Vias	Shift+Click
Add Vias to Guide	Shift+Alt+Click
End	Ctrl+Click
End Guide	Ctrl+Alt+Click
Complete	Double-click

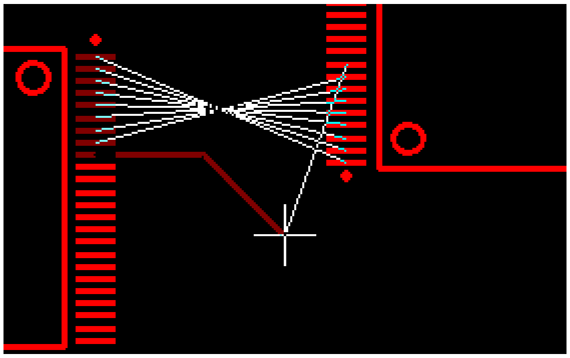
Table 31-1. Bus Router Shortcuts (cont.)

Command	Shortcut
Back Up	Backspace
Cancel	Esc

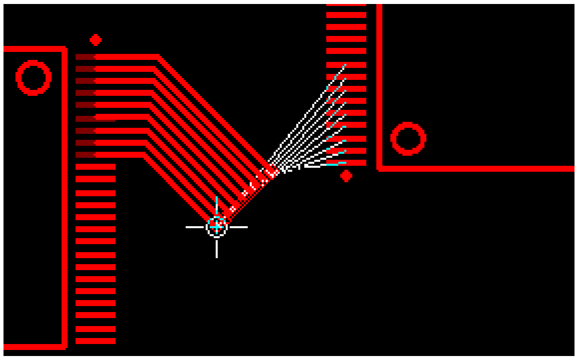
Examples

Listed below are examples of basic bus router operations:

- Moving the pointer before indicating a corner: only the guide route appears.

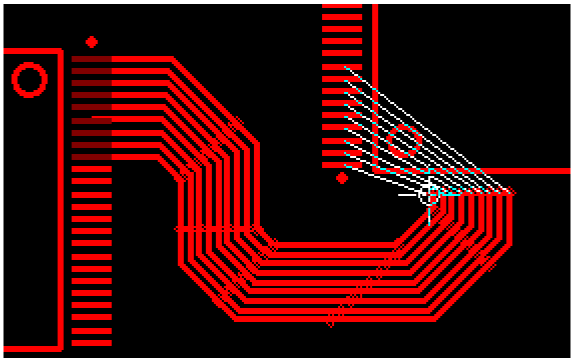


- Indicating a corner point for the bus: the other bus connections or follow routes, are routed following the guide route's path. After you indicate a corner, the guide route is attached to the pointer. Continue indicating corners for the bus.



- Bus routing in progress.

Tip: For best results when adding corners, use the inside connection as the guide.



Related Topics

Setting a Via Type

Selecting Routing Objects

Several route objects exist that you can select for routing. You can also confirm selected information.

To select an object to route, click the object. Objects you can route are: connections, pins, traces, vias, and tacks.

Tips:

- If you have trouble selecting the object, select the object name in the Selection Filter dialog box.
  See also: Using the Selection Filter
- For rectangular and oval pins, or pads, click in the center of the object or use area select.
  See also: Selecting Items
- If PADS Layout placed the tack, the tack is attached to the trace and you must select the trace. The layer on which the trace with the tack exists is the active layer.

Confirming Selected Information

To confirm basic information for a selected object, you can do one of the following:

- Open the status window by pressing **Ctrl+Alt+S**. For more complete information about a specific object, select it and then right-click and click **Properties**.
- When you select a connection, right-click and click **Select Net** to select the entire net.
- Use Transparent Mode (**T** modeless command) to view obstacles that may lie under traces on the current active layer.

For a visual aid set your dot grid to the same spacing as your routing grid. Use the **GD** modeless command and the grid value to do this. You can route on the polar grid as well.

See also: Typing Modeless Commands

To Assign Colors to Nets

To assign colors to nets:

1. **View** menu > **Nets**.
2. Add the net to which you want to assign color to the **View List**.
3. Select that net from the **View List**.
4. Select a color for the net from the **Color by Net** area. The Display Colors dialog box determines colors in this area. If you select **None**, the net does not have a color assignment and it appears as it normally does. All nets are assigned None by default.
5. Click **OK** or **Apply**. The color change takes place.

Tip: You cannot assign a color to net Default.

Related Topics

Modifying Net Properties

Net Properties Dialog Box

View Nets Dialog Box

To Find a Net

To quickly locate a specific net on a dense board:

1. Type the modeless command, **N**.
2. Type the netname and press **Enter**. The net appears in the color you chose in Display Colors. The net is not selected: you have to select a segment to start routing. The highlighting turns off when you complete the route or when you select the net or a pin pair and click **Unhighlight** from the **Edit** menu.

To search for and select a net, or several nets, use the Find dialog box. Finding by nets finds and either highlights or selects nets whether they are routed or not.

See also: Find Dialog Box

To View by Color

To globally hide or display connections:

- Click **Display Colors** from the **Setup** menu. If you change the connection color to the background color, the connections are effectively hidden.

You can also use the Highlight/Unhighlight command to make a particular unrouted connection or net stand out on the design:

1. Select the connection or net.
2. Click **Highlight** on the **Edit** menu. The net appears in the highlight color you set in the <Display Colors Setup dialog box>. The pair or net is not selected; you can select it as you would an unhighlighted item.

   To turn the highlight off, reselect the item and click **Unhighlight** from the **Edit** menu. The highlight is also canceled when the connection is routed.

To View by Netname

Use View Nets to hide or show connections by netname. You can change the appearance of nets, net classes, and nets with rules.

- To selectively hide or display connections, click View Nets from the **View** menu.

Routing To and From a Copper Shape

You can route to and from a copper shape. However, there are restrictions when routing from a copper shape. We recommend you route to the copper shape whenever possible.

Requirements

- You must assign a net to the copper shape before you can route to it. See also: Assigning Copper to a Net
- The net you are routing must be the same as the net assigned to the copper.
- If you are routing to pin copper, the copper must be associated to the pin in the decal in order to router to it. See also: Associating Copper with Terminals

Procedure –To Copper

Unlike routing to a pad where you must route to the center of the pad to complete the connection, you only need to route to the edge of the copper.

1. As you are routing a net, guide the trace to the copper.
2. Choose an end point inside the perimeter of the copper. A target-like shape indicates that you have connected with the copper.

   Alternative: If you are routing on another layer, you can end the trace with a via into the copper. See also: Using End Via Mode

3. Click to finish.

Procedure –From Copper

There are limitations in routing from the copper. You must always select an area where a connection (unroute) emanates from the copper to start the routing process. Because of this, we recommend that you route to the copper whenever possible for more flexible connection points.

1. While in a routing mode, locate the point where a connection (unroute) is emanating from the copper.

   Alternative: You can route and add a via within the copper if you need to immediately change layers.

2. Click to begin the trace.

3. Route your trace.

Restriction: While in ECO Add Route mode, you cannot terminate a newly created connection on copper. The connection must go between two component pins.

Result

After routing, the ratsnest connection continues to exist in the design between the copper and other unrouted objects of that net. The connection disappears after all objects associated with that net are connected.

Setting a Via Type

Use the Vias dialog box to determine what kind of via to install when you change layers during routing. When you choose a via type, that type is used with the Add Vias and Add Vias to Guide commands. You can also change the via type during routing.

See also: To Change the Via Type While Routing

Procedure

- Type one of the following modeless commands:

  VA (automatic via selection) - PADS Layout chooses from all vias, through or partial, that can handle the particular layer change. If PADS Layout finds partial vias dedicated to the layer change, it chooses from them. If PADS Layout can't find a dedicated partial via, it selects any through vias for a through or partial layer change. It then checks the Routing Rules dialog box for vias that are allowed for the net you are routing. If more than one via still passes, PADS Layout installs the one with the smallest drill diameter or smaller pad size. Automatic allows only vias that begin and end on the Layer Pair shown on the Routing tab of the Options dialog box.

  Requirement: To use automatic via mode, the layer pair for routing and the layer pair for the partial via definition need to match. For example, if you have a partial via set up for layers 1 through 4, and the layer pair for routing is set for layers 1 through 8, automatic mode will not insert a via.

VP (partial via) - The automatic via selection still occurs, but it is limited to the partial vias only.

VT (through via) - The list of through vias becomes active. Click the via you want to use as the default and click Apply. This is the via which is installed every time you change between layer pairs.

Or

- Type the **V** modeless command and press **Enter** to open the Vias dialog box where you can select the via mode you want.

The via must not create a clearance violation according to the default clearance rules or the clearance rules attached to the net you are routing, whichever takes precedence. If you set DRC to Prevent Errors, a layer change won't complete if the trace or via creates a clearance problem.

Once you install a via, you can change it to another type using the Via Properties dialog box.

See also: Via Properties Dialog Box

Using End Via Mode

To end a trace with vias, tacks, or test points use the **End Via Mode** command from the **End Via Mode** menu in the shortcut menu. The available modes are listed in Table 31-2. The selected mode is on until you select a different one or exit PADS Layout.

Table 31-2. End Via Modes

Mode	Description
End No Via	Ends the connection with a tack. End No Via is the default setting.
End Via	Ends the connection with a via.
End Test Point	Adds a test point via. **See also:** Placing Test Points, Setting Test Point Properties

Tip: If you try to add a test point to an area defined as a test point keepout area when DRC mode is set to Prevent Errors, the message “Test point keepout violation” appears and the test point is not added.

Related Topics

Bus Routing

Using the Layer Pair

If you're working extensively between two layers, set them as the default layer pair, and while routing, right-click and click **Layer Toggle**, or shift-click to install a via and change to the other layer.

Set the layer pair in the Routing / General page of the Options dialog box. You can also use the modeless command **PL**. Separate the modeless command and each layer number with a space.

To change layers outside the layer pair, use the **L** modeless command or right-click and click **Layer**.

Related Topics

To Change the Layer While Routing

Using Teardrops

Teardrops enhance connectivity between traces and pins. You can set PADS Layout so that it automatically creates teardrops as you route or you can insert teardrops after you route. You can set options for teardrops on new traces, or you can modify teardrops on existing traces.

In this section:

- Creating Teardrops
- Remove All Teardrops
- Disable the Display of Teardrops
- Selectively Disabling Teardrops
- Teardrop Restrictions
- Teardrops in CAM
- Modifying Teardrop Properties
- Checking Teardrops

Creating Teardrops

In the Options dialog box, use the Routing tab option to generate teardrops. Use the Teardrops tab to set options for teardrops on new traces. Any changes you make apply to new traces that you route. These settings are saved with the design.

To generate teardrops on existing pads:

1. **Tools** menu > **Options** > **Routing / General**.

2. In the **Options** area, select the **Generate Teardrops** check box. Teardrops are automatically added when you add a trace segment that enters or leaves a pad.
3. Click **OK**. Teardrops are enabled.

   Exception: Teardrops may not appear on all existing pads if the traces going in to the pads are not centered. Re-route the traces from the pads to the first corners to fix these exceptions.

If you haven't set a teardrop option in a previous PADS Layout session, PADS Layout uses the Default teardrop shape. You can set more specific options for teardrops if you have the additional teardrop functionality using the Routing / Teardrops page in the Options dialog box.

Tip: If you don't want a teardrop on a specific trace you are routing, click **Ignore Teardrop** from the shortcut menu after you click **Route** from the shortcut menu. A teardrop will not appear on the trace you are routing, but will on subsequent routes.

Remove All Teardrops

You can remove all teardrops by reversing the process for creating teardrops.

1. **Tools** menu > **Options** > **Routing / General**.
2. In the **Options** area, clear the Generate Teardrops check box.

Disable the Display of Teardrops

You can also temporarily disable the display of teardrops without removing them from the design.

1. **Tools** menu > **Options** > **Routing / Teardrops**.
2. In the Parameters area, clear the Display Teardrops check box.

Selectively Disabling Teardrops

You can selectively turn teardrops off at the pin level.

1. Select a pin > right-click > **Properties**.
2. Clear the **Teardrops** check box in the Pin Properties dialog box.

To clear multiple pins, set the Selection Filter to **Pins only**, and draw a selection rectangle to select multiple pins. As long as all the selected pins are teardrop-enabled, you can open the Properties dialog box to disable teardrops.

Teardrop Restrictions

Teardrops have the following placement restrictions:

- You cannot place default-shaped teardrops on square pads.
- You cannot place teardrops on traces when a trace corner is inside the pad or via or if the segment is too short. Use the additional teardrop option Auto Adjust to solve this. This option is in the Teardrops tab of the Options dialog box. Use Auto Adjust to set a custom length and width ratio. With Auto Adjust selected, PADS Layout attempts to adjust the length of the teardrop on traces where the trace corner is inside the pad or via or the segment is too short to contain the specified length ratio.

Teardrops in CAM

When you plot teardrops, the line width used is the same as the track width. If the teardrop is on an annular-shaped pad, PADS Layout uses the difference between the outer pad radius and the inner pad radius as the line width if that amount is less than the track width.

Related Topics

Modifying Teardrop Properties

Teardrop Properties on Traces Dialog Box

Modifying Teardrop Properties

You can modify the teardrop shape, length ratio, and width ratio for any selected teardrop, teardrops on all layers, or all teardrops. You can also remove an individual teardrop from a design.

To modify teardrop properties:

1. Select the trace to which the teardrop is attached. If you select in the middle of a trace with teardrops at each end, both teardrops are edited. Select the trace near the teardrop you want to modify to edit only one teardrop.
2. Right-click and click **Properties**.
3. Click the **Teardrop** button. The Teardrop Properties on Traces dialog box appears.

Exceptions:

- If you select multiple traces and modify teardrops on those traces, only existing teardrops are modified. You cannot apply teardrop settings to a trace. Newly created teardrops always use the settings in the Teardrop tab of the Options dialog box.

- If you select Layer or All from the Apply To area, Auto Adjust changes to a third state, neither on nor off. This occurs because Auto Adjust reads the Auto Adjust setting for more than one trace and they can vary. You can still change Auto Adjust.
- You cannot preview teardrops when you select Layer or All.

Related Topics

Using Teardrops

Teardrop Properties Dialog Box

Checking Teardrops

Use this dialog box to check for and view teardrop errors. The error location, layer, and a short explanation of the error are reported.

Requirement: You must enable teardrops before you can check them. **Tools** menu > **Options** > **Routing** tab > select **Generate Teardrops** check box > click **OK**.

In this topic:

- Running a Teardrop Check
- Checking Teardrop Error Results
- Saving and Printing Error Results
- Changing Default Teardrop Report File Names

Running a Teardrop Check

1. **Tools** menu > **Options** > **Teardrops** tab.
2. Click the **Check** button.
3. Click **Start** to run the check.
4. After the check process completes, a message window might open with the number of errors found. Click **OK**.

Checking Teardrop Error Results

Reading Error Details

Error results are listed in the Location box. Listed errors, contain their (X,Y) location, layer, and error type.

1. Select an error from the list.

2. In the Explanation box, view the more specific, detailed information about the error selected in the Location list. The explanation includes information about any conflicting object.

Tip: The current error list remains whether the dialog box is open or closed. The error markers remain until you click Clear Errors, which clears the markers, not the errors.

Viewing Errors in the Design

You can view error markers in the design.

Requirement: Colors must be assigned properly in the Display Colors dialog box in order to see error markers. Errors in the design area appear in the color of the **Errors** check boxes per layer of the Display Colors dialog box. When an error is selected in the Location box, the error in the design area appears in the **Highlight** color.

- Select an error from the list.

Result: The design window pans to the area of the error (unless Disable Panning is selected). This is more obvious if you are zoomed into your design.

Tips:

1. You can prevent the design area from panning to errors when you select them if you select the **Disable Panning** dialog box.
2. Error markers are not erased from the design until you click **Clear Errors** which clears the error markers, not the actual errors. You must correct the error in the design.

Saving and Printing Error Results

You can alternatively view the current error results in your default text editor.

- To view the most recently run report results in the default text editor, click **View Report**. You can print or save the results from your default text editor.

Changing Default Teardrop Report File Names

When you run a Teardrop check, a report file is created with a unique name for each check type.

- To specify the file name, click **Report File**.

  Result: Opens the Save As dialog box where you can name the report if you don't want to use the default.

  Tips:

  - If you do not give the report a unique name, it is always saved to the default file name.

- If you assign it a name, that name is the default for the report type until you change it again.

Managing Tacks

This topic discusses the following:

- Adding a Tack Manually
- Deleting Tacks Manually

Adding a Tack Manually

You can add tacks manually to lock a corner or prevent rerouting when using the dynamic route or bus route tool.

To add the tack:

1. Select a trace.
2. Right-click and click **Add Tack**. The new tack is dynamically attached to your pointer and you can place it like a trace corner.
3. Move the pointer to where you want the tack and click to place the tack.

Verb Mode

4. Click the **Design toolbar** button on the toolbar.
5. Click a routing button from the **Design** toolbar. The interactive router starts when you click an object.

Object Mode

6. Select connections, pins, existing traces, or manually added tacks.
7. Click a routing button from the **Design** toolbar.

Deleting Tacks Manually

- To delete a manually added tack select it and press **Delete**.

When the tack is not required for a route direction change, pressing Delete removes the corner beneath the tack and moves the tack to the next “U” point in the route.

Tip: The selection filter contains an option for tacks so you can set the filter to select only tacks.

Managing Jumpers

Using Jumpers

You can place jumpers in a trace either while routing or after you finish routing. PADS Layout considers jumper pins as vias. The space between the jumper pins does not have electrical properties. Jumper pins are checked for Design Rule clearances as vias.

Tip: PADS Router can load and autoroute jumpers.

In this topic:

- Add Jumpers
- Set Up Jumpers
- Jumper Limitations
- Creating Jumper List Reports

Add Jumpers

To add a jumper while routing:

1. Right-click at the point where you want the first pin of the jumper to appear and click **Add Jumper**. The first pin of the jumper appears.
2. Move the mouse to stretch the jumper. You can stretch between Minimum Length and Maximum Length jumper settings at a preset increment. The angle mode also applies.
3. Click to complete the jumper. Regular routing continues. When you place a jumper, it is checked for Design Rule clearances if DRC is active.

To add a jumper to a trace:

1. Select the trace to which you want to add a jumper at the point where you want the first pin of the jumper to appear.
2. Right-click and click **Add Jumper**. The first pin of the jumper appears.
3. Move the mouse to stretch the jumper. You can stretch between Minimum Length and Maximum Length jumper settings at a preset increment. The angle mode also applies.
4. Click to complete the jumper. Regular routing continues. When you place a jumper, it is checked for Design Rule clearances if DRC is active.

Note: You must enter the second pin on a trace in the same net.

Any Angle mode does not work when adding jumpers to a trace.

Set Up Jumpers

When you add a jumper to the design, the default jumper is used. You can change the default jumper settings or change the settings for a specific jumper in the design. The attributes of the default jumper are saved in the powerpcb.ini file. To change the setting of a specific pin in the design, open the pin Properties.

To change jumper settings:

- Click **Jumpers** from the **Setup** menu. The Jumpers dialog box appears. Use this dialog box to change the settings for jumpers to create and for jumpers existing in a design.

Jumper Limitations

- Jumpers do not follow the Body to Body Default Design Rule in batch and On-line DRC.
- Jumper silkscreens are not user configurable.
- You cannot use Find to search for jumpers.
- Cluster Place ignores jumpers.
- For CAM output, you must enable Component Outlines for the layer on which the jumper resides before jumper silkscreen outlines will plot. The outline for jumpers is set at 10 mils; you cannot modify this setting.

Creating Jumper List Reports

You can use the Reports dialog box to create a report containing information about all jumpers in the design.

To create a jumper list report:

1. **File** menu > **Reports**.
2. In the **Select Report Files for Output** list, select **Jumper List**.
3. Click **OK**.

   Result: The report is written to C:\PADS Projects\report.rep and displayed in the default text editor.

Related Topics

Reports

Setting Up Jumpers

When you add a jumper to the design, the Default jumper is used. You can change the Default jumper settings or change the settings for a specific jumper in the design. The attributes of the default jumper are saved in the powerpcb.ini file. To change the setting of a specific pin in the design, open the pin properties.

Use the Jumpers dialog box to set up and modify jumpers and jumper pad stacks. You can create and modify SMD jumpers (single layer jumpers) on the top or bottom mounting layer.

In this topic:

- Setting Up the Default Jumper
- Setting Up a Design Jumper
- Setting Up Jumper Pad Stacks

Setting Up the Default Jumper

You can change the settings of the Default jumper. The settings are saved in the powerpcb.ini file.

1. **Setup** menu > **Jumpers**.
2. In the **Apply to** area, click **Default**.
3. In the Jumper Sizes area, in the **Min. Length** and **Max. Length** boxes, type the minimum and maximum length of the Default jumper.
4. In the Jumper Sizes area, in the **Increment box**, type the increment, at which you can stretch the jumper between the minimum and maximum lengths.
5. Set up the pad stacks.
6. In the **Drill Size** box type the drill size if the jumper is a through hole jumper. Type a drill size of zero if you want a surface mount jumper with round pads.
7. Select the **Display Silk** check box to display the silkscreen outline for the jumper.

   Tips:

   - For CAM output, you must enable Component Outlines for the layer on which the jumper resides before jumper outlines will plot.
   - The outline for jumpers is set at 10 mils; you cannot modify this setting.

Setting Up a Design Jumper

You can change the settings of existing jumpers in the design.

1. **Setup** menu > **Jumpers**.
2. In the **Apply to** area, click **Design**.
3. Select a jumper name in the **Reference Name** list.
4. Set up the pad stacks.
5. In the **Drill Size** box type the drill size if the jumper is a through hole jumper. Type a drill size of zero if you want a surface mount jumper with round pads.

Restriction: Jumper Sizes and Display Silk settings are unavailable for Design Jumpers. To modify this information, use the Jumper Properties.

Setting Up Jumper Pad Stacks

You can set up pad stack information for jumpers or individual jumper pins.

1. In the **Pin** list, select the jumper (All) or the pin to change.

   Tip: You can add individual pins of the jumper to the list for customizing. Use the Add or Delete button to maintain the Pin list.

2. In the **Shape/Size/Layer** list select a layer on which to make jumper pad stack changes.

   Exceptions: When modifying Design jumpers, you can add individual layers of the design to the list for customizing the design jumper. Use the Add or Delete button to maintain the Shape/Size/Layer list.

3. In the Pad Parameters area, select the type of pad from the Pad Style list. Choose from (normal) pad, thermal (pad), or antipad.

   Restriction: You cannot create antipads on outer layers. When you select an outer layer (<Start> or <End>) in the Size/Shape/Layer list, Antipad is unavailable.

4. Select a Pad Shape. Choose from: Round, Square, Annular, or Odd.

   Restriction: Annular and Odd shapes are only available for the normal pad style.

5. In the Diameter box, type the diameter for the pad style.

   Exception: Thermals require additional settings. Read Design Rule Versus Pad Stack - Thermals and Antipads for details.

6. **(Square pads only.)** Select the Corner type and enter the Radius.

Tips:

- You can click Use Global Defaults to set the Thermal and Antipad shapes, of a layer and pin, to the default shapes specified by the design rules and the design Options > Thermals tab. This button is only available when the Pad Style list is set to Thermal or Antipad.

- Thermal and Antipad display configuration controls the size and shape of thermals and antipads used on split/mixed layers and CAM negative planes (for RS-274X output).
- You can select the Pad Size Relative to Drill Size check box to display inner and outer pad values relative to the drill size.
- You can view the effect of your pad settings in the Pad Preview display.

Related Topics

Modifying Jumper Properties

Modifying Jumper Name Properties

Modifying Jumper Pin Properties

Using Jumpers

Jumper Properties Dialog Box

Jumper Name Properties Dialog Box

Jumper Pin Properties Dialog Box

Modifying Jumper Properties

To modify jumper properties:

1. Select a jumper pin > right-click > **Properties**.
2. Click the **Jumpers** button. The Jumper Properties dialog box appears.

Alternatives:

- If you selected the Display Silk check box in the Jumpers dialog box, you can select the outline of a jumper, right-click and click Properties to access the Jumper Properties dialog box.
- You can click in the middle of the jumper to select it. Both jumper pins and the jumper name appears in the Selections color. Right-click and click Properties. The Jumper Properties dialog box appears.
- You can select the jumper name and then right-click and click Properties. Click the Jumper button to open the Jumper Properties dialog box.

Related Topics

Modifying Jumper Name Properties

Modifying Jumper Pin Properties

Using Jumpers

Jumper Properties Dialog Box

Jumper Name Properties Dialog Box

Jumper Pin Properties Dialog Box

Modifying Jumper Name Properties

To modify jumper name properties:

1. Select a jumper reference designator > right-click > **Properties**.
2. In the Jumper Name Properties dialog box, make edits.

   Alternative: Select the jumper. Right-click and click **Properties**. The Jumper Properties dialog box appears. Select the name to modify from the **Label** list. Click the **Label** button. The Jumper Name Properties dialog box appears.

Related Topics

Modifying Jumper Pin Properties

Modifying Jumper Properties

Using Jumpers

Jumper Properties Dialog Box

Jumper Name Properties Dialog Box

Jumper Pin Properties Dialog Box

Modifying Jumper Pin Properties

You can display the following information for the jumper pin: the net the jumper pin is part of, the reference designator, pin number, connection to which the jumper pin is attached, and the jumper pin coordinates.

To modify jumper pin properties:

1. Select the jumper pin.
2. Right-click and click **Properties**. The Jumper Pin Properties dialog box appears.

Note: If you change the pad stack of a jumper pin that is a locked test point, the Warning: Test Point Locked dialog box appears.

Related Topics

Modifying Jumper Name Properties

Modifying Jumper Properties

Using Jumpers

Jumper Properties Dialog Box

Jumper Name Properties Dialog Box

Jumper Pin Properties Dialog Box

During Routing

Using the Trace Length Monitor

The trace length monitor calculates and shows trace length as you route. As you move the pointer and route traces, the routed and unrouted length of nets or pin pairs and associated rules are shown near the pointer and on the status bar.

The trace length monitor is available for the following commands:

Route/Add Route
Dynamic Route
Move Segment
Move Corner
Move Via

Procedure

To turn the trace length monitor on:

1. **Tools** menu > **Options** > **Routing** tab.
2. Select the **Show Trace Length** check box.
3. Click **OK**.

Tip: You can also use the shortcut key Ctrl+PageUp to turn the trace length monitor on or off. Turning the trace length monitor on or off does not end the current routing command; you can continue to route.

This turns the display of the trace length monitor on or off at the pointer only. The trace length monitor always displays on the status bar.

Related Topics

Trace Length Monitor in the *Concepts Guide*

Changing the End of the Connection You're Routing

When you start routing, PADS Layout automatically chooses the end from which you start. If you want to start from the opposite end of the connection, use Swap End. Also at any point while you are routing, you can preserve what you've already routed and switch to the opposite end of the connection to continue routing.

Restriction: Swap End is available only in Add Route and Dynamic Route modes.

Procedure

- While routing, right-click and click **Swap End**.

To Select a Starting Layer for a Trace

To set the current routing layer before you start routing, do one of the following:

- Click the layer from the **Layer** list on the standard toolbar. A layer's current direction setting appears next to its entry in the Layer list.

  Or

- Change the current layer by typing the **L** modeless command, typing the layer number, and pressing **Enter**.

You can change the layer's current direction setting by typing the **LD** modeless command. When you select a connection and select a routing mode, the route begins on that layer. When you end a route on a different layer than where you began it, the layer you ended on becomes the active layer and appears in the Layer list box. The next connection you select starts on this active layer.

Adding a Via While Routing

Add a via while routing to either continue routing on another layer or to connect to a plane area.

Requirements

- You must have created a via for use with your design. For more information, see Creating a Through-hole Via, Creating a Partial Via.
- The via must be available for use, and both source and destination layers must be available for routing according to the applicable Routing Rules.

Procedure

Use one of the following three methods while routing:

- Right-click and click **Add Via** to continue routing on another layer.
- Click to place a tack and then change to the desired new layer to continue routing.

  Result: The via is added automatically in the location of the tack.
- Right-click, point to End Via Mode, and then click **End Via to set the mode**. Right-click and click **End to end the trace with a via.**

Result: The trace ends with a Via. If there is a plane area on the path of the via, it will either connect with a thermal connection if necessary or add an antipad if it shouldn't connect.

Troubleshooting

- If the via is a restricted via, ensure the requirements to this procedure are in place.
- If a violation is detected, the via grid or a clearance rule could be preventing the placement of the via. Also, the layer you are routing to could be unavailable for routing. Ensure the requirements to this procedure are in place.

Related Topics

Troubleshooting Routing on Another Layer

Setting a Via Type

Using End Via Mode

Using the Layer Pair

To Create Arcs

To enter arced corners during routing:

1. While the trace is in progress, right-click and click **Add Arc**. The segment extending from the last corner is converted to an arc.
2. Move the pointer to adjust and indicate the radius.
3. Click to complete the arc. Routing continues from this point.

See also: To Stretch an Arc or Miter

To Create Miters

You can accomplish a smaller, smoother arced effect by mitering a routed corner.

To create a mitered corner:

1. **Tools** menu > **Options**.
2. Set the **Miter** type to **Arc** in the Design page of the Options dialog box.
3. Set the **Ratio**.

   Ratio is the ratio of the arc radius to the trace width; for example, for a 12-mil trace, a ratio of 1 produces a radius of 12; a ratio of 2 produces a radius of 24.
4. To apply the miter, select the corner, right-click, and click **Add Miter**.

Auto Miter

To automatically miter corners of drafting shapes as you create them, select **Auto Miter** in the **Design page** of the Options dialog box. The **Miters** field determines the size and shape of the miter. You can also select or clear auto-miters from the shortcut menu while a drafting command is active.

Use the **Convert to Arc** command to create arcs from straight-line segments.

See also: To Stretch an Arc or Miter

To Change the Layer While Routing

To change layers while routing, use the **L** modeless command while the trace is active:

- Type **L**, the new layer number, and press Enter.

PADS Layout installs a via at the last corner and move your trace to the selected layer. The type of via placed is determined by the Via Type selection dialog box settings. The trace color changes accordingly and the new layer appears in the Layer list box.

You can also right-click and click **Layer**. If you are changing repeatedly between two layers, you can set a layer pair and use the layer toggle command F4.

Related Topics

Using the Layer Pair

To Change the Via Type While Routing

To change the via used while routing:

1. Right-click and click **Via Type**. The Vias dialog box appears.
2. Click a new via type from the **Via Mode** area.
3. If the via is a through hole, click the via to use in the **Via List** list.
4. Click **OK**. The new via type is used until you change it.

If you click automatic, PADS Layout selects a via based on:

- Which vias are allowed for the net in the Routing Rules dialog box.
- Which vias are legal for the layer change according to the Drill Pairs Setup dialog box.

If more than one via qualifies on both counts, the product uses the smaller drill diameter and/or smaller pad size.

Other options are:

Partial	Automatic search is limited to partial vias.
Through	List of through vias becomes active. Select a via for the default installed when you change layers.

To Change the Trace Width While Routing

When you change the trace width, the current routing width changes from the last corner, leaving segments before the last corner at their original width. All subsequent segments draw at the new width, until you reach the next pin in the net. The width stays associated with the connection until you change it.

The new width becomes the effective width for the rest of the connection, but it does not change the Recommended Trace Width setting for the net. If you end with a partial route and use it later, the width you set separately is still in effect.

To change the width of a route in progress:

1. Type the **W** modeless command.
2. Type the new width and press Enter. The line width area on the status bar changes accordingly.

You can also use the Grid/Width dialog box to change the width on the trace you are currently routing.

To change the width of a trace once it is routed:

1. Select the item.
2. Right-click and click **Properties**. You can also use **Find** from the **Edit** menu to find all traces of a similar width, select them, and change them to a new width.

See also: To Change the Width of an Existing Trace

Troubleshooting Routing on Another Layer

You can use a via to switch to another layer while routing. But there are many rules or settings than can prevent you from routing on another layer. If you have trouble, here is a list of things to check:

- Is the via type in the Selected layers list? In the Routing rules at each level of the hierarchy, you can disable a via for routing by keeping it out of the Selected vias list.

- Is the layer in the Selected layers list in the Layer biasing? In the Routing rules at each level of the Rules hierarchy, you can disable a layer for routing by keeping it out of the Selected Layers list in the Layer biasing area.
- Is the layer check box cleared in the Selection Filter? In the Selection Filter, on the Layer tab, you can clear the check box of a layer. Nothing on that layer is selectable - including the pad of the via from which you are trying to route.
- Is the active layer not coming to the front? In the Global Options, you can select the Active layer comes to front check box to bring the layer forward. Then you can select items on that layer more easily.
- Is there a conditional rule which prevents this net from routing on this layer, or near these components?

Ending a Trace on a Different Net

To end a trace on a segment with a different netname, use the ECO toolbar routing command. PADS Layout assumes you are intentionally joining two nets to make one, and consider it an ECO. It is recorded in the database as such. You must assign a new netname: either use one of the names from the nets you just combined or use a third name.

See also: Adding a Connection in ECO Mode, Adding a Route in ECO Mode

Selecting Objects Among Others

You can use additional selection filter shortcuts after selecting a design object. Right-click and click a selection shortcut.

In this topic:

Selecting Nets From an Electrical Object

You can select an assigned net of an electrical object quickly. Electrical objects include trace objects, pins, copper objects, planes, and unroutes.

To select an entire net, including routes, connections, and pins:

1. Select an object.
2. Right-click and click **Select Nets**, or press **F6**.

Selecting Pin Pairs From an Object

To select all pin pairs attached to a pin, trace segment, corner, tack, or via:

1. Select an object.
2. Right-click and click **Select Pin Pairs**, or press **F5**.

Selecting Classes from Nets

A class is a collection of nets with a common set of design rules. You can define classes graphically or by using the Class Rules dialog box.

See also: Design Rule Hierarchy in the *Concepts Guide*

If you have created net classes, you can quickly select all items such as pins, traces, and unrouted pin pairs within them, or determine the class of a net:

1. Select a net.
2. Right-click and click **Select Class**.

Selecting Groups from Pin Pairs

A group is a collection of pin pairs with a common set of design rules. You can define groups using the Group Rules dialog box.

See also: Design Rule Hierarchy in the *Concepts Guide*

To select all pin pairs in a group:

1. Select a pin pair of a group.
2. Right-click and click **Select Group**.

Selecting Drafting Objects from Segments/Corners

To select the entire drafting object from one of its segments or corners:

1. Select the drafting segment or corner.
2. Right-click and click **Select Shape**.

After Routing

The following section details tools to help you clean up or modify routed trace patterns, completely reroute, or protect routes from further modifications.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation. You can, however, copy trace patterns from a physical design reuse.

To view obstacles that may lie under traces on the current active layer use Transparent Mode, modeless command **T**.

To Protect Routes

To protect a route you must select the object and use the Properties dialog box to enable protection. You can set protection, for fully routed connections and the routed portion of partial routes, at the pin pair or net level.

See also: Route Protection in the *Concepts Guide*

Protecting Trace Segments

1. Select a trace segment or unroute > right-click > **Select Pin Pair** > right-click > **Properties**.
2. Select the **Protect Routes** check box.
3. Click **OK**.

Protecting Entire Nets

1. Select a trace segment or unroute > right-click > **Select Net** > right-click > **Properties**.
2. Click **Protect Routes**.
3. Click **OK**.

To Protect Unroutes

To protect unroutes you select the object and use the Properties dialog box to enable protection. You can set protection, for fully unrouted connections and the unrouted portion of partial routes, at a pin pair or net level.

See also: Route Protection in the *Concepts Guide*

Tip: You cannot protect the unrouted portion of a partial route unless you first protect the routed portion.

To protect an unroute segment:

1. Select an unroute.
2. Right-click and click **Properties**.

   See also: Pin Pairs Properties Dialog Box
3. To protect a fully unrouted connection click **Protect Unroutes**. If the net contains partial routes this option is unavailable.

   To protect the unrouted portion of a partial route, click **Protect Routes** and then click **Protect Unroutes**.
4. Click **OK**.

To protect an entire unrouted net:

1. Select an unroute.
2. Right-click and click **Select Net**.
3. Right-click and click **Properties**.

   See also: Net Properties Dialog Box
4. Click **Protect Unroutes**.
5. Click **OK**.

Moving a Trace Segment to Another Layer

Instead of rerouting an entire trace, you can move a trace segment to another layer and have the vias added automatically.

Procedure

1. Select one or more trace segments.
2. Right-click and click **Properties**.
3. In the Trace Properties dialog box, select the layer you want from the Layer list.
4. Click **OK**.

Result

The trace segment moves to the selected layer and vias are automatically placed at the segment end points.

Tip: If nothing happens, design rules may have been violated. For example, there may not be enough room for a via.

To Reroute with Route or Dynamic Route

You can reroute traces by starting and ending a new replacement trace anywhere along the existing trace. You can start and end reroutes on segments, pins, and vias. You must, however, meet two conditions:

- The existing and new traces must have the same netname.
- The net must be enabled for copper sharing.

  To enable copper sharing, use the Routing Rules dialog box for the net and select **Copper Sharing**. To enable sharing for all nets, click **Default** in the Design Rules dialog box and select **Copper Sharing** in the **Routing** area.

To reroute a route starting on a segment:

1. Select the segment to reroute.
2. Right-click and click **Route**. The new trace begins at the selection point.

   You can end on a trace when the pointer changes to a bull's-eye as you move the route in progress over the segment into which you want to T. If you have created a connection loop, the “Delete Loop?” message appears. Click **OK** to delete the highlighted segment or click **Next** to cycle through the other segments in the loop. Click **OK** to delete the highlighted segment.
3. To complete the route, double-click or right-click and click **Complete**.

Tips:

- If you reroute a segment that has a via or jumper pin that is a locked test point, the locked test point is not deleted. The test point remains connected by an unroute.
- If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

Use Transparent Mode, modeless command **T**, to view obstacles that may lie under traces on the current active layer.

To Reroute with Sketch Route

Use Sketch Route, when using Dynamic Route, to reroute a set of trace segments between two pins. A pin in this case may be a component pin, via, or tack.

Remember that Sketch Route cannot move a tack, so most traces that travel in the wrong direction, or have tacks for any other reason, are ineligible for Sketch Rerouting.

To reroute a trace using sketch:

1. Click **Prevent Errors** in the On-Line DRC area of the Design page (Options dialog box).

   See also: Design Rule Checking in the *Concepts Guide*

2. Select the first segment of the series you want to edit.
3. Right-click and click **Sketch**. A thin dotted line appears that moves with the pointer. Use this line to mark the general location of the new trace. You don't have to enter any corners.
4. Click to indicate the end of the sketched route. This completes the trace, routing through the defined path.

Sketch in Verb Mode

The Sketch Route button in the Design toolbar provides the same rerouting functionality, but uses a different set of procedures to start the rerouting process:

1. Click **Prevent Errors** in the On-Line DRC area of the Design page (Options dialog box).
2. Click the **Sketch Route** button.
3. Select the trace to edit. A thin dotted line appears that moves with the pointer. Use this line to mark the general location of the new trace. You don't have to enter any corners.
4. Click to indicate the end of the sketched route. This completes the trace, routing through the defined path.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

Using Smoothing Controls

Use smoothing options to smooth trace patterns (removing unneeded corners and segments and centering trace patterns between route obstacles), to protect traces for controlled length nets, to perform smooth operations after bus routing, and to smooth pad entry and exit.

These options, which are located on the Routing / General page of the Options dialog box, affect the results of the Smooth, Autoroute, and Bus Route commands.

To Smooth Trace Segments

Smoothing removes unnecessary segments from a trace and converts ninety-degree corners to forty-five degree corners whenever possible. This option is only available when you use Dynamic Routing.

To smooth trace segments:

1. Set the On-line DRC setting to **Prevent Errors**.
2. Select a trace segment.
3. Right-click and click **Smooth**.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

To Change the Pad Entry Angle

The Trace/Line Angle setting in the Design page of the Options dialog box determines the angle of pad entry when you complete a trace. You can also use Trace/Line Angle in the Status window. After the trace completes, you can change the pad entry angle into the pin.

1. Select the segment.
2. Right-click and click **Pad Entry**. The final segment changes its orientation and adjusts the attached segments accordingly if possible.

   If the new pad entry causes a clearance error, the process is canceled.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

To Copy and Paste Trace Patterns

You can reproduce repetitive routing patterns by making one example of a segment pattern, copying it, and pasting it on any similar connections. Layer changes and vias in the path are included in the selection. Memory patterns or SMD fanouts are ideal applications for copied routes.

To copy and paste a route:

1. Select a segment or pin pair.
2. Click **Copy** from the **Edit** menu, or press Ctrl+C. All included segments and vias are animated, or copied and attached to your pointer in Move mode.
3. If necessary, use the shortcut menu commands to rotate or flip the copy.
4. Indicate the location so it joins the pins to which you want to paste. A copy is pasted and another copy remains attached to the pointer so you can paste it again.

   The pointer then snaps automatically to a point the same distance from and in the same direction as the last placement. This makes it easy to install repetitive route patterns.
5. Press **Esc** to cancel further copies.

A copied trace only pastes to a valid electrical connection: you cannot paste to empty space. If the copied traces are shorter than the target pin pair, a connection is created from the end of the copy to the other pin of the pair.

Related Topics

To Copy and Paste a Route in BGAs

Cut, Copy, and Delete in the *Concepts Guide*

To Create Route Loops

You can reroute from point to point along the same net without deleting the previous path. This causes the new route to branch off the existing route, creating a loop.

Procedure

1. Indicate the location on the route segment where you want the branch to occur.
2. Right-click and click **Route Loop**. A new route starts at the selection point.
3. Indicate the location for each corner of the new route.
4. Indicate the location where the loop branches back into the existing route.

You can also start Route Loop using the Ctrl+J shortcut keys.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

To Move a Trace Segment

To move a trace segment:

1. Select the trace segment.
2. Use one of these methods to start the move.
    - Right-click and click **Move**.
    - Press Ctrl+E.
    - Select the trace segment and drag. Use the Global / General page of the Options dialog box to click a drag and drop method, or to prohibit pointer-drag moves.

      The selection attaches to the pointer.
3. Click to indicate the route location.

"Shove" When Moving

If you use the Dynamic Route tool, you can use the shortcut menu to enable or disable trace shoving, or moving, when you move traces. If you want to move traces aside, temporarily creating overlaps, right-click and click Shove (to remove the check mark) when the trace is in Move mode.

Tips:

- If you set DRC to Prevent or Warn, you can't create a clearance violation when you relocate the trace.
- If you move a trace that is attached to a via or pin that is a locked test point, a Warning dialog box appears informing you of changes to the test point and giving you options.

  See also: Troubleshooting
- If you try to modify route objects in a physical design reuse, the message "Reuse objects cannot be modified. Break the reuse first" appears. Click OK to cancel the editing operation.

Use Transparent Mode, the modeless command **T**, to view obstacles that may lie under traces on the current active layer.

To Delete a Trace Segment

To delete a trace segment:

1. Select the segment.
2. Press **Del**, or click **Delete** from the **Edit** menu.

When you delete a trace segment, a connection line is restored.

Tips:

- If you delete a trace that is attached to a via that is a test point, the via is automatically deleted with the trace unless the test point is locked. If the test point is locked, it remains connected by an unroute.
- If you try to delete a segment that is part of a physical design reuse, the message "The command cannot be applied to reuse elements" appears in the status bar.

To Unroute a Segment Attached to a Pin

You can unroute the first segment attached to each pin of a component. Use Unroute Attached Segments to reroute each connection that terminates at the part. When segments are unrouted, tacks appear at the end of the connection where it meets the route. If the unrouted segment ends at a via, a connection is created between the via and the unrouted pin.

To unroute a segment attached to a pin:

1. Select the component from which you want to unroute attached segments.
2. Right-click and click **Unroute Attached Segments**. The first segments are unrouted and tacks appear at the end of the routes.

Tip: If you try unroute segments that are part of a physical design reuse, the message "The command cannot be applied to reuse elements" appears on the status bar.

To Change the Width of an Existing Trace

You can change the width for routed traces at the segment, pin pair, or net level, using the Properties dialog box.

Modifying the width of a segment

1. Select a segment > right-click > **Properties**.
2. Type the new width in the **Width** field and click **Apply** or **OK**.

Modifying the width of pin pair or net

1. Select a segment > right-click > **Select Pin Pair** or **Select Net**.
2. When the pin pair or net is highlighted, right-click and click **Properties** or click the **Properties** button on the toolbar. The Pin Pair Properties or Net Properties dialog box appears.

   See also: Pin Pairs Properties Dialog Box
3. Type the new width in the **Width** box and click **Apply** or **OK**.

Tip: If you try to modify route objects in a physical design reuse, the message "Reuse objects cannot be modified. Break the reuse first" appears. Click OK to cancel the editing operation.

To Convert a Trace Corner to an Arc

Use Convert to Arc to create an arc from two intersecting route segments. The Angle box in the Miters area of the Options Dialog Box, Design page controls which trace corners will be converted to arcs. For example, if the Angle box is set to 90 degrees, then only trace corners that form angles less than or equal to 90 degrees can be converted to arcs.

To convert trace corners to arcs:

1. Select the corner of a route.
2. Right-click and click **Convert to Arc**.

3. Move the pointer to adjust the radius of the new arc and click to finish.

Use Stretch from the routing shortcut menu to adjust the radius of a created arc or to convert the arc back to a corner.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

Related Topics

To Create Miters

To Stretch an Arc or Miter

Use Stretch to expand or contract a miter or arc on a route segment.

To adjust an arc or miter:

1. Select the miter segment or arc.
2. Right-click and click **Stretch**. The object attaches to your pointer.
3. Move the pointer to expand or contract the object.
4. Click to finish.

Click outside the point of intersection of the two lines to return the arc or miter to a corner.

Tips:

- If you move or stretch a route that is attached to a via or pin that is a locked test point, a Warning dialog box appears informing you of changes to the test point and giving you options.

  See also: Troubleshooting
- If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

Using Stretch to Move a Route Segment

You can also use Stretch to move a straight route segment that has corners at the end points. The corner angles are maintained and the length of the segment is adjusted during the stretch. **Exception:** If you stretch a segment so that it reaches another segment’s corner, corner angles may change.

Restriction: You cannot use Stretch to move a straight route segment with an arc at its end point.

Tip: Use the Pull Arc and Move Miter commands to modify arcs and miters for all drafting objects.

Related Topics

To Create Arcs

To Create Miters

To Move a Corner

You can move any corner on a trace or drafting object:

1. Select the corner to move.
2. Drag the corner using the mouse.
3. To convert the corner to an arc, right-click and click **Convert To Arc**. The corner becomes an arc and remains attached to your pointer. Move the arc to the desired position, and click to complete.
4. To shove other routes as you move the selected corner, right-click and click **Shove**.
5. To limit the via or tack to horizontal or vertical segments, right-click and click **Orthogonal**.
6. To allow 45 degree segments, right-click and click **Diagonal**.
7. To allow segments of any angle, right-click and click **Any Angle**.
8. To ignore clearance rules regardless of the On-line DRC setting in the Design page in the Options dialog box, right-click and click **Ignore Clearance**.

Tips:

- If you click the Move button in the Design toolbar, a click attaches the corner to your pointer.
- If you select Keep Traces Attached After Move Component in the Design page of the Options dialog box, the segment attaches to your pointer when you release the button.
- If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

Related Topics

To Delete a Corner

Modifying Drafting Corner Properties

To Move a Via or Tack

To move a via or tack:

1. Select the via or tack.
2. Drag the via or tack using the pointer.
3. To shove other routes as you move the selected corner, right-click and click **Shove**.
4. To limit the via or tack to horizontal or vertical segments, right-click and click **Orthogonal**.
5. To allow 45 degree segments, right-click and click **Diagonal**.
6. To allow segments of any angle, right-click and click **Any Angle**.
7. To ignore clearance rules regardless of the On-line DRC setting in the Design page in the Options dialog box, right-click and click **Ignore Clearance**.

Tips:

- If you move a via that is a locked test point, a Warning dialog box appears informing you of changes you are making to the test point and giving you options.

  See also: Troubleshooting
- If you move a via that is a test point into an area defined as a test point keepout area, and DRC mode is ON, a message appears and the test point is cleared.
- If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

Deleting Dangling Routes

Dangling routes are stubs or spurs off of traces that are not tied to any pin by a ratsnest. By contrast, partial routes are uncompleted routes where the ratsnest flightline is still visible.

In PADS Layout you can end the route on a test point by choosing End Test Point from the Route shortcut menu. This creates a dangling via and dangling route. If you later want to remove the dangling via, right-click and click Select Dangling Routes. PADS Layout allows you to select all of these route types on the board.

You can selectively unroute or delete the dangling via and dangling route.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the modifying operation.

To Split a Trace

The Split command divides a selected line segment into two segments.

To split a segment:

1. Select the segment to split.
2. Right-click and click **Split**. The segment dynamically attaches to the pointer.
3. Click to indicate where to split the segment. The segment is divided into two segments.

Split in Verb Mode

4. Click the **Split** button from the **Design** toolbar.
5. Select the trace to split. The segment dynamically attaches to the pointer.
6. Click to indicate where to split the segment. The segment is divided into two segments.

Swapping Segment Sides while Splitting

While splitting a trace, you can switch the trace side that is attached to the pointer.

1. Before completing the split, right-click and click **Swap**. The pointer attaches to the other end of the segment.

   The current Angle Mode setting effects the movement of the attached segments.
2. Continue to split the segment as described in the previous section.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

To Add a Corner to a Trace Segment

Add Corner divides the selected line segment into two segments.

To add a corner:

1. Select the segment to which to add a corner.
2. Right-click and click **Add Corner**. The segment dynamically attaches to the pointer.
3. Click to indicate the location for the new corner. The segment is divided into two segments.

Add Corner in Verb Mode

4. Click the **Add Corner** button from the **Design** toolbar.
5. Select the trace to add the corner to. The segment dynamically attaches to the pointer.

6. Click to indicate the location for the new corner. The segment is divided into two segments.

Tips:

- The current Angle Mode setting effects movement of the attached segments.
- When On-line DRC is set to Prevent Errors and errors occur, (guard band appears), the corner is added at the closest legal position to the indicated point.
- If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

Adding Vias to an Existing Trace

The Add Via command adds a feed-through to the selected trace at the selection point location. This action does not change the current layer setting.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the modifying operation.

Using Stitching Vias

Use stitching vias to interconnect copper areas of nets including copper pour, split planes, or CAM planes. Stitching vias are not considered orphaned vias and are not deleted during routing optimization.

Exception: While any number of vias can be added to a trace segment, corner, or end point, it is not considered a stitching via.

In this topic:

- Add Stitching Vias
- Select Stitching Vias
- Change Stitching Via Types
- Delete Stitching Vias
- Convert Routing Vias into Stitching Vias

Add Stitching Vias

Stitching vias must be associated with a net. They are added to the design through the net selection of an object in the design.

To add a stitching via:

1. With nothing selected, right-click and click **Select Nets**.
2. Select a net object.

   Tip: Net objects include trace segments, vias, pins, unroutes, copper, copper pours, and split planes.
3. Right-click and click **Add Via**. The new via is attached to the pointer.
4. Click to place the stitching via. Another stitching via attaches to the pointer. You can place vias repeatedly.

   Tip: If unroutes are visible for the selected net, a new unroute will connect to the stitching via.
5. Right-click and click **Cancel** to stop adding stitching vias.

Result: When you add a via to an unroute, the result is a via with a Stitching setting enabled. When you add a via to a trace segment or trace corner, the result is a plain via (unglued). By default, stitching vias added to the design are always given the Stitching setting as they are added. Vias added to traces are not.

Warning: You can prevent the deletion of stitching vias during ECO operations such as Delete Component, Delete Connection, and Delete Pin, as well as during a Change Part or an Unroute operation. To preserve stitching vias select the Keep Stitching Vias check box in the Group Editing area on the Design page of the Options dialog box.

Select Stitching Vias

You must enable the selection of stitching vias before you can select a stitching via.

To select a stitching via:

1. With nothing selected, right-click and click **Filter**.
2. In the Selection Filter, on the Object tab, click **Stitching**. It is a subset of Vias.
3. Click **Close**.
4. Select a stitching via.

Change Stitching Via Types

You can use any via type, including SMD, through-hole, or partial as a stitching via. If the default via is not the correct via, you can modify the via type.

1. Select one or multiple stitching vias.
2. Right-click and click Properties.
3. In the Via Name list, select the correct via.

Tip: You can use the Pad Stacks dialog box to create new vias.

Delete Stitching Vias

1. Select the stitching via to delete.
2. Press **Delete**.

Convert Routing Vias into Stitching Vias

Vias added to a trace segment, corner, or end point, are not considered stitching vias. Like stitching vias, these routing vias can be used to interconnect areas of a net, but they are considered plain vias. You enable the Stitching setting for a routing via to make it a stitching via.

To convert a via into a stitching via:

1. Select one or multiple vias.
2. Right-click and click **Properties**.
3. Select the **Stitching** check box.
4. Click **OK**.

Related Topics

To Glue a Via

Adding Stitching Vias in the *Concepts Guide*

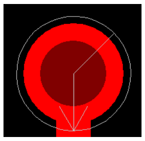
To Add a Test Point

You can manually add a test point along an existing segment or trace of a route. Test points, by default, are not glued and are accessible from the bottom.

To add a test point to an existing segment or trace:

1. Select a trace segment or corner. You cannot use multiple selection when adding test points to a route.
2. Right-click and click **Add Test Point**. A test point is added to the route. The via inserted is the type that you click in the Options tab of the DFT Audit dialog box.

Tip: When the via or pin is flagged as a test point, and Show Test Points is checked on the Routing tab of the Options dialog box, an arrow is drawn on it in the design:



In Verb Mode

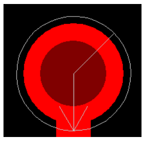
You can also add test points in Verb mode:

1. Click the **Design toolbar** button.
2. Click the **Add Test Point** button on the **Design** toolbar.
3. Select any trace segment or corner to which you want to add a test point. A test point is added to the route. The via inserted is the type that you click in the Options tab of the DFT Audit dialog box.

Tips:

- You can also make an existing via or pin a test point.

  See also: Via Properties Dialog Box, Pin Properties Dialog Box, Jumper Pin Properties
- If you try to add a test point in an area defined as a test point keepout area and DRC mode is set to Prevent Errors, the message "Test point keepout violation" appears and the test point is not added.
- If you try to modify route objects in a physical design reuse, the message "Reuse objects cannot be modified. Break the reuse first" appears. Click OK to cancel the editing operation.
- **Tip:** When the via or pin is flagged as a test point, and Show Test Points is checked on the Routing tab of the Options dialog box, an arrow is drawn on it in the design:



To Delete a Corner

To delete a trace or drafting corners:

1. Select the corner.
2. Click **Delete** from the **Edit** menu. The two segments joined at the corner become one segment.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

Related Topics

To Move a Corner

Modifying Drafting Corner Properties

To Delete a Via

To delete a via:

1. Select the via.
2. Click **Delete** on the **Edit** menu. The via is deleted. The layers of the attached segments are changed if necessary.

Tip: If you try to modify route objects in a physical design reuse, the message “Reuse objects cannot be modified. Break the reuse first” appears. Click OK to cancel the editing operation.

To Glue a Via

A copied trace only pastes to a valid electrical connection; you cannot paste to empty space. If the copied traces are shorter than the target pin pair connection, the ratsnest is continued from the end of the copy to the other pin of the pair.

To fix a via against movement for any reason:

1. Select a via > right-click > **Properties**.
2. Select the **Glued** check box.
3. Click **Apply** or **OK**.

You can fix a via's location to establish it as a test point on the fabricated board. The points of a bed of nails test apparatus are matched to the test point locations. So if you redesign and remanufacture the board, the glued test points prevent the vias from moving in the new design, thus preventing costly retooling of the test equipment.

If you click Test Point, you include the via's location in a report file in a Test Point section. The Test Point check box also indicates if the via was automatically installed and glued as a test point by an autorouter. Installing test points this way returns test points from the router with checked Glue and Test Point check boxes.

To Delete a Miter From a Path

A miter is a diagonal segment or arc that replaces a corner. You can delete all miters from one or more routed pin pairs, drafting paths, or polygons:

1. Select the routed pin pairs, drafting paths, or polygons.
2. Right-click and click **Delete Miters**. All miters in the selected path are deleted and replaced by a 90-degree corner.

To Delete a Route from a Pin Pair

To delete all traces and vias in a routed pin pair:

1. Select the pin pair.
2. Press **Del**, or click **Delete** from the **Edit** menu. A ratsnest line between the two pins replaces the routed pin pair.

Connecting a Net with a Plane

Establishing a plane area and connecting the appropriate nets to it is usually one of the first routing tasks in the design process. The following two methods establish plane areas:

- **Define a Layer as a Plane Type**

  Use Layer Definition to set an entire layer to Plane type. Use Net Associations to define one or more netnames to connect to this layer with thermal reliefs.

- **Draw a Copper Pour Area**

  Use this method when you want insulated traces passing through the plane area. The copper pour area does not require you to define its layer as a plane layer. Instead, assign the netname of the net to which you want to connect to the copper pour outline.

  See also: Connecting a Net with a Plane in the *Concepts Guide*

To Connect SMD Pads to Planes

Routing an SMD pad to a plane involves placing a via to the plane under or somewhere near the pad. When vias are in plane nets, they receive a thermal relief pad for the plane layer based on the pad outer diameter for the layer, defined in the via's pad stack.

To set the display and connect the SMD pads:

1. **View** menu > **Nets**.
2. Select **Default** in the **View List** list.
3. Select **Traces Plus the Following Unroutes**, if necessary, and set **View Details** to **None**.
4. Click **Apply**. This removes all the unrouted connections from the display.

5. Locate your plane nets in the **Net List** list and add them to the **View List** list.
6. Select the plane nets and set **View Details** to **All Except Connected Plane Nets**. This shows full pin-to-pin connections and partially routed copper traces, but not partial connections.
7. Click **Apply**. Now only the plane net connections are visible.

When you install the via, either under the pad or a short distance away, the connection for that pin-to-pin connection disappears if you route the connection to the plane layer.

See also: Checking the Plane Connection for Continuity

Converting a Trace to a Copper Chamfered Path

You can convert traces to chamfered copper. Converting traces to a copper chamfered path has two advantages over simply creating a copper chamfered path. You can also use the more powerful interactive router to initially route the trace and when you convert the trace to a chamfered path, the net assignment is automatically made.

1. Select a pin pair, multiple pin pairs, or a net.

   Restriction: Unrouted or partially routed pin pairs, or pin pairs belonging to reuse blocks, are excluded from selection.

2. Right-click and click **Convert to Chamfered Paths**.
3. In the Convert Pin Pairs to Chamfered Paths dialog box, select the pins pairs desired. Use Ctrl+click, Shift+click, or the Select All and Unselect All buttons as shortcuts.

   Restriction: Only the pins pairs selected in the design appear in the Selected pin pairs list.

4. In the Conversion parameters area, in the Polygon outline width box, type a width value for the width of the copper outline.

   Tip: Since copper is created with an outline and a fill, you can specify a very narrow outline width to achieve very sharp corners. Decrease the value for sharper corners and increase the value for more blunt corners. All corners are rounded with a radius equal to one half of the outline width.

5. Select the **Use trace width** check box to use the trace width as the width of the chamfered path. Where multiple trace widths exist, the actual trace widths are used. The Selected trace widths value at the top of the Conversion parameters area displays the range of trace widths of the items in the Selected pin pairs list.

   or

   Clear the **Use trace width** check box and enter a value in the Converted path width box.

6. In the Corner chamfer width ratio box type a value specifying the ratio of the chamfered corner width to the chamfered path width. If the ratio is 1.0, the width of the chamfered corner is the same as the chamfered path. Reduce the radio for a more narrow chamfered corner.

7. Clear the **Chamfer corners with angles less than or equal to** check box to chamfer only angles less than 90 degrees (acute angles).

   or

   Select the **Chamfer corner with angles less than or equal to** check box to specify an angle between 90 and 180 degrees as the upper limit beneath which all angles are chamfered. Outside corners less than 90 degrees are always chamfered.

8. Click **OK**.

   Result: The trace is converted to a chamfered path and original trace segments are unrouted. Existing vias are retained and are glued.

Related Topics

Creating Copper Chamfered Paths

Restoring Traces After Conversion to Copper Chamfered Paths

Modifying Drafting Object Properties

Restoring Traces After Conversion to Copper Chamfered Paths

Original traces are unrouted after conversion to copper. You must re-route traces to replace chamfered paths.

1. Use the Select Nets selection filter to select the net.

2. Right-click and click **Select Chamfered Paths**.

3. Press **Delete**.

4. Re-route the pin pair, pin pairs, or net.

   Restriction: Vias in the chamfered path are not deleted since they are glued during conversion from trace to chamfered path.

Editing Properties of Routing Objects

In this section:

- Modifying Net Properties

- Modifying Pin Properties
- Modifying Pin Pair Properties
- Modifying Trace Corner or Tack Properties
- Modifying Via Properties
- Modifying Trace Segment Properties

Modifying Net Properties

To modify net properties:

1. Select a pin pair > right-click > **Select Net** > right-click > **Properties**.
2. Modify settings in the dialog box, if necessary.
3. Click **OK** when finished.

Tip: Several of the options in this dialog box are unavailable if the net is part of a physical design reuse.

Related Topics

Net Properties Dialog Box

View Nets Dialog Box

Modifying Pin Properties

To modify pin properties:

1. Select a pin.
2. Right-click and click **Properties**. The Pin Properties dialog box appears. You can modify the pad stack for the via.

Tips:

- If you modify a pin that is a locked test point, a Warning dialog box may appear informing you of changes you are making to the test point and giving you options.

  See also: Troubleshooting
- Several of the options in this dialog box are unavailable if the pin pair is part of a physical design reuse.

Modifying Pin Pair Properties

To modify pin pair properties:

1. Select a pin pair.
2. Right-click and click **Properties**. The Pin Pair Properties dialog box appears.
3. Modify the settings in the dialog box, if necessary.
4. Click **OK** when finished.

Related Topics

Pin Pairs Properties Dialog Box

Modifying Trace Corner or Tack Properties

To modify trace corners properties:

1. Select a trace corner > right-click > **Properties**.
2. In the Connection area, select the connection to which the trace corner belongs.
3. In the X and Y boxes, type the X and Y locations.
4. Click the **Pin Pair** button to modify settings for the pin pair to which the trace corner belongs.
5. Click the **Net** button to modify settings for the net to which the trace corner belongs.
6. Click the **Rules** button to modify the pin pair rules for the trace corner.
7. The other areas of the dialog box provide information about the trace corner:
    - Net—Shows the net to which the trace corner belongs.
    - Protect Status—Shows whether the net (to which the trace corner belongs) is protected.
    - Layer—Shows the layer on which the trace corner is located.
    - Trc/Trc Clearance—Shows the trace to trace clearance rule.
    - Rule Hierarchy—Shows the level of the hierarchy at which the pin pair rules are set.

Tips:

- If you select another trace corner while the dialog box is open, the selected corner updates with the information.
- Several of the options in this dialog box are unavailable if the corner is part of a physical design reuse.

Related Topics

Modifying Pin Pair Properties

Modifying Net Properties

Design Rule Hierarchy in the *Concepts Guide*

Modifying Via Properties

To modify via properties:

1. Select a via.
2. Right-click and click **Properties**. The Via Properties dialog box appears. You can modify the following information about all selected vias:
   - The via coordinates. You can also use the Move command.
   - The via type.

   If you select another trace corner while the dialog box is open, the information updates for the selected corner.

Tips:

- If you modify a via that is a locked test point, a Warning dialog box may appear informing you of changes you are making to the test point and giving you options.

  See also: Troubleshooting
- Several of the options in this dialog box are unavailable if the via is part of a physical design reuse.

Related Topics

Via Properties Dialog Box

Modifying Trace Segment Properties

To modify trace segment properties:

1. Select the trace segment.
2. Right-click and click **Properties**. The Trace Properties dialog box appears. You can modify the following information about all selected segments:
   - The segment end point coordinates. You can also use the Move command.
   - The segment width.
   - The segment layer. If you change the layer, vias are automatically added.
3. Click **OK**.

If you select another trace segment while the dialog box is open, the information is updated for the selected segment.

Tip: Several of the options in this dialog box are unavailable if the trace is part of a physical design reuse.

Troubleshooting Constraints While Routing

When you're routing and you're constrained by the design rules, you can access the rules for a design object (class, net, group, pin pair, component, or unroute) by selecting the object in the design and showing the rules. For example, if there are specific rules defined for the pin pair associated with the unrouted connection, the Pin Pair Rules dialog box appears.

To display the Rules dialog box:

1. Select the object whose rules you want to display.
2. Right-click and click **Show Rules**.

Viewing Protected Routes with Outline Mode

If your design contains protected routes, you can set them to display with the opposite outline pattern of other routes. In other words, if normal routes are solid, protected routes are outlined, and visa-versa.

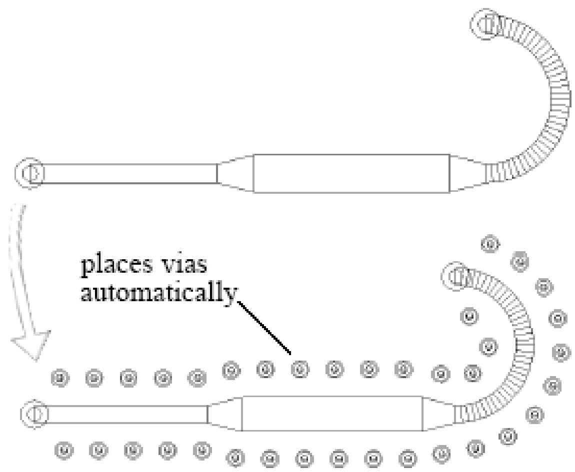
See also: Route Protection in the *Concepts Guide*

To display protected routes in the opposite outline:

1. **Tools** menu > **Options** > **Routing** tab.
2. Select the **Show Protection** check box.
3. Click **OK**.

Adding a Via Shield

A via shield is a collection of stitching vias that surround a routed net, pin-pair or copper area. You might use a via shield, for example, to surround a net in order to limit noise susceptibility. For example:



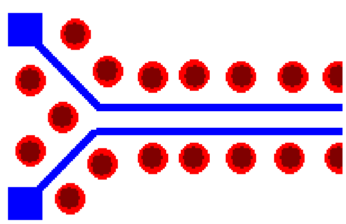
To add a via shield:

1. If you have not done so, set the options for shielding on the Via Patterns tab. For information, see Setting Via Shielding Options.

   Tip: PADS Layout stores your Via Pattern settings for future operations. You need to perform this step only if you want to change the settings you selected previously.

2. Select a net, pin-pair or copper area.

   Tip: To add a via shield around a differential pair, first set the filter mode to either Net or Pin-Pair. Then select the two nets or two pin-pairs that make up the differential pair and add a via shield around them. For example:

   The shielding operation applies a shield to each net; therefore, unwanted vias may appear between differential pair traces. If this result happens, you would need to delete those vias.

3. Right-click and click **Add Via Shield**.

Result: PADS Layout places the vias around the selected net, pin-pair, or copper.

Exception: If the Design Rule Checking setting is Prevent Errors, the Add Via Shield operation does not add any via that violates design rules.

Restriction: When you edit traces, PADS Layout does not update the pattern of vias in the via shield. After editing traces, you may need to add the via shield again. If that is the case, delete the old via shield before adding the new one.

Related Topics

Vias

Options Dialog Box, Via Patterns Page

Autorouting using PADS Router Link

Autorouting Your Design

The PADS Router Link sets up autorouting options for PADS Router and passes information from PADS Layout to PADS Router. Data is stored in the design file. Using the link you can either run PADS Router and automatically open the current design file in the foreground, so you can view the autorouter's progress, or you can run PADS Router in the background. You can also save a .pcb file for further use and route it in PADS Router later.

To set up routing options and autoroute your design in PADS Router from PADS Layout:

1. **Tools** > **PADS Router**.
2. In the Action area, Select the action you want to perform. The options are:

   Open PADS Router — Opens PADS Router and loads the design file into it. You can make changes to the design file in PADS Router (rather than PADS Layout) before routing.

   Autoroute in Background — Opens PADS Router and PADS Router Monitor, but runs PADS Router in the background. Layout commands are disabled and a wait cursor shown until autorouting is completed or the Stop button is selected in the Router Monitor.

   Autoroute in Foreground — Opens PADS Router and PADS Router Monitor, and runs PADS Router in the foreground making it the active program.

   Save PCB File — Stores the selected routing strategy in the current design. This option does not open PADS Router or PADS Router Monitor, but saves a .pcb file that you can later open and route in PADS Router.
3. Click **Setup** in the Routing Strategy area to set the routing strategy.

   See also: Setting up a Routing Strategy

4. In the Options area, click which Options dialog box tab you want to appear when you click **Setup**.

   See also: Setting Options

5. Type an output file name or click **Save As** to select one. Output files to PADS Router have the same .pcb extension as other PADS Layout design files.

   Tip: You can use only PowerPCB 3.5 and higher files in PADS Router.

6. Click **Proceed**.

If you clicked Run in Background, the PADS Router Monitor appears, but PADS Router does not appear on your desktop.

If you clicked Run in Foreground, the PADS Router Monitor appears. PADS Router also opens and automatically autoroutes the current PADS Layout design.

See also: PADS Router Monitor Dialog Box

Setting up a Routing Strategy

Use the Routing Strategy dialog box to define a strategy for autorouting a design. The strategy defines the sequential operations to perform during autorouting, including:

- Passes that the autorouter should run.
- Order in which to autoroute components, nets, net classes, differential pairs, and matched length groups.

Use the Routing Strategy dialog to specify which passes to run, routing intensity, whether or not to protect the generated traces, whether to pause after a pass completes, and the order in which to autoroute components, nets, net classes, differential pairs, and matched length groups for the selected pass.

Procedure

1. **Tools menu > PADS Router**.
2. In the Routing Strategy area, click **Setup**.
3. In the Routing Strategy dialog box, in the Pass column, select each pass type you want to run. You can run any combination of passes.
4. In the Protect column, select those passes after which to protect the generated traces. This protects traces and glues vias that are completed during the corresponding pass type.
5. In the Pause column, select the pass after which you want to pause routing.

   Tips:

- To continue after a paused pass when autorouting in the foreground, click the Resume Autrouting button on the Routing toolbar in PADS Router (or press F10).
- To continue after a paused pass when autorouting in the background, click the Stop button in the PADS Router Monitor dialog box.

6. In the Intensity column, select the appropriate intensity. Intensity determines the effort and time the router can spend on a pass.

   Restriction: You cannot set an intensity for the Center pass.

7. To set the routing order:

   a. Click the field for the Pass Type you want in the Routing Order column.

   b. To add items to the routing order, do any of the following:

      - To add all nets associated with plane layers, click **Plane Nets**.
      - To add individual objects, select the ones you want to add in the left area (Routing order definition), and then click **Selected**.
        Tip: Objects available in this area change depending on what you select in the filter list above it.
      - To remove all items from the Routing Order list, click **Clear**.

   c. Use the Routing Order area buttons to delete and move items.

8. Click **OK**.

Results

The routing strategy is saved in the \PADS projects folder. When you autoroute the design, these settings are used. If the dialog box is open, a check mark appears in the Done column upon completion of each pass.

Related Topics

Autorouting Pass Types in the *Router Concepts Guide*

Clearance and Checking After Routing

Trace Clearance is the distance allowed between a trace edge and the edge of another conducting item. Assign this property to nets in the Design Rules Clearance dialog box. If you use the Extended design rules option, you can assign nondefault clearance on the pin pair, pin pair group, net, and net class level.

Use the Verify Design command from the Tools menu to search the design for clearance violations. Verify Design marks each violation and enables you to center the view on each violation.

Setting up clearances for traces, as well as pads and vias, is described in Design Rules. Checking for errors (spacing errors, airgap errors, and high-speed problems) is discussed in Verify the Design.

Chapter 32
Filling Copper, Copper Pour Areas, and Plane Areas

In this chapter:

Flooding a Copper Pour Area

Flooding recalculates the pour area and recreates all clearances for the current obstacles within the pour outline, observing clearance rules. Use Flood if you change clearance rules, or make changes to—or within—a copper pour area that might create clearance violations.

Flooding fills the copper pour area with copper in a hatched or solid pattern, creating isolation areas around copper, traces, and pads that are inside the copper pour outline but are not part of the same net. It also creates thermal relief connections around pins that belong to the same net.

The flood operation uses the settings in the Drafting, Thermals, and Grids tabs in the Options dialog box, as well as the flood and hatch options.

Tip: If you have overlapping outlines on the same layer having the same flood priority, PADS Layout cannot determine which one is the obstacle and which one is the standoff around the obstacle. To avoid conflicts, the overlapping or common section is drawn as a non-hatch area.

There are three methods to flood one or more copper pours.

Method 1

1. Select the copper pour area(s).

2. Right-click and click **Flood**.

Method 2

1. **Drafting** toolbar > **Flood** button
2. Select a copper pour area to Flood.
3. Click and flood other copper pour areas as needed.

Method 3

1. **Tools** menu > **Pour Manager**
2. In the Pour Manager dialog box, click the Flood tab.
3. Set the Flood Mode.
4. Click **Start** to start the flood process.

Results

Is the resulting shape what you expected?

- If the gap between the pad and the copper for thermals or antipads is incorrect, see Customizing Design Rule Thermals or Customizing Design Rule Antipads of Copper Pours.
- If the fill is not correct, see The Fill of Copper, Copper Pour and Plane Shapes.
- If the shape edges are not correct, see Edge Precision of Drafting Shapes.
- If the shape needs to be modified, see Modifying Drafting Object Properties.
- If you want to start over, see Deleting a Drafting Segment or Object.

Related Topics

Assigning Copper to a Net

Hatching a Copper Pour Area

Hatching a Copper Pour Area

Hatching refills (with hatch lines) existing pour areas for the current session; it does not recalculate the pour area. Each time you open a design file, you must hatch the design; the fill is not saved. Use Flood if you change clearance rules, or make changes to—or within—a copper pour area that might create clearance violations.

Restriction

You can only hatch a copper pour area that has already been flooded. This converts the pour outline to a hatch Outline.

There are three methods to hatch one or more copper pours.

Method 1

1. Select the copper pour area(s).
2. Right-click and click **Hatch**.

Method 2

1. **Drafting** toolbar > **Hatch** button
2. Select a copper pour area to Hatch.
3. Click and hatch other copper pour areas as needed.

Method 3

1. **Tools** menu > **Pour Manager**
2. In the Pour Manager dialog box, click the Hatch tab.
3. Set the Hatch Mode.
4. Click **Start** to start the hatch process.

Results

Is the resulting shape what you expected?

- If the gap between the pad and the copper for thermals or antipads is incorrect, see Customizing Design Rule Thermals or Customizing Design Rule Antipads of Copper Pours.
- If the fill is not correct, see The Fill of Copper, Copper Pour and Plane Shapes.
- If the shape edges are not correct, see Edge Precision of Drafting Shapes.
- If the shape needs to be modified, see Modifying Drafting Object Properties.
- If you want to start over, see Deleting a Drafting Segment or Object.

Related Topics

Flooding a Copper Pour Area

Flooding a Plane Area

Flooding recalculates the plane area and recreates all clearances for the current obstacles within the plane outline, observing clearance rules. Use Flood if you change clearance rules, or make changes to—or within—a plane area that might create clearance violations.

Flooding fills the plane area with copper in a hatched or solid pattern, creating isolation areas around copper, traces, and pads that are inside the plane outline but are not part of the same net. It also creates thermal relief connections around pins that belong to the same net.

The flood operation uses the settings in the Drafting, Thermals, Split/Mixed Plane and Grids tabs in the Options dialog box, as well as the flood and hatch options.

Tip: If you have overlapping outlines on the same layer having the same flood priority, PADS Layout cannot determine which one is the obstacle and which one is the standoff around the obstacle. To avoid conflicts, the overlapping or common section is drawn as a non-hatch area.

There are three methods to flood one or more plane areas.

Method 1

1. Select the plane area(s).
2. Right-click and click **Flood**.

Method 2

1. **Drafting** toolbar > **Flood** button
2. Select a plane area to Flood.
3. Click and flood other plane areas as needed.

Method 3

1. **Tools** menu > **Pour Manager**
2. In the Pour Manager dialog box, click the Plane Connect tab.
3. Select the layer to flood.
4. Click **Start** to start the flood process.

Results

Is the resulting shape what you expected?

- If the plane did not fill, see Troubleshooting Plane Area Fills.

- If the plane filled, but opened a Thermal Relief Errors Report (therm.err), see Troubleshooting Thermal Results.
- If the gap between the pad and the copper for thermals or antipads is incorrect, see Customizing Design Rule Thermals or Customizing Design Rule Antipads of Plane Areas.
- If the fill is not correct, see The Fill of Copper, Copper Pour and Plane Shapes.
- If the shape edges are not correct, see Edge Precision of Drafting Shapes.
- If the shape needs to be modified, see Modifying Drafting Object Properties.
- If you want to start over, see Deleting a Drafting Segment or Object.

Tips:

- To see the plane data in a negative mode, use the C modeless command.
- You can use the Split/Mixed Plane tab in the Options dialog box to view only the thermals or only the plane polygon outlines. The Mixed plane display setting in the Split/Mixed Plane tab is automatically changed to Generated plane data when you flood the plane.

See also: Split Planes in the *Concepts Guide*

Related Topics

Setting Flooding Order of Overlapping Copper Pour and Plane Areas

Setting Flooding Order of Overlapping Copper Pour and Plane Areas

To determine which copper pour or plane area should be flooded first, you must set a flood priority for each object. An object with a lower priority number will be flooded before an object with a higher one. Copper pours or plane areas on different layers are processed independently.

Procedure

1. Right-click > **Select Shapes**.
2. Select the copper pour or plane area. The area you select cannot be a hatch outline or plane hatch outline.

   Tip: To quickly give the area the highest priority, bring it to the front (right-click > **Bring to front**; to give it the lowest priority, send it to the back (right-click > **Send to back**).
3. Right-click > **Properties**

4. In the Drafting Properties dialog box, click the **Options** button. If the button doesn't appear, you've selected a hatch outline.
5. In the Flood and Hatch Options dialog box, set the flood priority. 0 (zero) is the highest priority (that is, it is flooded first and can't be flooded over), and 250 the lowest.

   Tip: You can also see the new flood priority number in the status bar as P:<number>, when you select the area.
6. Click OK to close all open dialog boxes.

Related Topics

Copper Pour and Plane Area Flood Priorities in the *Concepts Guide*

Using Keepouts

Flooding Over Pads in a Copper Pour or Plane Area

You can flood over pads or vias in a copper pour or plane area. This topic describes how to flood over pads. To flood over vias, see Flooding Over Vias in a Copper Pour or Plane Area.

There are two ways to flood over pads in a plane area: using the Thermals Options dialog box and using a custom thermal in the pad stack. There is only one way to flood over a copper pour: using the Thermal Options dialog box.

Using the Thermal Options Dialog Box

Use this procedure to flood over pads in a copper pour or a plane area.

Procedure

1. **Tools** menu > **Options**.
2. Click the Thermals tab.
3. In the area you want (drilled or non-drilled) select the pad shape to flood over.
4. Click **Flood over**.
5. Repeat steps 2 and 3 for each pad shape you need to flood over.
6. Click **OK**.
7. Re-flood the area.

Results

The selected area is flooded with copper. All of the pad shapes you chose in Step 2 are flooded over.

Using a Custom Thermal in the Pad Stack

Use this procedure to flood over pads in a plane area. Use this procedure if you don't want to flood over every object in a certain shape; you can flood over selected pad stacks.

Procedure

1. **Setup menu** > **Pad Stacks**.
2. In the Pad Stacks Properties dialog box, in the Pad Stack Type area, select **Decal**.
3. Select the decal name in the **Decal name** list.
4. Select the pin(s) you want from the **Pin: Plated** list.
5. Choose the layer you want from the **Sh: Sz: Layer** list.
6. In the Parameters area, select **Thermal** from the **Pad style** list.
7. Select the appropriate pad shape button.
8. Select the Flood over check box.
9. Click **OK** and then click **Yes** to apply the changes to all components with that decal type.

Results

The selected pads stacks are flooded over in the copper area.

Related Topics

Flooding Over Vias in a Copper Pour or Plane Area

Flooding Over Vias in a Copper Pour or Plane Area

You can flood over pads or vias in a copper pour or plane area, this topic describes how to flood over vias. To flood over pads, see Flooding Over Pads in a Copper Pour or Plane Area.

There are two ways to flood over vias in a plane area: setting the drafting properties of the area and using a custom thermal in the pad stack. There is only one way to flood over vias in a copper pour: setting the drafting properties of the area.

Setting the Drafting Properties of the Area

Use this procedure to flood over vias in a copper pour or a plane area.

1. Select the pour or plan area outline.
2. Right-click and click **Properties**.
3. Click the **Options** button.
4. In the Flood and Hatch Options dialog box, select the **Flood over vias** check box.
5. Click **OK** and then click **Yes to proceed with the flood**.
   Tip: Click No to flood the area later on.
   See also: Flooding a Copper Pour Area.
6. Close the Drafting Properties dialog box.

Results

The selected area is flooded with copper. Any vias connected to the net will be flooded over, and there are no longer any thermals on vias.

Using a Custom Thermal in the Pad Stack

Use this procedure to flood over vias in a plane area. Use this procedure if you don't want to flood over every object in a certain shape; you can flood over selected pad stacks.

1. **Setup menu** > **Pad Stacks**.
2. In the Pad Stacks Properties dialog box, in the Pad Stack Type area, select **via**.
3. Select the via name in the **Decal name** list.
4. Choose the layer you want from the **Sh: Sz: Layer** list.
5. In the Parameters area, select **Thermal** from the **Pad style** list.
6. Select the appropriate pad shape button.
7. Select the Flood over check box.
8. Click **OK** and then click **Yes** to apply the changes to all vias of that type.

Results

The selected pads stacks are flooded over in the copper area.

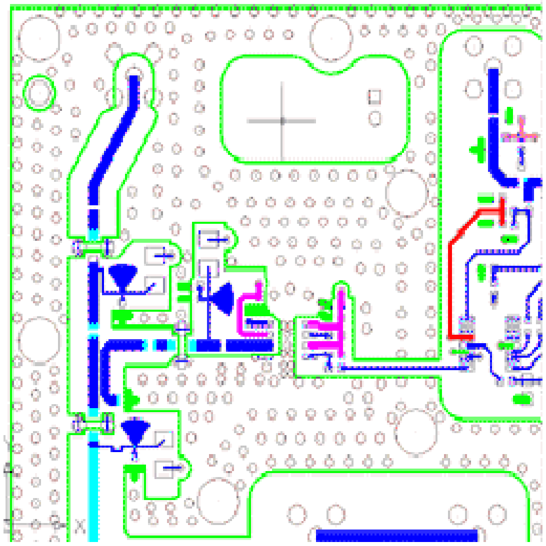
Related Topics

Flooding Over Pads in a Copper Pour or Plane Area

Filling a Shape with a Pattern of Vias

To fill a shape (such as a flooded copper pour or a hatch outline for a copper pour) with a pattern of stitching vias, use the Via Stitch operation in Fill mode. For example:

Figure 32-1. Shape Stitched with Vias



To fill a shape with vias:

1. If you have not done so, set the options for stitching shapes in the Via Patterns Options. For information, see Setting Options for Via Patterns.

   Set the Pattern setting to **Fill** on the Via Patterns tab or set it after you select the object. (See Step 3.)

   Tip: PADS Layout stores your Via Pattern settings for future operations. You need to perform this step only if you want to change the settings

2. Select the shape you want to stitch with vias. The shape can be:

   - Drawn copper
   - A flooded copper pour or a plane area
   - A hatch outline for a copper pour or a plane area

   Requirement: The copper or hatch outline must be associated with a net.

3. Right-click and click **Via Stitch**.

   Tip: To override the Pattern (Fill or Perimeter) setting in the Via Patterns Options, select **Via Stitch Mode**. Then select the pattern.

Result: PADS Layout fills the selected copper or hatch outline with the pattern of vias.

Exception: If the Design Rule Checking (DRC) setting is Prevent Errors, the Via Stitch operation does not add any via that violates design rules.

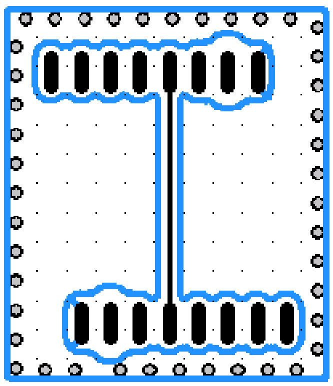
Related Topics

Vias

Placing Vias Inside the Perimeter of a Shape

Placing Vias Inside the Perimeter of a Shape

You place vias inside the perimeter of a copper area or a hatch outline by using the Via Stitch command in Perimeter mode. For example:



To place vias inside the perimeter of a shape:

1. If you have not done so, set the options for stitching shapes in the Via Patterns Options. For information, see Setting Options for Via Patterns.

   Set the Pattern setting to **Perimeter** on the Via Patterns tab or set it after you select the object. (See Step 3.)

   Tip: PADS Layout stores your Via Pattern settings for future operations. You need to perform this step only if you want to change the settings

2. Select the shape you want to stitch with vias. The shape can be:

   - Drawn copper
   - A flooded copper pour or a plane area
   - A hatch outline for a copper pour or a plane area

3. Right-click and click **Via Stitch**.

   Tip: To override the Pattern (Fill or Perimeter) setting in the Via Patterns Options, select **Via Stitch Mode**. Then select the pattern.

Exception: If the Design Rule Checking (DRC) setting is Prevent Errors, the Via Stitch operation does not add any via that violates design rules.

Related Topics

Filling a Shape with a Pattern of Vias

Surrounding a Void with Vias

When you use the Via Stitch operation to place vias inside the perimeter of a shape, by default the operation does not place vias around a void within that shape.

Requirement: Exit PADS Layout before you edit the powerpcb.ini file.

To change the default behavior for the Via Stitch operation:

1. In a text editor, open the powerpcb.ini file. The file's location is:

   C;\MentorGraphics\<PADSrelease>\SDD_Home\Programs

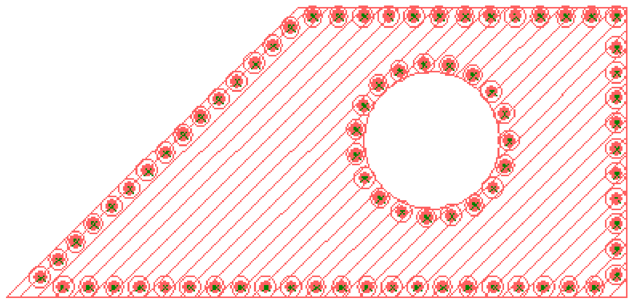
2. Change the VS_StitchVoids value to 1. For example:

```
[ViaShield]
VS_StitchVoids=1
```

3. Restart PADS Layout.

Result: The Via Stitch operation not only places vias inside the perimeter of a shape but also surrounds voids within the shape. For example:

Figure 32-2. Vias Surrounding a Void



Chapter 33 Reference Designators

In this chapter:

- Generating a Second Set of Reference Designators for Assembly Drawings
- To Move a Reference Designator
- To Move Reference Designators for Silkscreens
- To Hide Reference Designators

Generating a Second Set of Reference Designators for Assembly Drawings

You can generate a second set of reference designators, from the components on your board, to place on the Assembly Drawing layers. For assembly drawings, you typically place the reference designators in the center of the components. But for the silkscreen, you place the reference designators outside the component. Instead of moving the reference designators for the silkscreen gerber file, and the assembly drawings, you create a second set for the assembly drawings. While you can add two sets of reference designators when you create the decal, the following procedure assumes you only have one set of reference designators on the silkscreen layers, placed optimally for the silkscreen.

Procedure

1. Select all components on the top or bottom layer. You might need to hide the opposite mounted layer or set your selection filter to select only objects on the one layer.
2. Right-click and click **Add New Label**.
3. In the Add New Part Label dialog box, ensure the Attribute list is set to **Ref.Des**.
4. In the Layer list, select **Assembly Drawing Top or Assembly Drawing Bottom**.
5. Use the Position and sizes area to move all reference designators to the origin of components. This is a quick way to center reference designators of surface mount components and any other components whose origin is at the center and not pin 1.
    a. In the Position and sizes area, select the **Relative to Component** check box.
    b. In the X and Y boxes, type **0** (zero).
    c. In the Justification area, for both Horizontal and Vertical lists, click **Center**.

6. Click **OK**.
7. You will need to manually center any components with origins not a the center of the component. See To Move a Reference Designator.

Related Topics

To Hide Reference Designators

To Move Reference Designators for Silkscreens

Selecting Items

To Move a Reference Designator

Using Object Mode

To move a reference designator:

1. Select a reference designator > **Design Toolbar** button > **Move Reference Designator** button.
2. Click to indicate a new location for the reference designator.

Using Verb Mode

To move a reference designator:

1. **Design Toolbar** button > **Move Reference Designator** button.
2. Select the reference designator that you want to move. The reference designator attaches to the pointer.
3. Click to indicate a new location for the reference designator.

To Move Reference Designators for Silkscreens

To move reference designators to silkscreen layers:

1. Click **Filter** from the **Edit** menu to open the Selection Filter dialog box and turn on only the **Labels** check box. Click to clear all other check boxes. Close the Selection Filter.
2. Click **Display Colors** from the **Setup** menu to open the Display Colors Setup dialog box and click to clear all layers except Top and Silkscreen Top. Set a non-background color to the Reference Designator column for the Top and Silkscreen Top layers. Close the Display Colors Setup dialog box.
3. Using the **Layer** list on the standard toolbar, click **Top** as the active layer.

4. With nothing selected, right-click and click **Select All**. The reference designator labels on the Top layer are selected. If the components do not have reference designator labels, create them.
5. Right-click and click **Properties**.
6. Change the layer for the labels to Silkscreen Top, and click **OK**. The reference designator labels move to the Silkscreen Top layer.
7. Repeat this procedure for labels on the bottom layer, moving them to the Silkscreen Bottom layer.

Related Topics

Generating a Second Set of Reference Designators for Assembly Drawings

To Hide Reference Designators

Selecting Items

To Hide Reference Designators

Result: The Part Label Properties dialog box appears.

To hide the display of reference designators:

1. Select a reference designator > right-click > **Properties**.
2. Click **None** from the **Show** list.
3. Click **OK**. The reference designators are hidden.

Tip: You can hide the display of reference designators by layer using the Display Colors Setup dialog box.

Related Topics

Generating a Second Set of Reference Designators for Assembly Drawings

To Move Reference Designators for Silkscreens

Selecting Items

Chapter 34 Using the ECO Toolbar

Use the ECO Toolbar to:

- Make engineering (netlist) changes to a schematic-driven design.

  When laying out a schematic-driven design, you need to record any netlist changes you make in order to back-annotate them to the schematic. To do this, you work in ECO mode to record your changes, and use the ECO toolbar tools to make the changes, which are saved in a text file called the ECO file (.eco). Then you use the ECO file to back-annotate the changes to the schematic.

  Tip: You can also create an ECO file by using the Compare/ECO dialog box (**Tools menu** > **Compare/ECO**).

- Create a layout-driven design.

  In a layout-driven design (with no schematic), you use the tools in the ECO toolbar to create the netlist (add parts and make connections). Since there is no schematic to back-annotate to, you don't need to save changes in an ECO file.

In this chapter:

Recording ECO Changes

If you are working on a schematic-driven design, you will need to back-annotate design changes to the schematic. Use the following procedure to record your netlist changes and save them to an ECO file.

Procedure

1. **Standard Toolbar** > **ECO Toolbar** button. The first time you use the ECO toolbar in a session, the ECO Options dialog box appears. If it doesn't appear, click the **ECO Options** button on the ECO toolbar.

2. Select the **Write ECO file** check box.

3. Make sure that the pathname of the .eco file in the **Filename** box is correct.

   Tip: If the file doesn't exist, it is created and recorded changes are written to it. Existing .eco files are left untouched.

4. If the .eco file already exists, specify what to do with it:
   - To append ECO changes you make in this session to the existing file, select **Append to file**.
   - To overwrite the existing file with the changes you make in this session, clear **Append to file**.
5. Specify when ECO changes will be saved:
   - To be able to save ECO changes incrementally during the session, so you can view them in the ECO file, select **Write ECO file after closing ECO toolbox**.
   - To save ECO changes only when you save the .pcb file, open a new file, or exit PADS Layout, clear **Write ECO file after closing ECO toolbox**.
6. Set the other options as appropriate. See ECO Options Dialog Box.
7. Click **OK.**

Result

Any netlist changes you make will be recorded. If you have checked the **Write ECO file after closing ECO toolbox** check box, the changes will be written to the ECO file when you close the ECO toolbox; otherwise they will be saved when you save the design, open a new file, or exit the software.

If you clear the **Write ECO file** or the **Write ECO file after closing ECO toolbox** option during a session, all recorded change data, both in memory and in the ECO file, are deleted.

ECO Mode Operations (Layout Driven Design Tools)

The topics in this section describe operations you can perform only in ECO mode.

Restriction

A protected route or physical design reuse element can prevent you from making ECO changes. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Component Operations

Net/Connection Operations

Adding a Connection in ECO Mode

You can create a new pin pair to change the current netlist.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Procedure

1. **ECO Toolbar** button > **Add Connection** button.

Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Select the first pin in the connection.

   Tips:

   - You can rename the net of the current pin. Right-click and click Rename Current Net. In the Rename Net prompt type the new name, and click **OK**. The maximum netname length is 47 characters.
     See also: Illegal Characters in Netnames and Part Names
   - You can choose to use the pin function as the net name. Right-click and click Derive Net Name from Pin Function.

3. Select the second pin in the connection. If you are connecting to a pin of a different net (merging two nets), the Define Name of Merged Net dialog box appears. Choose a new netname. (Netnames are automatically generated for each newly created connection.)

   See also: Predefined Netnames in *The PADS Layout Concepts Guide*

4. Click another pin to add additional connections, or right-click and click **Cancel** to stop adding connections.

Related Topics

Illegal Characters in Netnames and Part Names

Adding a Route in ECO Mode

You can create routes that add to or change the current netlist, or that end a route on a segment with a different netname.

Caution
When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Procedure

1. **ECO Toolbar** button > **Add Route** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Select the pin at which to start the trace.

   Restriction: With On-line DRC in Prevent errors mode**, w**hen routing a trace between pins that are not part of the same net or that have no netlist - all items are considered obstacles. To connect to the pin, right-click and click **Select Target** and select the pin at which you want to end the trace.

Tip: During ECO-mode routing, with Online DRC disabled, the pointer changes to a bull's-eye when it is near any eligible completion point, regardless of the net.

3. Add corners and vias as described in "Routing Manually." You can also use the shortcut menu to control routing operations.

   Tips:

   - You can rename the net of the current pin. Right-click and click Rename Current Net. In the Rename Net prompt type the new name, and click **OK**. The maximum netname length is 47 characters.
     See also: Illegal Characters in Netnames and Part Names
   - You can choose to use the pin function as the net name. Right-click and click Derive Net Name from Pin Function.

4. If you are connecting to a pin of a different net (merging two nets), the Define Name of Merged Net dialog box appears. Choose a new netname. (Netnames are automatically generated for each new connection.)

   See also: Predefined Netnames in the *Concepts Guide*

5. Right-click and click **Cancel** to exit Add Route mode.

Related Topics

Illegal Characters in Netnames and Part Names

Adding a Component in ECO Mode

You can add a component and change the netlist of a design.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Procedure

1. **ECO Toolbar** button > **Add Component** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. In the Get Part Type from Library dialog box, to filter the part types, in the filter area, type a wildcard or expression in the Items box and click **Apply**.

Restriction: The Filter Items box cannot be empty. Add a * (asterisk) to see all items of the library in the Part Types list.

3. Select the part name in the **Part Types** list and click **Add**. The component attaches to the pointer.

    - If the selected part type already exists in the design with the specified name, the message “Part Type <name> already exists in design. Loading from Library is skipped, design's Part Type will be used” appears. Click **OK** to continue. Go to step 4.

    - If the specified part type does not exist in the design, but the decal name does exist, the message “Decal(s) <name> already exists in design. Loading of decal(s) from Library is skipped, design's data will be used” appears. This message appears only if the number of terminals in the library part and the design part are the same. Click **OK** to continue. Go to step 4.

    - If the specified part type does not exist, and the number of terminals in the new decal does not match the number of terminals in the existing decal, the message “Decal <name> already exists in design. Library decal has different number of terminals than decal in design. Aborting Get Part Type from Library” appears. The Add Part process cancels in this case.

    - If the number of terminals does not match in the alternate decals, the message “Library Part Type <library name:part type name> has decals with different number of terminals. Aborting Get Part Type from Library” appears. The Add Part process cancels in this case.

    Tip: You cannot add a decal to a design if the design is in default layer mode and the decal to be added is in increased layer mode. Use the Layers Setup dialog box to change the design to increased layer mode.

4. Click **Close** to close the dialog box.

5. **Optional: Assign a reference designator.** Unless you assign a reference designator, the new part is automatically annotated with the next available reference designator. You can change the reference designator before placing the new component.

    - Right-click and click **Rename Current Part**. The Rename Part prompt appears asking for a new reference designator. Type the new reference designator, and click **OK**.

6. Move the part to the new location and click to indicate the new position.

Tip: A part must be ECO-registered to be included in the ECO file. You set the ECO registration for parts in the Part Information dialog box.

Related Topics

Updating a Design from the Library

Changing the Reference Designators of Multiple Components in ECO Mode (Autorenumbering)

After you complete the final placement of a design, you can renumber the parts to order the reference designators in a recognizable pattern, such as top to bottom, left to right. This makes it easier to find a component on a fabricated board.

Autorenumbering is an ECO operation, so you must start ECO mode before you can autorenumber parts. The resulting ECO file contains part name changes for backward annotation to the associated design file.

Caution

If you are renumbering a schematic-driven design that will be back-annotated to the schematic, you *must* record the autorenumber operation by checking the **Write ECO file** check box in the ECO Options dialog box before you renumber. Recording the exact changes in an .eco file gives the best back annotation results. For more information, see Recording Versus Generating an ECO File in the *Concepts Guide*.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Jumpers are excluded from autorenumbering.

Restriction

A protected route or physical design reuse element can prevent you from changing a reference designator. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. **ECO Toolbar** > **Auto Renumber** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. In the AutoRenumber Dialog Box, make sure the reference designation prefixes you want to use are selected in **Prefix List**. You must select at least one prefix.

3. In the **Cell Size** area, type the matrix size for the renumbering sweep.

4. Make your renumbering choices in the **Precedence** area:

- For designs with parts on both top and bottom, select the **Continuous numbering side to side check box t**o number all parts sequentially. In **Start Renumbering From** click either **Top** or **Bottom**.
- For the Top and/or Bottom select the **Renumber** check box and indicate the **Start at** value and the **Increment** value if applicable.
- Click a renumbering pattern button for one or both sides.

5. Click **OK**.

Result

- Are the results not what you expected? You can undo the renumbering and start over.
- Did the board get renumbered properly? Are there unexpected numbering results? See AutoRenumber Sweeps.

Related Topic

Recording Versus Generating an ECO File in the *Concepts Guide*.

Recording ECO Changes

Backward Annotating from PADS Layout to PADS Logic

Back Annotating from PADS Layout to DxDesigner

Changing a Component in ECO Mode

Use Change Component to change the part type of one or more components, or all part types of the same name, to the part type of another component in the design or to a part type in the library. The updated part type may contain a different decal assignment which can affect routing in the design. You can also update the part type with a newer version from the library.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restrictions

- A protected route or physical design reuse element can prevent you from changing a component in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

- You cannot add a decal to a design if the design is in default layer mode and the decal to add is in increased layer mode. Use the Layers Setup dialog box to change the design to increased layer mode.

Procedure

1. **ECO Toolbar** button > **Change Component** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Turn Online DRC (Design Rules Checking) off by typing **DRO** and pressing Enter.

3. Select the component or components whose part type you want to change. To select a single component, click the component in the design area. To select multiple components, right-click and click **Find** to use the Find dialog box. For more information, see Find Objects dialog box.

4. Change the component using one of the following:

   a. To change the component to a different part type from the design, click the component whose part type you want to use as the replacement.

   b. To change the component to a part type from the library, right-click and click **Library Browse**. The Get Part Type from Library dialog box appears. To filter the part types, in the filter area, type a wildcard or expression in the Items box and click **Apply**. Locate and select the item, and then click **Replace**.

      - If the selected part type already exists in the design with the specified name, the message “Part Type <name> already exists in design. Loading from Library is skipped, design's Part Type will be used” appears. Click **OK** to continue.

5. In the Change Component dialog box, you are prompted to confirm the change. Select the **Keep Attributes** check box if you want to retain the attributes of the part in the design.

6. If the modified Part Type changes the PCB decal or contains different pin numbers, you must map the old decal pin numbers to the new decal pin numbers in the Assign Pin Numbers Dialog Box.

Result

- Change Component retains test points if the pin position is the same and the new terminal still has pads on the testing side; otherwise the test point is removed.

Related Topics

Updating a Part Type from the Library

Updating a Part Type from the Library in ECO Mode

You can update part type data from a new version in the library. The updated part type may contain a different decal assignment which can affect routing in the design. You can also change a component to another part type in the design or in the library.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restrictions

- You cannot add a decal to a design if the design is in default layer mode and the decal to add is in increased layer mode. Use the Layers Setup dialog box to change the design to increased layer mode.
- A protected route or physical design reuse element can prevent you from updating a part type in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. **ECO Toolbar** button > **Change Component** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Turn On-line DRC (Design Rules Checking) off by typing **DRO** and pressing Enter.
3. Select the component or components whose part type you want to update.
4. Right-click and click **Library Browse**.
5. In the Get Part Type from Library dialog box, select the **Update Part Type from Library check box** to update the selected part type with a new library definition of the same name. The search locates and automatically highlights the library definition for the selected part type.
6. Click **Replace.**
7. If there are additional components with the selected part type that you did not select in the design, you are prompted with the message, “Part type <name> will be replaced with Library item for ALL parts of this type in the design”. Click **OK**.
   - **Restriction:** You must update all components which use the same part type.
8. You are prompted to confirm the change. Select the **Keep Attributes** check box if you want to retain attributes.

9. If the modified Part Type changes the PCB decal or contains different pin numbers, you must map the old decal pin numbers to the new decal pin numbers in the Assign Pin Numbers Dialog Box.

Result

- Change Component retains test points if the pin position is the same and the new terminal still has pads on the testing side; otherwise the test point is removed.

Related Topics

Assign Pin Numbers Dialog Box.

Changing a Component in ECO Mode

Copying a Part in ECO Mode

Creating a copy of an existing part changes the netlist, is considered an engineering change and requires ECO mode.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Procedure

1. **In the design, select the part to copy.**
2. **ECO Toolbar** button > **Add Component** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

   Alternative: On the Edit menu, click **Copy and then Paste**.
3. The component attaches to the pointer.
4. **Optional:** Assign a reference designator. Unless you assign a new reference designator, the new part is automatically annotated with the next available reference designator. You can change the reference designator before placing the new component.
   - Right-click and click **Rename Current Part**. The Rename Part prompt appears asking for a new reference designator. Type the new reference designator, and click **OK**.
5. Move the part to the new location, and click to indicate the new position.

Deleting a Component in ECO Mode

Deleting a component from the board changes the netlist, is considered an engineering change and requires ECO mode.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restriction

A protected route or physical design reuse element can prevent you from deleting a component in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. **ECO Toolbar** button > **Delete Component** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Set the selection filter to **Select Components**.
3. Select one or multiple components to delete.

   Tip: You can select multiple components by dragging a box around them.

4. In the Delete Part dialog box, you can optionally select the Delete Routes check box to delete routes that are connected to the component.
5. Click **OK** to confirm the deletion.
6. If the deletion of the component(s) creates single-pin nets, a second Delete Part dialog box appears and displays which nets will also be deleted. This is not an optionally step. **Click Close**.

Result

Test points are deleted with the part.

Deleting a Connection in ECO Mode

Use Delete Connection to delete a pin pair, disconnect a pin from a net, or split a net into two different nets. This also deletes test point vias belonging to the connection. Deleting a connection changes the netlist, is considered an engineering change and requires ECO mode.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restriction

A protected route or physical design reuse element can prevent you from deleting a connection in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. **ECO Toolbar** button > **Delete Connection** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Do one of the following:

   - **Delete Pin Pair**—Select the connection or trace between the pin pair to delete it. If the net contains only one pin pair, selecting either pin deletes the pin pair.
   - **Disconnect a Pin**—Select the pin to disconnect. If a net contains more than one pin pair, the Disconnect Pin dialog box appears with an option to delete the route segment(s) going to the pin.
   - **Split a Net**—See Splitting a Net in ECO Mode.

Splitting a Net in ECO Mode

You can split a net by deleting a pin pair in the middle of multi-pin pair net. Splitting a net changes the netlist, is considered an engineering change and requires ECO mode.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restriction

A protected route or physical design reuse element can prevent you from splitting a net in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

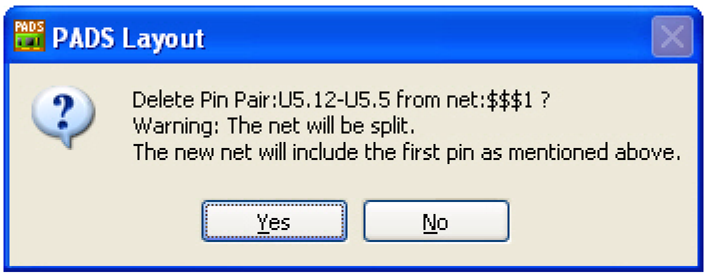
1. **ECO Toolbar** button > **Delete Connection** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Select the connection between pin pairs where the net should be split.

3. You are prompted with the message, “Delete Pin Pair:<refdes.pin-refdes.pin> from net:<name>? Warning: The net will be split. The new net will include the first pin as mentioned above.” Click **Yes**.

   The following is an example of the prompt that appears.



4. In the Define Name of New Net dialog box, name the new net or let PADS Layout automatically generate the new netname.

   Restrictions:

   - The maximum netname length is 47 characters.
   - You can use any alphanumeric characters except brackets { }, asterisks *, commas (,), spaces, question marks, or commas.

Result

- **Tip:** If a Color by Net setting exists for a net that is split, the new net uses the same color setting. **See also:** To Assign Colors to Nets
- **Warning:** If you split a net associated to bridge copper, the association is removed from the copper.
- Test points belonging to the connection are deleted.

Deleting a Net in ECO Mode

You can delete all pin pairs and remove all pin connections from a net. Deleting a net changes the netlist, is considered an engineering change and requires ECO mode.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restriction

A protected route or physical design reuse element can prevent you from deleting a net in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. **ECO Toolbar** button > **Delete Net** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Select a pin, unrouted pin pair, trace, or via in the net to delete.

Result

Delete Net also deletes test point vias that belong to the net.

Changing the Reference Designator of a Component in ECO Mode

You can change a single component's reference designator using the Rename Component ECO tool. Changing a reference designator changes the netlist, is considered an engineering change and requires ECO mode.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restriction

A protected route or physical design reuse element can prevent you from changing a reference designator in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. **ECO Toolbar** button > **Rename Component** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Select a single component.
3. In the Rename Part prompt, in the New Name box, type the new reference designator.
4. Click **OK**.

Related Topics

Recording Versus Generating an ECO File in the *Concepts Guide*.

Recording ECO Changes

Backward Annotating from PADS Layout to PADS Logic

Back Annotating from PADS Layout to DxDesigner

Changing the Reference Designator Prefix of Multiple Components in ECO Mode

You can change the reference designator prefix of multiple design components using the Rename Component ECO tool. This changes the netlist, is considered an engineering change and requires ECO mode.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restriction

A protected route or physical design reuse element can prevent you from changing a reference designator prefix in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. Select multiple components whose reference designator prefix you want to change.

   Restriction: All selected components must currently have the same prefix.

2. **ECO Toolbar** button > **Rename Component** button.

Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

3. Choose one of the following:
    - Add to the existing prefix—Type the new prefix and then the current prefix.
    - Change the existing prefix—Type a new prefix.
4. Click **OK**.

Related Topics

Recording Versus Generating an ECO File in the *Concepts Guide*.

Recording ECO Changes

Backward Annotating from PADS Layout to PADS Logic

Back Annotating from PADS Layout to DxDesigner

Renaming a Net in ECO Mode

You can change the name of a net using the Rename Net ECO tool. This changes the netlist, is considered an engineering change and requires ECO mode.

Restriction

A protected route or physical design reuse element can prevent you from renaming a net in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. **ECO Toolbar** button > **Rename Net** button.

    Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Select a pin, unrouted pin pair, trace, or via in the net to rename.
3. In the Rename Net dialog box, type the new name, and click **OK**. The maximum netname length is 47 characters.
    See also: Illegal Characters in Netnames and Part Names

Swapping ECL Terminators Automatically in ECO Mode

You can automatically swap ECL terminator assignments, or swap netnames between ECL pins.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restriction

A protected route or physical design reuse element can prevent you from swapping ECL terminators in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. **ECO Toolbar** button > **Auto Terminator Assign** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

Tip: This command ignores objects that are part of a physical design reuse.

Swapping Gates

You can swap gates manually or automatically.

- Swapping a Gate Manually in ECO Mode
- Swapping All Gates Automatically in ECO Mode

Swapping a Gate Manually in ECO Mode

You can swap a gate with any equivalent gate to optimize routing and/or minimize trace lengths. Gates are equivalent when:

- They share the same package.
- They have matching, nonzero Swap IDs.
- They have an equal pin count.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restrictions

- You cannot swap a gate that has any pins from a pin pair with rules.
- A protected route or physical design reuse element can prevent you from swapping gates in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. **ECO Toolbar** button > **Swap Gate** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Select a pin.

   All candidates for swapping are highlighted. Candidates with a nonzero Swap ID that matches the selected pin's Swap ID are highlighted in the highlight color. Candidates with different or undefined Swap IDs are highlighted in a complementary color.

3. Select one of the highlighted pins with which to swap.

Result

If Stretch Traces During Component Move check box is selected (on the Design page of the Options dialog box), traces attached to pins of the swapped gates are rerouted. If Stretch Traces During Component Move is cleared, trace segments are unrouted before the swap is performed.

Related Topics

Part Information Dialog Box, Gates Tab

Undoing the Last Swap

Swapping All Gates Automatically in ECO Mode

You can swap all gates with any equivalent gates based on shortest total unrouted pin pair lengths. Gates are equivalent when:

- They share the same package.
- They have matching, nonzero Swap IDs.
- They have an equal pin count.

Caution

When you modify or delete design objects, associated nets are regenerated. This can cause existing associated nets that include these objects to be truncated, split, or deleted altogether.

Restrictions

- You cannot swap a gate that has any pins from a pin pair with rules.
- Objects that are part of a physical design reuse are ignored.
- A protected route or physical design reuse element can prevent you from swapping gates in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. Click the **ECO Toolbar** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Click the **Auto Swap Gate** button on the **ECO** toolbar.

Result

If Stretch Traces During Component Move check box is selected (on the Design page of the Options dialog box), traces attached to pins of the swapped gates are rerouted. If Stretch Traces During Component Move is cleared, trace segments are unrouted before the swap is performed.

Related Topics

Part Information Dialog Box, Gates Tab

Undoing the Last Swap

Swapping Pins

You can swap pins manually or automatically.

- Swapping a Pin Manually in ECO Mode
- Swapping All Pins Automatically in ECO Mode

Swapping a Pin Manually in ECO Mode

You can swap a pin with any equivalent pin to optimize routing and/or minimize trace lengths. Pins are equivalent when they have matching, nonzero Swap IDs.

Restrictions

- You cannot swap a pin that has any pins from a pin pair with rules.
- **Connectors that are glued down are not included in the swap.**
- A protected route or physical design reuse element can prevent you from swapping pins in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. **ECO Toolbar** button > **Swap Pin** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Select a pin.

   All candidates for swapping are highlighted. Candidates with a nonzero Swap ID that matches the selected pin's Swap ID are highlighted in the highlight color. Candidates with different or undefined Swap IDs are highlighted in a complementary color.

3. Select one of the highlighted pins with which to swap.
   Tip: If you attempt to swap pins with undefined swap IDs or with different swap IDs, the Confirm Pin Swap dialog box appears and you must confirm that the swap is legitimate. If these pins should be swappable, the correct process would be to give them identical swap IDs in the Pins tab of the Part Information dialog box.

4. Click **OK**.

Result

If Stretch Traces During Component Move check box is selected (on the Design page of the Options dialog box), traces attached to pins of the swapped gates are rerouted. If Stretch Traces During Component Move is cleared, trace segments are unrouted before the swap is performed.

Related Topics

Part Information Dialog Box, Pins Tab

Undoing the Last Swap

Swapping All Pins Automatically in ECO Mode

You can swap all pins with equivalent pins automatically based on shortest total unrouted pin pair lengths. Pins are equivalent when they have matching, nonzero Swap IDs.

Restrictions

- You cannot swap a pin that has any pins from a pin pair with rules.
- **Connectors that are glued down are not included in the swap.**
- Objects that are part of a physical design reuse are ignored.
- A protected route or physical design reuse element can prevent you from swapping pins in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Procedure

1. Click the **ECO Toolbar** button.

   Tip: The first time you use the ECO Toolbar, the ECO Options dialog box appears. Set the options you want and click OK to use the ECO tools.

2. Click the **Auto Swap Pins** button on the **ECO** toolbar.

Result

If Stretch Traces During Component Move check box is selected (on the Design page of the Options dialog box), traces attached to pins of the swapped gates are rerouted. If Stretch Traces During Component Move is cleared, trace segments are unrouted before the swap is performed.

Related Topics

Part Information Dialog Box, Pins Tab

Undoing the Last Swap

Undoing the Last Swap

You can undo your last Swap Pin or Auto Swap Pin command immediately after performing the swap.

Procedures

Undo a Swap Pin command

- Right-click and click **Undo Last Swap** while still in Swap mode, or

- Exit Swap mode and click **Undo** on the toolbar.

Undo an Auto Swap Pin command

- Exit Swap mode and click **Undo** on the toolbar.

Copying Bridge Copper in ECO Mode

When you copy bridge copper in ECO mode and at least one assigned net trace with it, the pasted copy retains its bridge status and association to any attached trace(s). According to the circumstances of the pasted traces, their net names may be renamed or merged with existing net(s).

Illegal Characters in Netnames and Part Names

You can use any alphanumeric character for netnames and part names, with the exceptions below.

The following characters are illegal in netnames:

Spaces
Commas (,)
Braces ({ })
Asterisks (*)
Question marks (?)

The following characters are illegal in part names:

Spaces
Commas (,)
Braces ({ })
Asterisks (*)
Question marks (?)
Periods (.)
Ampersands (&)

Chapter 35
Comparing Designs

In this chapter:

- Comparing Two Versions of a Design
- Comparing Designs Using ECOGEN from the Command Prompt

Comparing Two Versions of a Design

You can compare two versions of a design to view their differences or to create an ECO file.

- Creating a Report of Differences Between Two Versions of a Design
- Creating an ECO File by Comparing Two Versions of a Design

Creating a Report of Differences Between Two Versions of a Design

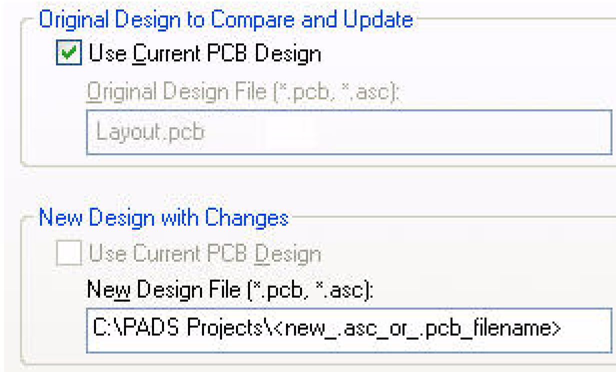
You can compare two versions of the design stored in any of the following forms:

- PCB Layout in memory
- PADS-format ASCII netlist file (.asc) representing the schematic or the PCB layout
- PCB layout file (.pcb)

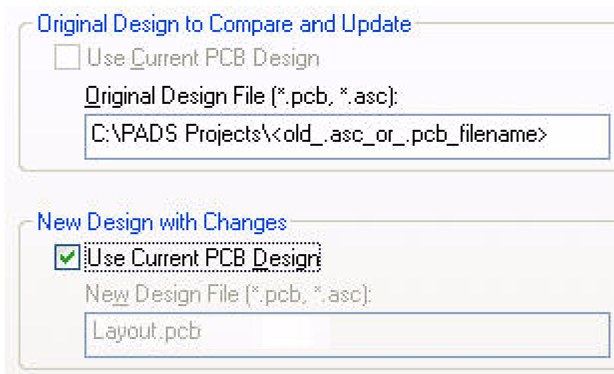
Procedure

1. **Tools** menu > **Compare/ECO**
2. In the Compare/ECO dialog box, on the Documents tab, select the designs to compare and the files you want to create, as follows:

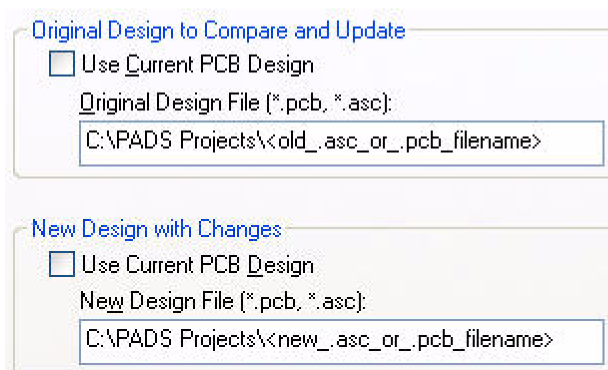
- If the older design is in memory, and you want to compare it with a newer .asc or .pcb file, use these settings:



- If the newer design is in memory, and you want to compare it with an older .asc or .pcb file,, use these settings:



- If neither design is in memory, and you want to compare an older .asc or .pcb file with a newer .asc or .pcb file, , use these settings:



3. Select the **Generate Differences Report** check box.
4. Set the appropriate options on the Comparisonand Update tabs.
5. Click **Run** to compare the designs. (The **Run** button becomes available when you select an option in the Output Options area on the Documents tab.)

6. Click the **Show Report** button in the Process Status dialog box to view the report.

7. Click the **Close** button.

Creating an ECO File by Comparing Two Versions of a Design

You can compare two versions of the design stored in any of the following forms:

- PCB Layout in memory
- PADS-format ASCII netlist file (.asc) representing the schematic or the PCB layout
- PCB layout file (.pcb)

This procedure generates a new .eco file by comparing two designs. If you are trying to forward or backward annotate design changes between PADS Layout and PADS Logic or DxDesigner, you should record changes into an .eco file as you make them. This gives the best back-annotation results. For more information, see Recording ECO Changes, Backward Annotating from PADS Layout to PADS Logic or Back Annotating from PADS Layout to DxDesigner For more details on the differences of these methods, see Recording Versus Generating an ECO File in the *Concepts Guide*.

Tip: If PADS Layout and the schematic tool are on the same computer, you can use DxDesigner Link in PADS Layout or PADS Layout Link in PADS Logic to both create the ECO file and forward or back-annotate the schematic.

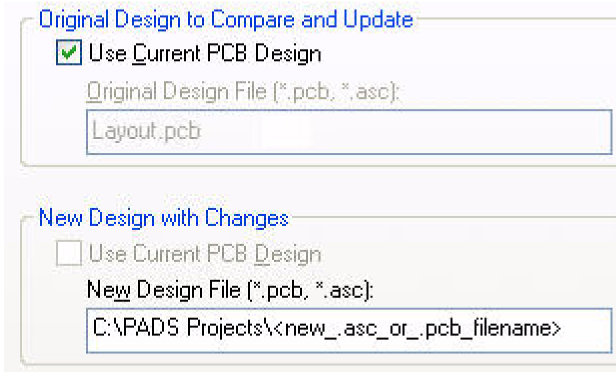
- Using DxDesigner: Working with DxDesigner
- Using PADS Logic: Working with PADS Logic

Procedure

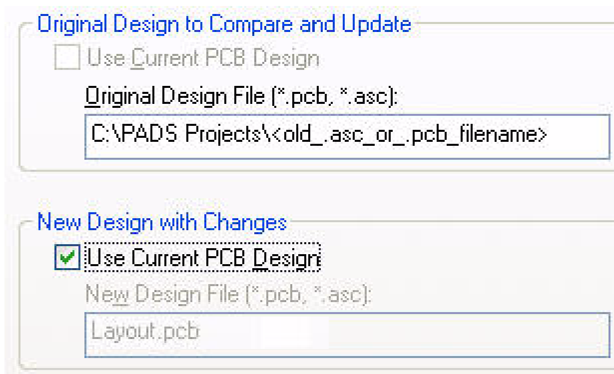
1. **Tools menu > Compare/ECO**

2. In the Compare/ECO dialog box, on the Documents tab, select the designs to compare and the files you want to create, as follows:

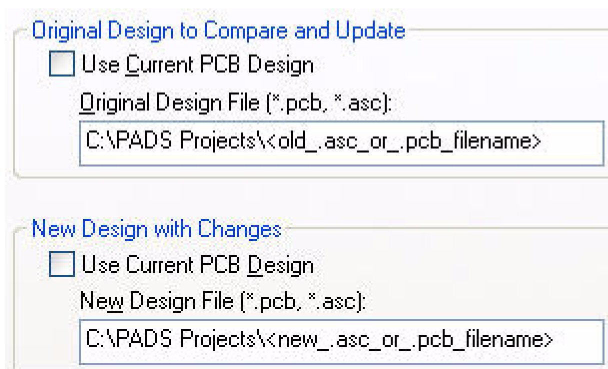
- If the older design is in memory, and you want to compare it with a newer .asc or .pcb file, use these settings:



- If the newer design is in memory, and you want to compare it with an older .asc or .pcb file,, use these settings:



- If neither design is in memory, and you want to compare an older .asc or .pcb file with a newer .asc or .pcb file, , use these settings:



3. **Optional:** Select the **Generate Differences Report** check box.
4. Select the **Generate ECO File** check box.
5. Set the appropriate options on the Comparisonand Update tabs.

6. Click **Run** to compare the designs and create the .eco file. (The **Run** button becomes available when you select an option in the Output Options area on the Documents tab.)
7. Click the **Show Report** button in the Process Status dialog box to view the report, the .eco file, and the log file.
8. Click the **Close** button.
9. To confirm that the updated design matches the new design, generate a differences report. If you've updated the .pcb design, and differences still exist, use the ECO toolbar commands to correct the design.

Result

- If the original design is in memory and you selected **Update Original Design** on the Update tab of the Compare/ECO dialog box, the updates are automatically imported into the design when comparison is completed.
- Otherwise, use **File > Import** to import the .eco file and update the original design.

Tips:

- Any messages or errors that occur during comparison are written to Layout.err, which is stored in the \PADS Projects folder.
- If commands in the ECO file added any parts at the origin in the updated original design, be sure to place these parts and route any newly unrouted pin pairs.
- After verification is complete, check the design integrity. On the Tools menu, click **Verify Design**.

Related Topics

Comparing and Updating Designs in the *Concepts Guide*

Comparing Designs Using ECOGEN from the Command Prompt

After you make changes to a design, you can run the ECOGEN executable file from the command prompt to:

- Compare the new version of the design to the original
- Create the files needed to update the original design to match the new design

Alternatively, you can use the Compare/ECO Tools dialog box in PADS Layout to compare designs.

You should record changes into an .eco file as you make them. This gives the best back-annotation results. For more information, see Recording ECO Changes.

For more details on the differences of these methods, see Recording Versus Generating an ECO File in the *Concepts Guide*.

See also: Comparing Two Versions of a Design

Tip: If you want to compare the schematic to the PCB layout, export a PADS-format ASCII netlist file (.asc) from the schematic tool before comparing the designs.

Procedure

1. Open the Windows command prompt.

   For Windows 2000, Windows NT, and Windows XP, do the following:

   - From the **Start menu,** select **Programs > Accessories > Command Prompt**.

2. Change to the folder where the ecogen.exe file resides, for example, C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs.

3. To run ECOGEN, enter the **ecogen** command using the following format:

   ```
   ecogen <new_design> <original_design> [<eco_out>] [-u<unused net
   name>] [-e<error file>] [-d<report file>] [-r[a][p]] [-a<attr mask>]
   [-f] [-l <rules mask>] [-n<new_design_title>][-
   o<original_design_title>] [-g] [-s] [-m] [-q] [-i <output unit>]
   ```

   Tips:

   - You must enter the newDesign, originalDesign, and ecoFile command-line arguments in the given order.
   - If an argument contains spaces, enclose the argument with double-quotes " ".
   - You can also type ECOGEN@<command_file>, where the command file contains all the parameters on one line.
   - ECOGEN ignores the reuse definition and uses the actual elements in the physical design reuse for comparisons.

Table 35-1 contains the parameter usage. Square brackets [ ] enclose optional parameters.

Table 35-1. Parameter Usage

Parameter	Specifies
newDesign	The design file containing the changes you want to place into the original design.
originalDesign	The design file that you want to update so that it matches the new design.
[eco_out]	The ECO command file containing ECO directives. You can import this file into the original design to update the original file.

Table 35-1. Parameter Usage (cont.)

Parameter	Specifies
[-aAttributeMask]	Attribute comparison and the attributes to be compared (AttributeMask) This mask selects the object types for which to compare attributes. The mask is a string of object types separated by commas without spaces. You can specify these object types: PCB,PART,PARTTYPE,PARTDECAL,NET,NETCLASS,PIN Specify the AttributeMask as a string of object types separated by commas without spaces. for example: -aPART,NET If you want to ignore all attributes during comparison, do not specify the -a switch. If you do not use this switch, existing attributes are preserved and no ECO commands are generated for attribute modification. To compare part and net attributes, for example, for a PADS Logic schematic, use the -aPART,NET switch value. To compare board, part, net, and pin attributes, for example, for a DxDesigner schematic, use the -aPCB,PART,NET,PIN switch value To compare the attributes for all object types, use the -a PCB,PART,PARTTYPE,PARTDECAL,NET,NETCLASS,PIN switch value.
-dReportFile	The file containing the differences between the original design and the new design.
[-eErrorFile]	The file containing ECOGEN status and error messages. ECOGEN automatically opens the default editor you chose during installation to display the error file.
[-f]	Enables comparison of part decal assignment.

Table 35-1. Parameter Usage (cont.)

Parameter	Specifies
[-g], [-q]	Tips: • Do not use either the -g or -q switch if you want to compare net names and reference designator names, and rename as necessary. • These switches are best used to minimize changes to routed traces. Selecting this option may result in the positional swapping of parts. **Example:** If routed trace changes are minimized when R1 and R12 are swapped, simultaneously rename R12 to R1 and R1 to R12, and then reconnect R1 and R12 to the original nets.
	Use the -g switch to disable reference designator and net name comparison. Use this switch to compare connectivity and topology (not names) and rename as necessary. Compare differences using pin names, part type names, and so on.
	Use the -q switch to suppress part renames. Use this switch to compare net names and reference designators, but prefer adding or deleting parts to renaming parts. Compare differences using reference designators or net names on the basis that few reference designators have been renamed, but nets have not been renamed.
[-iOutputUnit]	Specifies an output unit for dimensional values in the ECO and report files. (If you do not specify this switch, ECOGEN uses the output unit of the new design.) You can specify an output unit of: BASIC, MILS, INCHES, or METRIC. If you specify an output unit that ECOGEN does not recognize (for example, if you mistype the unit name), the command uses BASIC.

Table 35-1. Parameter Usage (cont.)

Parameter	Specifies
[-lRulesMask]	Specifies rules comparison and the object types, rule types, and rule kinds to be compared (RulesMask). For FulesMask, you can specify: • Object types: PCB, NET, NETCLASS, PINPAIR, and GROUP. If you do not specify any object types, rules for all object types are compared. • Rule types: Clearance, Routing and High Speed (CLR, RT, HS). If you do not specify any rule types, all three types are compared. • Rule kinds: General, Conditional, and Differential Pairs (GEN, CON, DFP). If you do not specify any rule kinds, all three kinds are compared. Specify the RulesMask as a string of object types separated by commas without spaces. For example: `-lNETCLASS,GROUP,CLR` If you do not specify a RulesMask, ECOGEN, compares rules of all types and kinds on all object types.
[-m]	Enables comparison of part placement. When comparing the PCB layout to the schematic, this feature is available only for DxDesigner schematics containing placement information created by ePlanner.
[-nNewDesignTitle]	The string used in the report header for the new design name.
[-oOldDesignTitle]	The string used in the report header for the original design name.
[-r[a][p]]	Enables comparison of only ECO registered [p]arts or [a]ttributes. Use the -rap switch to compare only ECO registered parts and attributes. Use the -ra switch to compare only ECO registered attributes. Via attributes are not ECO registered and cannot be added, deleted, or changed during the ECO process. Use the -rp switch to compare only ECO registered parts. Do not use the -r switch if you want ECOGEN to compare all parts and attributes. Do not use the -rap or -rp switches if you want to compare mechanical or non-electrical parts on the designs.
[-uUnusedNetName]	Indicates an unused pins net that contains pins that have no logical net association. This net is a result of SPECCTRA routing. Use the netname you used in SPECCTRA. The maximum netname length is 47 characters. You can use any alphanumeric characters except curly braces { }, asterisks *, spaces question marks, or commas.

For example:

```
ecogen \ePD\3.1\project\logic.asc C:\PADS Projects\pcb.asc design.eco
-lNET,CLR -uNOT_CONNECTED
```

In this example, ECOGEN:

- Compares the original design (before changes), logic.asc, with the new design, pcb.asc
- Creates an ECO file, design.eco, which can be used for updating the original design, logic.asc
- Compares Clearances Rules for nets
- Specifies that NOT_CONNECTED is an unused pins net as a result of SPECCTRA routing.

Limitations of Rules Comparison

- ECOGEN always compares component and decal rules and Fanout and Pad entry rules, even if you do not specify the -l switch.
- ECOGEN does not generate change information for layers. For layer-dependent rules such as conditional rules, make sure the designs being compared have the same number of layers.

Related Topics

Comparing Two Versions of a Design

Chapter 36
Reports

In this chapter:

- Creating Reports
- Creating Physical Design Reuse Reports

Creating Reports

You can use the Reports dialog box to create reports containing properties or netlist information for the design. In addition to several predefined report formats, you can create custom report formats.

In this topic:

- Creating a Report
- Creating a Report Using an Assembly Variant
- Adding or Removing Report Formats

Creating a Report

To create a report:

1. **File** menu > **Reports**.
2. In the Select Report Files for Output list, select one or more report formats.

   Tip: If the report format you want is not available, you can create a custom format.
3. Click **OK** or **Apply**.

   Result: The report is written to C:\PADS Projects\report.rep and displayed in the default text editor.

   Tip: If you selected more than one report format, report.rep contains all the reports.

Creating a Report Using an Assembly Variant

You can create a report based on an assembly variant instead of the base design.

1. **File** menu > **Reports**.

2. In the Select Report Files for Output list, select one or more report formats.
3. Select the **Use Assembly Variant** check box, and then select a variant from the Name list.

   Tip: Uninstalled parts are listed as <<Not Installed>> and substituted parts are listed with the correct part type. Some data may not reflect the substituted parts data, for example, the Unused report reflects the gate assignments for the Default part type, but not the gate assignments for the Substituted part. If the Use Assembly Option check box is cleared, raw database data is reported.
4. Click **OK** or **Apply**.

   Result: The report is written to C:\PADS Projects\report.rep and displayed in the default text editor.

   Tip: If you selected more than one report format, report.rep contains all the reports.

Adding or Removing Report Formats

You can add or remove report formats from the Select Report Files for Output list. If you want to add a report format, you first create the format file using Report Generation Language.

See also: Customizable Reports in the *Concepts Guide*

To add or remove report formats:

1. **File** menu > **Reports**.
2. If you want to add to the list a custom report format, or import a report format file, do the following:
   a. Click **Add**.
   b. In the Report Format File dialog box, browse to the format file and click **Open**.

      Result: The format is added to the list and copied to the *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Settings* folder.
3. If you want to remove a report format from the list, select the format and click **Delete**. You cannot delete the Unused, Statistics, and Limits report types.

   Result: The format is removed from the list and the file name extension is changed from .fmt to .del.
4. Click **OK** or **Apply**.

Related Topics

Creating Reports in the *Concepts Guide*

Creating Jumper List Reports

Creating Physical Design Reuse Reports

You can create a report that contains the following information of the selected physical design reuse: name, type, and elements.

To create a report:

1. Select the physical design reuse.

   See also: To Select a Physical Design Reuse

2. Right-click and click **Report Contents**.

Result: The report is written to C:\PADS Projects\rules.rep and is displayed by the default text editor.

Chapter 37
Checking a Design for Errors

In this chapter:

To Compare Test Points

Use Compare Test Points to compare test point settings in ASCII format, between the current file and another file. Compare Test Points differs from a complete database integrity check because it only compares test point lists.

To compare test points:

1. You need an ASCII file of test points from an older version of the design file.

   See also: To Create a Test Point ASCII File

2. Click **Compare Test Points** from the **Tools** menu. The Compare Test Points dialog box appears.

3. Locate or type the name of the ASCII test point file to compare against. You do not need to type the .asc extension.

4. Click **OK**. PADS Layout creates an internal ASCII test point list of the open file and compares it to the one you specified.

   If the ASCII file you compare against is a version earlier than PowerPCB 2.x, or if a test point section is not present in the ASCII file, an error message appears. Create another file to compare against.

   When differences exist, the output error file tpasc.lst opens listing the differences.

To Create a Test Point ASCII File

To create an ASCII test point list of an older design:

1. Open the .pcb file in PADS Layout.
2. Click **Export** from the **File** menu. The File Export dialog box appears.
3. Type a name in the **File name** box.
4. Click **Save**. The ASCII Output dialog box appears.
5. Click the PowerPCB V3.0 Format from the list.
6. Click **Current** from the **Units** list.
7. Click **Select All** to select all check boxes.
8. Click **OK**.

DFF, Design For Fabrication

Process Flow for Using DFF Audit

Requirement: You need the CAM350 Link license option to use this. On the Help menu, click Installed Options and on the Options tab, check if the option is available in your license.

Use the following process to resolve any fabrication errors:

1. Lay out the design using PADS Layout and PADS Router.
2. Create CAM documents.
3. Set up Verify Design preferences for fabrication checks.
4. Run Verify the Design.
5. Use the layout editor in conjunction with the Verify Design dialog box to view and correct fabrication errors.
6. Repeat steps 3 through 5 until fabrication errors are resolved.

DFF Audit Process Flow using CAM350 Link

Requirement: You need the CAM350 Link license option to use this. On the Help menu, click Installed Options and on the Options tab, check if the option is available in your license.

If you use CAM350, use the following process to resolve fabrication errors:

1. Lay out the design using PADS Layout and PADS Router.

2. Create CAM documents.
3. Output the CAM documents to CAM350.
4. Set up Verify Design preferences for fabrication checks.
5. Run Verify the Design.
6. Use the layout editor in conjunction with the Verify Design dialog box to view and correct fabrication errors.
7. Use CAM350 to view errors. Use PADS Layout to correct fabrication errors.
8. Repeat steps 3 through 7 until fabrication errors are resolved.

Performing a Test Point Audit

Use DFT Audit to manage test points used to support In Circuit Testing (ICT) procedures. Test points help you meet your ICT requirements early in the design process and can improve your productivity by reducing design iterations.

When you run DFT Audit, PADS Layout automatically transfers the design to PADS Router. Using options you set in PADS Layout, PADS Router analyzes all nets for accessibility and adds test points to routed accessible nets. Note that PADS Router may reroute nets during DFT Audit. When PADS Router is done, it transfers the design back to PADS Layout. For nets that PADS Router determines to be inaccessible, PADS Layout can optionally add test points, which are placed outside the board edge. When DFT Audit finishes, the DFT Audit Board Report appears.

Requirement: PADS Router must be installed and licensed for you to run DFT Audit in PADS Layout. You can run DFT Audit regardless of the layer limits in the PADS Router license.

Tip: If you are using BGAs, fan out the BGAs before running DFT Audit.

To run DFT Audit:

1. **Tools** menu > **DFT Audit**.
2. On the DFT Audit dialog box, click the tab for which you want to modify the test point placement options and properties:
   - Options
   - Properties
   - Assignment
3. Modify the properties on the tab as required.
4. Click **Run**. The automatic audit process starts.

Tip: DFT Audit honors any test point keepouts defined in the design.

Related Topics

Design for Test in the *Concepts Guide*

Placing Test Points Automatically in the *PADS Router Concepts Guide*

Placing Test Points

Several options for placing test points are available to you on the Options tab of the DFT Audit dialog box.

Tip: To reduce design iterations, consult with your automated test engineers when setting DFT Audit options.

Use the Options tab to:

1. **Tools** menu > **DFT Audit** > **Options** tab.
2. Define how test points are created using options in the Create test points area. You can preserve existing via properties and add test point vias to routed, accessible nets.
3. Set probe via test point properties using options in the Probe through area.
4. Place via test points on a grid using options in the Place via points using area.
5. Specify test point nail diameters using options in the Available Nail Diameters area.
6. Set minimum pad areas using options in the Minimum Pad Probing Sizes area. Set minimum pad probing sizes for both vias and component pins to ensure that there is sufficient pad area for probe contact.

Related Topics

Performing a Test Point Audit

Setting Test Point Properties

Several test point properties are available to you on the Properties tab of the DFT Audit dialog box.

Tip: To reduce design iterations, consult with your automated test engineers when setting DFT Audit options.

Use the Properties tab to:

1. **Tools** menu > **DFT Audit** > **Properties** tab.
2. Specify the minimum distances between the probe and other design objects using options in the Probe Minimum Distances area.

Tip: The clearances needed between a probe and another design object are mostly based on the physical constraints of the Automated Test Equipment (ATE) used by In Circuit Testing (ICT) procedures. The probes extending out of the ATE fixture must make contact with the PCB without any obstacle. This means that test points must keep a fixed distance from component bodies, pads, mounting holes, and board edge, and must also have the minimum spacing between them.

3. Specify the maximum length of trace stubs required to make a net accessible to a test probe by typing the length into the **Stub Length** box.

4. Specify zero or more than one nail pins on a net using options in the Multiple Test Point Nets area. By default, all nets are set to receive one probe, or nail pin.

   If you want to change the number of nail pins for a net to receive, in the Multiple Test Point Nets area, type the new value in the **Nail Pins** cell for the net.

   Tips:

   - To display only nets with no nail pin or with more than one nail pin, select the Show Only Nets with Nail Pins Not Equal to One check box.
   - If you do not want any nail pins on a net, double-click the Nail Pins cell for the net and type zero (0).
   - To sort the list by a different column, click the column header at the top of the list.

Related Topics

Performing a Test Point Audit

Setting Test Point Assignment Eligibility

Use the Assignment tab to prevent or favor assigning test points to components or to via types. By default, all pins on a net are available for test pin assignment and are evenly weighted as test point candidates.

Tip: To reduce design iterations, consult with your automated test engineers when setting DFT Audit options.

Setting test point assignment eligibility for a component or via:

1. **Tools** menu > **DFT Audit** > **Assignment** tab.
2. To prevent use as a test point, select the **Exclude** check box.
3. To favor use as a test point, select the **Prefer** check box.
4. To apply even weighting, clear the **Exclude** and **Prefer** check boxes.

   Tips:

- To change the check box value for multiple rows, select the rows and then click a check box.
- To sort the spreadsheet by a specific column, click that column's header.

Related Topics

Performing a Test Point Audit

Probing the PCB Top Side Only

To probe the PCB top side only:

1. Create a test point keepout on the bottom side of the board.
2. On the Tools menu click **DFT Audit**. The DFT Audit dialog box appears.
3. Click the Options tab.
4. In the Probe through area, select the **Probe Top Side** check box.
5. Click **Run**.

   Result: DFT Audit ignores the bottom side because it has a test point keepout defined. Only the top side of the PCB is probed.

Modifying a Jumper Pin that is a Locked Test Point

If you are modifying the pad stack of a jumper pin that is a locked test point, the “One of the jumper pins is locked as a test point. Do you want to apply the changes to jumper?” message appears. Click one of the following:

- **Yes**—Applies the change and maintains the locked test point status.
- **No**—Cancels the change.

If you move a jumper pin that is a locked test point, the “Via marked as Test Point. Proceed Anyway?” message appears. Click one of the following

- **Disable Lock Test Points**—Turns the Lock Test Points check box off in the Routing / General page of the Options dialog box.
- **OK**—Allows you to move the jumper pin and maintain the test point status; the jumper pin is locked in its new position.
- **Cancel**—Cancels the move.

Modifying a Pin that is a Locked Test Point

If you are modifying the pad stack of a component pin, the “Do you want to apply the changes to all components with decal type xxx or just the selected components?” message appears. Click All to apply the change to all decal types. Click Selected to apply the change to the selected decal. Click Cancel to cancel the change.

If the component pin is a locked test point, the “Include locked test points in Pad Stack update?” message appears when you click All or Selected. Click one of the following:

- **Yes**—Applies the change and maintains the locked test point status.
- **No**—Cancels the change.

Modifying a Route Attached to a Locked Test Point

If you attempt to move or modify a route that is attached to a via that is a locked test point, the “Command causes the move of Via(s) marked as Test Point(s). Continue command ignoring Test Points or keep them locked during modification of route?” message appears. Click one of the following:

- **Disable Lock Test Points**—Turns the Lock Test Points check box off in the Routing / General page of the Options dialog box.
- **OK**—Performs the modification and moves the via along with the route, ignoring the locked test point setting. The test point will be locked again in its new position.
- **Keep Locked**—Performs the modification but does not move the via; the locked test point location is maintained. You can modify the route, but the test point via does not move.
- **Cancel**—Cancels the modification.

Modifying a Via that is a Locked Test Point

If you modify the pad stack of a via, the “Are you sure you want to change all vias of type xxx?” message appears. Click Yes to apply the change; click No to cancel the change.

If the via is a locked test point, the “Include locked test points in Pad Stack update?” message appears when you click Yes. Click one of the following:

- **Yes**—Applies the change and maintains the locked test point status.
- **No**—Cancels the change.

If you do not include test points in the update, then the pad stacks for the test points are preserved by renaming them with a TP_ prefix; for example, changing the pad stacks for STANDARDVIA, the test point pad stack is renamed TP_STANDARDVIA.

If you modify the Via Name of a via that is a locked test point, the “Command causes change of via type for Via(s) marked as Test Point. Proceed Anyway?” message appears. Click one of the following:

- **Disable Lock Test Points**—Turns the Lock Test Points check box off in the Routing / General page of the Options dialog box.
- **OK**—Applies the change and maintains the test point status.
- **Cancel**—Cancels the change.

If you move a via that is a locked test point, the “Via marked as Test Point. Proceed Anyway?” message appears. Click one of the following

- **Disable Lock Test Points**—Turns the Lock Test Points check box off in the Routing / General page of the Options dialog box.
- **OK**—Allows you to move the via and maintain the test point status; the via is locked in its new position.
- **Cancel**—Cancels the move.

Move Sequential to Move Components, Unions, or Clusters with a Locked Test Point

If a part or union selected for Move Sequential contains a pin that is a locked test point, the “Component/Union xxx have pins marked as Test Points. Continue?” message appears. Click one of the following:

- **Yes**—Moves the part although it is a test point. The test point is locked in its new position.
- **No**—Skips the component or union.
- **Cancel**—Cancels the move sequence.

If a cluster selected for Move Sequential contains a pin that is a locked test point and Collapse Mode is on, the “Cluster xxx has members with pins marked as Test Points. Continue?” message appears. Click one of the following:

- **Yes**—Moves the cluster even though it is a test point. The test point is locked in its new position.
- **No**—Skips the cluster.
- **Cancel**—Cancels the move sequence.

Moving, Dispersing, or Aligning a Component, Cluster, or Union with a Locked Test Point

If you attempt to move a component with a pin that is a locked test point, the “Component pin(s) marked as Test Point. Proceed Anyway?” message appears.

Note: This applies to Rotate 90, Spin, Flip Side, Radial Move, Group Rotate 90, and Flip Group. This also applies to Automatic Cluster Placement operations, such as Collapse Cluster, and changing the decal of a component. Click one of the following:

- **Disable Lock Test Points**—Turns the Lock Test Points check box off in the Routing / Genearl page of the Options dialog box.
- **OK**—Performs the modification and moves the component, ignoring the locked test point setting. The test point will be locked again in its new position.
- **Cancel**—Cancels the modification.

Exporting to CAM350

You can translate a PADS Layout design using CAM350 Link.

Requirement: You need the CAM350 Link license option to use this. On the Help menu, click Installed Options and on the Options tab, check if the option is available in your license.

Procedure

1. Create a PADS Layout design.
2. On the File menu, click **Export**, and in the Save as type list, click **CAM350** and then click **Save**. The CAM350 dialog box appears.
3. Either create the CAM350 .cam file only or create the file and automatically launch CAM350.
4. Click a layer option for the CAM document.
5. Click an arc approximation tolerance value.
6. Type a file name and location for the .cam file to generate.
7. Click **OK**.
8. Run your manufacturing process in CAM350.

Result

Once the design file is in CAM350, you can perform DFM analysis on the design. If DFM errors exist, you can back-annotate the errors to PADS Layout to identify and correct them.

Subsequently, you can generate a new CAM350 database and verify that all DFM errors have been corrected. You can then create CAM outputs in either PADS Layout or CAM350.

The processing routines that translate PADS Layout data into CAM350 .cam files detect errors and warnings and report them to the standard PADS Layout error file (Layout.err).

Working with Markups

In this section:

- Adding Markups
- Saving Markups
- Exporting Markups
- Importing Markups
- Linking Design Objects to Markup Issues
- Unlinking Design Objects from Markup Issues

Adding Markups

Use the Markups dialog box to log issues concerning the design. You can add 2D line markups to issues in order to outline or highlight their location. You can also link design objects to markups. You can export the issues alone or export the design and any issues for viewing and logging additional information in visECAD.

Requirement

You must create at least one topic section to begin adding issues and 2D line markups.

Procedure

1. On the Edit menu, click **Markups**.
2. Click the **Add Topic** button to add a topic.
   For example, as you review the design for another designer, you spot several silkscreen placement issues. You could name the markup topic - Silkscreen.
3. Click the **Add Issue** button to add an issue.
   For example, your first issue could be the placement of a reference designator beneath a component. You could name the markup issue - Placement of R54 refdes.
4. Click the **Add Markup** button.
5. Right-click and choose your options before adding the 2D line.

Restriction: Line widths or the layer location of your 2D line markers are not maintained when exporting and re-importing.

6. Draw a 2D line to highlight the area of concern.
   For example, you could draw a rectangle around the offending refdes.
7. You can also link design items to the markup.
8. Save the Markups.

Result

- Issues are logged against the design and ready to be shared.
- Markups take the color of 2D lines on any given layer.

Related Topics

Importing Markups

Saving Markups

There are two ways to save Markups added to the Markups dialog box. The markups are not saved when you use File menu > Save (or the Save button on the dialog box, or Ctrl+S).

You must do one of the following:

- Click the Export button in the Markups dialog box to export the markups into a .cle encrypted collaboration file.
- Click File > Save as, to generate a snapshot of the design and create a .cle file to match it.

Exporting Markups

You can export markups for use in another software tool like CAMCAD and visECAD. There are several ways to export Markups. You can export them from the Markups dialog box or using File > Export into a simple .cle file which only contains the markup information. Alternatively, you can export a CC file which includes most design elements along with the markups.

Using the Markups Dialog Box

1. On the **Markups** dialog box, click the **Export** button.
2. In the Collaboration Data Save As dialog box, type a name for the .cle file.
3. Click **Save**.

Using File > Export

1. On the **File** menu, click **Export**.
2. In the File Export dialog box, in the Save as type list, click either **Collaboration Files (*.cle)**, or **CCE Files (*.cce).**
3. Type a name for the .cle file or the .cce file.
4. Click **Save**.

Result

Markups are exported into the file. If you placed markup lines on different layers in the design, this information is ignored and all markups are exported into a single layer in the .cle file.

Related Topics

Exporting CCE Files

Importing Markups

You can import Markups created in another software tool, like CAMCAD or visECAD. There are two ways to import Markups. You can import them using the Markups dialog box, or using File > Import.

Restriction

- Any markups that exist in the Markups dialog box will be deleted before the new data is imported.

Using the Markups Dialog Box

1. On the **Markups** dialog box, click the **Import** button.
2. In the Collaboration Data Import dialog box, browse for and select the .clb or .cle file to import.
3. Click **Open**.

Using File > Import

1. On the **File** menu, click **Import**.
2. In the File Import dialog box, in the Files of type list, click **Collaboration Files (*.clb, *.cle)**.
3. Browse for and select the .clb or .cle file to import.
4. Click **Open**.

Result

- The Markups dialog box is populated with the imported collaboration data.
- Some shapes imported from other software have no equivalent in PADS Layout (for example, sticky notes). They are represented as accurately as possible with 2D lines in PADS Layout.
- The text that accompanies any imported shapes is not located in the design, but appears in the Text box in the Markups dialog box.

Linking Design Objects to Markup Issues

You can link design objects to markups of issues. You can link components, nets, vias and drawings (board outline, 2D lines, keepouts).

Procedure

1. In the design, select one or more design objects.
2. In the Markups dialog box, right-click over the markup and click **Link Selected**.

Result

The design items are listed under each element type in the elements tree of the Markups dialog box.

Related Topics

Unlinking Design Objects from Markup Issues

Adding Markups

Unlinking Design Objects from Markup Issues

You can unlink items from being linked to markups and listed in the elements tree of the Markups dialog box.

Procedure

- In the Markups dialog box, in the elements tree, right-click over an item and click **Unlink**.

Related Topics

Linking Design Objects to Markup Issues

Using the 3D PCB Viewer

You can send your design to the 3D PCB Viewer to view your design in 3D. You can choose between running 3D Viewer in either dynamic view mode or in snapshot (static) view mode.

Requirement

In order to generate a 3D view of your design, you must have applied values to your design elements. For example, you need to set up thickness values in the Layer definition for the layers in the 3D view to show a thickness. And you must use the geometry.height attribute to give your decals a height value.

Snapshot View

Use Snapshot view to see what the current design looks like in 3D. You can have multiple Snapshot windows open.

- **View** menu > **3D View > Snapshot View**

  Result: A new instance of 3D PCB Viewer window is opened and it displays the 3D image of the current snapshot of PADS Layout design. Changes made in PADS Layout are not passed to the open 3D Viewer window.

Dynamic View

Use Dynamic view to observe the 3D view of changes made in your design. You can have only one Dynamic window open.

- **View** menu > **3D View > Dynamic View**

  Result: The Dynamic View command opens new 3D Viewer window that will operate in dynamic mode. If the dynamic version of 3D Viewer is already open, click the Dynamic View command to switch focus to the existing 3D Viewer window. Changes made in PADS Layout are passed to the open 3D Viewer window.

Running a Thermal Analysis

You can send your design information directly to HyperLynx Thermal. The .hyp, .emn, and .emp files are created and imported into HyperLynx Thermal.

Requirement

You must flood copper pours, and plane areas before exporting the files into HyperLynx Thermal.

Procedure

- On the **Tools** menu, point to **Analysis** and then click **Thermal Analysis**.

Result

As long as all the data is available, the design opens in HyperLynx Thermal ready to be analyzed. If copper pours and plane areas are not flooded, you must flood them and try again. If all the heights of components are not added to the geometry.height attribute of components, you are prompted for their heights.

Related Topics

Missing Height Dialog Box

Verify the Design

Verifying the Design

You can check for individual or all design errors using the Verify Design dialog box. Check for the following types of errors: clearance, connectivity, high speed, number of vias, plane connection, test point, fabrication, wire bond. It does not check reference designator, part type, or attribute labels for clearance violations.

In this topic:

- Running a Design Check
- Checking Design Error Results
- Troubleshooting Errors
- Save and Print Error Results
- Previewing Fabrication Errors in CAM files

Check Type Information

- **Tools** menu > **Verify Design**.

Running a Design Check

You can check for individual or all design errors using the Verify Design dialog box:

1. **Tools** menu > **Verify Design**.
2. Select a type of check in the Check area.

3. If the Setup button is available (unavailable for Maximum via count and Test Points checks):
    a. Click **Setup** to specify additional settings for the check.
    b. Specify settings in the Setup dialog box and then click **OK**.
4. Click **Start** to run the check.
5. After the check process completes, a message window might open with the number of errors found. Click **OK**.

Result: Error markers appear in the design at error locations and error details populate the Verify Design dialog box.

See also: Interpreting Error Markers

Checking Design Error Results

Reading Error Details

Error results for a type of check are listed in the Location box. Listed errors, contain their (X,Y) location, layer, and error type.

1. **Tools** menu > **Verify Design**.
2. Select an error from the list.

    Result: The explanation for the error appears in the Explanation box.
3. View the more specific, detailed information about the error selected in the Location list. The explanation includes information about any conflicting object.
4. Click **Close** when you are finished with the Verify Design dialog box.

The current error list remains whether the dialog box is open or closed. The error markers remain until you click Clear Errors, which clears the markers, not the errors.

Viewing Errors in the Design

You can view error markers in the design. Error markers in the work indicate the error type.

Requirement: Colors must be assigned properly in the Display Colors dialog box in order to see error markers. Errors in the design area appear in the color of the **Errors** check boxes per layer of the Display Colors dialog box. When an error is selected in the Location box, the error in the design area appears in the **Highlight** color.

- Select an error from the list.

    Result: The design window pans to the area of the error (unless Disable Panning is selected). This is more obvious if you are zoomed into your design.

Tips:

- You can prevent the design area from panning to errors when you select them if you select the **Disable Panning** dialog box.
- Error markers are not erased from the design until you click **Clear Errors** which clears the error markers, not the actual errors. You must correct the error in the design.

The current error list remains whether the dialog box is open or closed. The error markers remain until you click Clear Errors, which clears the markers, not the errors.

Interpreting Error Markers

After verifying the design, error markers are inserted in a PADS Layout database as database objects. Error objects present in a PADS Layout database transfer into PADS Router and are converted into PADS Router error objects during file load.

Most error objects in PADS Layout are converted to an identical PADS Router error object. The exceptions to these conversions are listed in the Notes column of the table below.

Table 37-1. PADS Layout/Router Error Objects

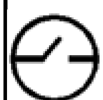
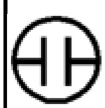
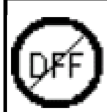
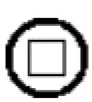
Marker	Error	Notes
	Testability and Connectivity	
	High-speed	
	Fabrication	
	Minimum/Maximum Length	
	Assembly (Latium Only)	Component height errors become assembly errors in PADS Router. Consequently, PADS Router assembly errors in PADS Router are converted to PADS Layout keepout errors in PADS Layout when the file is saved.

Table 37-1. PADS Layout/Router Error Objects (cont.)

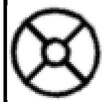
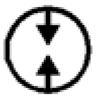
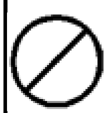
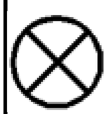
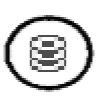
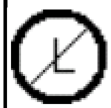
Marker	Error	Notes
	Drill to Drill	Drill to Drill errors in PADS Layout become fabrication errors in PADS Router. Consequently, drill to drill errors in PADS Router are converted to drill to drill errors in PADS Layout when the file is saved. Drill to Drill errors are reported for only one layer in a drill pair.
	Clearance	Body to body errors are clearance errors in PADS Layout. Body to body checking is not performed in PADS Router, only placement outline checking is performed as an assembly check. Therefore, body to body errors cannot exist in PADS Router. PADS Layout body to body errors are converted to placement outline errors in PADS Router - identified by Assembly error markers. Consequently, PADS Router Placement Outline errors in PADS Router are converted to PADS Layout placement outline errors in PADS Layout when the file is saved. **Tip:** Body to body errors are not returned from PADS Router.
	Keepout	Component height errors become assembly errors in PADS Router. Consequently, PADS Router assembly errors in PADS Router are converted to PADS Layout keepout errors in PADS Layout when the file is saved.
	Board Outline	
	Maximum Angle	
	Maximum Via Number	One marker is added for every net with too many vias.

Table 37-1. PADS Layout/Router Error Objects (cont.)

Marker	Error	Notes
	Latium Check	The Latium error marker indicates that you have Latium rules in your design. You can only check these rules by using the Latium Design Verification check in the Verify Design dialog box. Therefore, to be sure you do not have Latium errors, run the Latium check.

Tip: Within PADS Router you can assign an error as ignored. An ignored error status makes the error marker invisible but leaves the error in the database. PADS Layout supports storage of the ignored status on errors; however, ignored status can only be assigned or unassigned within PADS Router. In PADS Layout, ignored errors are not invisible. Ignored errors invisible in PADS Router saved to a PADS Layout file will be visible when the file is loaded into PADS Layout. But ignored errors remain as ignored after a file is saved in PADS Layout and loaded PADS Router.

Troubleshooting Errors

Clearance Errors

Clearance errors can report a *Same net clearance error* with an error explanation of *Distance between pads too small*. Although there is no specific SMD-to-SMD and SMD-to-Pad clearances, the SMD-to-Via clearance rule is used.

Subnet Errors

Connectivity checks often report a subnet error. There are several causes of subnet errors.

- An unassigned net—the net is not assigned to the plane area. Open the properties of the plane area shape and ensure that a net is assigned to the shape. You can only assign a net while the shape is in outline mode.
- A small unroute—turn off all layers in the Display colors dialog box. Assign a bright color to Connections and search the design for unroutes.

  Tip: You might find a connection to a pin that is not fully routed to the center of the pad. It's also easier to spot these if you assign a different color between pads and traces and enable the transparent mode to see the trace “underneath” the pad.
- An unplated component pin—check the pad stack of the component pin and ensure the Plated check box is selected.
- A via without a plane thermal—open the Via Properties dialog box and ensure the Plane Thermal check box is selected.

Exception: A Full Plane Check with Same Layer Connectivity Check will also display a subnet error which is caused only by isolated planes.
See also: Setting Up Plane Checking

Untied Plane Pin Errors

Plane checks sometimes report untied plane pin errors. There are a few causes of untied plane pin errors.

- An unplated pin or via
- A pin with a pad size on a plane layer that is greater than the drill size plus the drill oversize
- A pin without the thermal attribute set

Save and Print Error Results

You can alternatively view the current error results in your default text editor.

1. **Tools** menu > **Verify Design**.
2. To view the most recently run report results in the default text editor, click **View Report**. You can print or save the results from your default text editor.

Setting Default Report File Names

When you run a Verify Design check, a report file is created with a unique name for each check type.

1. **Tools** menu > **Verify Design**.
2. To specify the default file name of each report type click **Report File**.

   Result: Opens the Save As dialog box where you can name the report that you will run, if you don't want to use the default reference name for the report type.

   Tips:

   - If you do not give the report a unique name, it is always saved to the default file name.
   - If you assign it a name, that name is the default for the report type until you change it again.

Previewing Fabrication Errors in CAM files

After you run a Fabrication check that reports errors, you can preview the CAM layer document associated with the error. The Preview button is available only when fabrication checking is enabled and lists errors.

Requirement: You need the CAM350 Link license option to use this. On the Help menu, click Installed Options and on the Options tab, check if the option is available in your license.

1. **Tools** menu > **Verify Design**.
2. Select an **error** in the Location box.
3. Click **Preview**.

Result: The CAM Preview dialog box opens.

Tips:

- Since error markers are not added to the CAM document, it helps to zoom into the design area before you press Preview. If the CAM Preview doesn't open zoomed into the same area, you can press the Workspace button in the Cam Preview window to match the workspace view.
- In the CAM Preview window, click Setup to change the preview settings of the specified document in the preview area.

Back-annotating CAM350 Files

Backward annotation begins by parsing the specified CAM350 file for DFM errors. For each error listed in the file an error marker is added to the PADS Layout database, at the error location.

Requirement: You need the CAM350 Link license option to use this. On the Help menu, click Installed Options and on the Options tab, check if the option is available in your license.

Tip: Existing DFM error markers in the PADS Layout database are cleared before the CAM350 file is parsed.

Procedure

1. Perform DFM error analysis in CAM350 and save the file.
2. Open the design file in PADS Layout.
3. On the **Tools** menu, click **Verify Design**.
4. In the Verify Design dialog box, in the Check area, click **Fabrication**.
5. Click **Setup**.
6. In the Fabrication Checking Setup dialog box click Annotate DFF Errors.
7. Type the name or browse for the CAM350 file in which DFM errors were saved.
8. Click **Start** in the Verify Design dialog box. DFM error markers appear wherever errors exist in the design. They also appear in the Location list with their location.

9. Correct existing errors and regenerate the CAM350 database.

Adding Nets or Classes for Specific High-Speed Checks

You can run specific electrodynamic checks on nets or classes.

1. **Tools** menu > **Verify Design** > select **High Speed** check > click **Setup** > click **Add Nets** or **Add Classes**
2. Select a net or class and then click OK.

   Tips:

   - You can select more than one item using Ctrl+click
   - You can select a range using Shift+click or by dragging the cursor.

Related Topics

Setting Up High Speed (Electrodynamic) Checking

Setting Up EDC Parameters

Setting Up Clearance Checking

Use the Clearance Checking Setup dialog box to specify which clearances to check during a Clearance verification.

1. **Tools** menu > **Verify Design** > select **Clearance** check > click **Setup**.
2. Select the check box beside any types of clearance checks you want to enable and then click **OK**.

Restrictions:

- Clearance checking only checks the **visible** area of the design.
- Reference designator, part type, and attribute labels are not checked for clearance violations.
- Clearance checking does not check against 2D lines.
- Clearance checking is not performed on CAM Planes.

  Tip: Acute angles create solder bridging between conductive objects during board manufacturing.

Related Topics

Verify the Design

Design Rule Hierarchy in the *Concepts Guide*

Setting Up Checking for Isolated Stitching Vias

During design verification, you can use the Connectivity Check to report isolated routing vias and isolated stitching vias. (An isolated stitching via is a stitching via that is not connected to any hatch outline or copper area.) However, to have the check report isolated stitching vias, you must first set it up to ignore connections to CAM planes.

To find and report isolated stitching vias:

1. **Tools menu** > **Verify Design**.
2. In the Check area, select the **Connectivity** option and then click **Setup**.
3. In the Connectivity Checking Setup dialog box, select the **Ignore CAM plane connection for isolated stitching vias** check box.

Result: When you verify the design, the checking operation reports isolated stitching vias as errors and marks them in the design, along with the other errors found during checking.

Setting Up Latium Checking

To set up Latium checking:

1. **Tools** menu > **Verify Design** > select **Latium Design Verification** check > click **Setup**.
2. Select the **Net to All** check box to check clearance rules on each net or hierarchical level against any other obstacle type.
3. Select the **Board Outline** check box to check clearance rules for the board outline and board cut outs.
4. Select the **Off Board Text** check box to check for off-board text and to flag all instances of off-board text as clearance errors.
5. Select the **Keepout** check box to check for keepout restriction violations.
6. Select the **Same Net** check box to check clearances between objects along the same net, as specified in the Clearance Rules dialog box.

   See also: Design Rule Hierarchy in the *Concepts Guide*

7. Select the **Drill to Drill** check box to check clearances between all drill holes. Pad stack drill size plus the drill oversize value calculate the diameter for plated holes.

   Tip: Drill to drill errors are reported for only one layer in a drill pair.

8. Select the **Trace Width** check box to check traces in excess of minimum and maximum widths, specified in the Clearance Rules dialog box.

   See also: Design Rule Hierarchy in the *Concepts Guide*

9. Select the **Placement Outline** check box to:

- In default layer mode, check outline against outline on layer 20, not on electrical layers.
- In increased layer mode, check outline against outline on layer 120.

Tip: You can create outlines on layer 20 (or 120) that do not exactly match the actual component outline. By setting a larger outline on this layer, you can leave an area near a component open for other purposes. This check ensures this area is left open.

10. Select the **Via at SMD** check box to check for via at SMD restriction violations.

    See also: Pad Entry Rules dialog box

11. Select the **Differential Pairs** check box to check for differential pair restriction violations.

    See also: Creating Differential Pair Design Rules

12. Select the **Trace Length** check box to check for length restriction violations.

    See also: HiSpeed Rules dialog box

13. Select the **Test Point** check box to check test points on the design. Test Points checks for probe clearances, minimum via/pad sizes for probing, SMD pin probing, test points on component pin on the component side, test point count per net settings and nail diameter settings, and compares the settings against the setting in the DFT Audit program.

    See also: Performing a Test Point Audit

Tips:

- Reference designator, part type, and attribute labels are not checked for clearance violations.
- Latium design verification performs clearance checking on only the visible area of the design. When clearance checking against the board outline, the edge of the object is checked against the centerline of the board outline.
- Drill to Drill errors are reported for only one layer in a drill pair.
- Test point checking is the same whether you enable the checking on the Latium Checking Setup dialog box or on the Verify Design dialog box. If you plan to perform Latium design verification, you can eliminate an extra design transfer between PADS Layout and PADS Router by running test point checking with the other Latium checks.

Related Topics

Verify the Design

Setting Up Fabrication Checking

Use the Fabrication Checking Setup dialog box to enable fabrication checks, or to load DFF errors from a preexisting CAM350 database and annotate into the design.

Requirement: You need the CAM350 Link license option to use this. On the Help menu, click Installed Options and on the Options tab, check if the option is available in your license.

In this topic:

Running Fabrication Checks

Requirement: Fabrication checks require certain CAM documents, which are listed in Table 37-2. You may consider creating these files before running the Fabrication check.

Table 37-2. CAM Documents Required for Fabrication Checks

Check	CAM Document Layers Required
Acid Traps	All Electrical Layers
Copper Slivers	All Electrical Layers
Solder Mask Slivers	Top-Bottom Solder Mask Layers
Solder Mask Bridges	Top-Bottom Electrical & Solder Mask
Starved Thermals	All Negative CAM Plane Layers
Minimum Annular Rings	All Electrical Layers Solder Mask Layers
Silkscreen Over Pads	Top-Bottom Solder Mask & Silkscreen

- **Tools** menu > **Verify Design** > select **Fabrication** check > click **Setup**.

Checking Acid Traps

Use this check to flag small areas where acid will pool up. The check is run on all visible electrical layers as defined in the CAM documents.

1. Select the **Acid Trap** check box.
2. In the **Maximum Size** box, type a maximum value of the acid traps to detect. The areas of the “pools” that are flagged will be less than this value.
3. In the **Maximum Angle** box, type a maximum angle for traces, pads, or any other data that exists on the layer. Any items that form an angle smaller than this will be flagged as an acid trap.

Checking Slivers

Use this check to flag copper sliver and solder mask sliver areas. This compares the top solder mask layer against the top electrical layer and the bottom solder mask layer against the bottom electrical layer as defined in the CAM documents.

1. Select the **Slivers** check box.
2. In the **Minimum Copper** box, type a minimum value for copper slivers. This flags slivers with less area than this value.
3. In the **Minimum Mask** box, type a minimum value for solder mask slivers. This flags the slivers with a width less than this value, checking the top and bottom solder mask layers if they are visible.

Checking Starved Thermals

Use this check to flag invalid thermals where adjacent data overlaps the thermal spokes.

Restriction: Starved Thermals are only checked on (negative) CAM planes.

1. Select the **Starved Thermals** check box.
2. In the **Minimum Clearance** box type the percentage of the thermal's spoke that can be unblocked by another object. Any less of an opening will be considered “starved.”
3. In the **Minimum Spokes** list, select the minimum allowable number of the thermal's spokes that cannot be blocked by another object. Any less will be considered “starved.”

Checking Trace Width/Pad Size

Use this check to flag traces and pads that are too small. Checks all electrical layers as defined in the CAM documents.

1. Select the **Trace Width/Pad Size** check box.

2. In the **Minimum Trace** box, type a minimum trace width value. This flags traces with a width less than this value. This check runs on all visible electrical layers.
3. In the **Minimum Pad** box, type a minimum pad size. This flags pads with a diameter of less than this value. This check runs on all visible electrical layers.

Checking Silkscreen Over Pads

Use this check to flag silkscreen over pads on top and bottom layers as defined in the CAM documents.

1. Select the **Silkscreen Over Pads** check box.
2. In the **Minimum Gap** box, type the minimum allowable distance between silkscreen features and a region exposed by solder mask.

Checking Annular Ring

Use this series of checks to flag minimum annular rings on top and bottom layers, comparing electrical, drill, and mask layers.

- Select the **Annular Ring** check box to enable the Pad to Mask, Drill to Mask and Drill to Pad checks.

Pad to Mask

Use the Pad to Mask check to flag minimum clearance distances between a pad and its solder mask opening. The offset and annular ring is checked against the specified clearance value. This compares the top electrical layer against the top solder mask layer or the bottom electrical layer against the bottom solder mask layer.

1. Select the **Pad to Mask** check box.
2. In the box, type the minimum clearance value.
3. In the **Layers** list, select the layer to use for checking.

Drill to Mask

Use the Drill to Mask check to flag minimum clearance distances between a drill and its solder mask opening. The offset and annular ring is checked against the specified clearance value. This compares the top drill layer against the top solder mask layer or the bottom drill layer against the bottom solder mask layer.

1. Select the **Drill to Mask** check box.
2. In the box, type the minimum clearance value.
3. In the **Layers** list, select the layer to use for checking.

Drill to Pad

Use the Drill to Pad check to flag minimum clearance distances between a drill and its associated pad. The offset and annular ring is checked against the specified clearance value. This check is run on each specified layer.

1. Select the **Drill to Pad** check box.
2. In the box, type the minimum clearance value.
3. In the **Layers** list, select the layer to use for checking.

Checking Solder Bridges

Use this check to flag solder mask bridging. Solder can bridge and cause a connection to an adjacent object within the same mask opening. If the adjacent object is farther from the pad than this distance, even if it is exposed by the mask layer, it will not be identified as a bridge. This compares the top solder mask layer against the top electrical layer or the bottom solder mask layer against the bottom electrical layer as defined in the CAM documents.

1. Select the **Solder Bridges** check box.
2. In the **Minimum Gap** box, type the minimum clearance value.
3. In the **Layers** list, select the layer to use for checking.

Annotating DFF Errors

If you use CAM350 for checking fabrication errors, you can load DFF errors from a CAM350 file.

- In the **CAM350 File Name** box, Type a .cam path and file name or click Browse to navigate to the location of a file to back-annotate DFF errors into PADS Layout for design verification.

Related Topics

Fabrication Checks Definition in the *Concepts Guide*

Verify the Design

Setting Up High Speed (Electrodynamic) Checking

Use the Electrodynamic Check dialog box to enable high-speed checks for individual nets and classes or for the whole design.

Requirement: You must have specified your plane layers in the Layers Setup dialog box before you run an electrodynamic check. For a two-layer board, temporarily identify one of the layers as a plane layer.

In this topic:

Enabling High Speed Checks

You can enable checks for the entire design or if you need to make specific checks of certain nets or classes, you can add them to the Task List. You can enable different checks for each item in the Task List.

1. **Tools** menu > **Verify Design** > select **High Speed** check box > click **Setup**.
2. If you need to add specific nets or classes to the Task List, click **Add Nets** or **Add Classes**.

   Tip: (N) is added to Task list items that are nets and (C) is added to items that are classes.
3. Select an item from the Task List to enable High Speed checks for that item.
4. Select the check boxes for the types of checks you want to enable.

   Tips: To define checks for several nets or classes simultaneously, click more than one item in the list. If there are conflicting values between two selected items, the check boxes are dimmed but can be cleared and selected.
5. Repeat steps 2 and 3 for each remaining item.

Deleting Tasks

You can delete specific nets and classes from the Task List - deleting the custom checks.

1. **Tools** menu > **Verify Design** > select **High Speed** check box > click **Setup**.
2. Select the task in the Task List and click **Delete**.

Setting Electrodynamic Check Parameters

You can set up Electrodynamic Check Parameters in addition to activating the Electrodynamic checks. Use the Parameters dialog box to enter physical properties of the PCB, to customize checks and to request report detail level.

1. **Tools** menu > **Verify Design** > select **High Speed** check box > click **Setup**.

2. Click **Parameters**.

See also: Setting Up EDC Parameters

Setting Design Rules

You can access the design rules from the Electrodynamic check dialog box. Use the Rules dialog box to enter high-speed rules such as minimum and maximum lengths, gaps for parallelism and tandem checking, and other limits like stub length and daisy chaining.

1. **Tools** menu > **Verify Design** > select **High Speed** check box > click **Setup**.
2. If you need to edit or assign rules for checking, click the **Rules** button as a shortcut to the Rules dialog box.

See also: Design Rule Hierarchy in the *Concepts Guide*

Reusing Electrodynamic Check Settings

You can reuse your electrodynamic check settings. You can save the current electrodynamic check task, parameter and rules settings.

1. **Tools** menu > **Verify Design** > select **High Speed** check box > click **Setup**.
2. Click **Save** or **Save As** to store your settings to an .edp file.
3. Click **Open** to retrieve settings from an .edp file.

Related Topics

Setting Up EDC Parameters

Verify the Design

Setting Up EDC Parameters

Use the EDC Parameters dialog box to define global rules like layer thickness and copper thickness. You can also specify how detailed a design verification report you want.

In this topic:

- Setting Up Layer Definitions
- Setting Parallelism Check Details
- Setting Daisy Chain Report Details
- Setting Details for Other Checks
- Including Segment Coordinates in Segments Reports

- Listing Violations Only
- Excluding Segments under/within Pads from Calculations

Setting Up Layer Definitions

You may have already set up your layer definitions in the Layers Setup dialog box, but this duplicate of the Layer Thickness table allows you to make modifications or to set definitions for the first time if needed. Set these definitions before you run an electrodynamic check.

Board Thickness—is the total value of material and layer thicknesses in the current design units.

To set layer values:

1. If the layers are not set up properly, use the **Layers** button to gain easy access to the Layers Setup dialog box where you can define layer properties, names, and functionality.

   Requirement: You must have specified your plane layers in the Layers Setup dialog box before you run an electrodynamic check. For a two-layer board, temporarily identify one of the layers as a plane layer.

2. For each dielectric material layer, double-click the **Type cell** to select whether the "layer" is a Prepreg or Substrate layer.

   Alternative: Although it is a longer process, you can click a box and then click Edit instead of using the double-click method.

3. For each layer, double-click the **Thickness cell** and type a value.

   Exception: If no coating is required, set thickness to zero.

4. For dielectric material layers, double-click the **Dielectric cell** and type a dielectric constant value.

Tip: You can view and edit copper thicknesses by weight or design units. Click the copper thickness unit you want:

Weight (oz)	Weight of copper in ounces, per square foot
Design Units	Same unit of measure as the current database unit of measure

Setting Parallelism Check Details

You can select the extent of checking and reporting for Parallelism and Tandem checks.

1. In the Parallelism area, in the Check Against list, select the extent of checking. Select from:

Nets/Pin Pairs—Checks the parallelism and tandem rules against the entire net or pin pair.

Segments—Checks the parallelism and tandem rules against only individual segments.

2. In the Parallelism area, in the Report Detail list, select the extent of reporting. Select from:

   Net Names Only—Displays only net names and violations.

   Aggressors/Victims—Displays specific aggressor and victim nets.

   Segments—Displays segment coordinates and layers in addition to aggressor and victim nets.

Setting Daisy Chain Report Details

You can select the extent of reporting for the Stubs (Daisy Chain) check.

- In the Daisy Chain area, in the Report Detail list, select the extent of reporting. Select from:

  Net Names Only—Include the number of T points and whether the net is daisy chained.

  Stubs—Include the group of pins within each stub, the total stub length for each group, the number of T points, and whether or not the net is daisy chained.

  Pin Pairs—Include the pin to pin length of all pin pairs, the total pin pair length added together to form stubs, the number of T points, and whether the net is daisy chained.

  Segments—Include the coordinates and layer of all track corners, the pin to pin length of all pin pairs, the total pin pair length added together to form stubs, the number of T points, and the nets being daisy chained.

Setting Details for Other Checks

You can set various setting for other checks.

1. In the Other Checks area, in the Check Against list, select the extent of checking of Length and Delay rules. Select from:

   Nets/Pin Pairs—Check the Length and Delay rules against the entire net or pin pair.

   Segments—Check the Length and Delay rules against individual segments.

2. In the Other Checks area, in the Report Detail list, select the extent of reporting for capacitance, impedance, delay, and length. Select from:

   Nets—Include the starting and ending pins of nets and net values for capacitance, impedance, delay, and length.

Pin Pairs—Include pin-to-pin points, pin pair values, and net values for capacitance, impedance, delay, and length.

Segments—Include individual segment coordinates and segment values for capacitance, impedance, delay, and length.

3. Select the **Include Copper** check box to include copper polygons with signal names in the capacitance calculations.
4. Select the **Use FieldSolver for Calculations** check box to calculate electric parameters of transmission lines such as: impedance, delay (per unit length), and capacitance (per unit length).

See also: the BoardSim User's Guide.

Including Segment Coordinates in Segments Reports

You can include segment coordinates in reports where Segments has been selected in the Report Detail in any one of Parallelism, Daisy Chain, or Other Checks sections.

- Select the **Report Segment Coordinates** check box.

Listing Violations Only

You can list only items that contain violations in the high-speed report.

- Select the **Report Violations Only** check box.

Excluding Segments under/within Pads from Calculations

You can exclude trace segments under pads from calculations. When routing, traces are routed into the middle of pads. This final segment is excluded.

- Select the **Remove Segments under Pads** check box.

Related Topics

Verify the Design

Setting Up Plane Checking

Use the Mixed Plane Setup dialog box to set the type of plane checking.

In this topic:

- Checking Thermal Connectivity Only
- Checking Clearance and Net Connectivity

- Checking Same Layer Connectivity

Checking Thermal Connectivity Only

You can check your design for split/mixed or CAM plane thermal connectivity. Use this check to find pins or vias that do not have the thermal attribute set, or to find pins that are not within a plane area (thermals will not connect). You can set the thermal attribute in Jumper Pin, Pin, or Via Properties dialog boxes.

1. **Tools** menu > **Verify Design** > select **Plane** check > click **Setup**.
2. Click **Check thermal connectivity only**.
3. Click **OK**.
4. Click **Start** in the Verify Design dialog box to run the check.

Tip: Routed pins without the thermal attribute set are not marked as errors.

See also: Assigning Plane Thermal Attributes

Checking Clearance and Net Connectivity

You can check your design for split/mixed plane clearance and net connectivity errors. If the split/mixed plane is not connected in the design, a Plane Connect will be performed before the check is run.

Requirement: Before you check for clearance and net connectivity, a plane area for each net assigned to the plane layer in the Layers Setup dialog box must be present.

1. **Tools** menu > **Verify Design** > select **Plane** check box > click **Setup**.
2. Click **Check clearance and connectivity**.
3. Click **Ok**.
4. Click **Start** in the Verify Design dialog box to run the check.

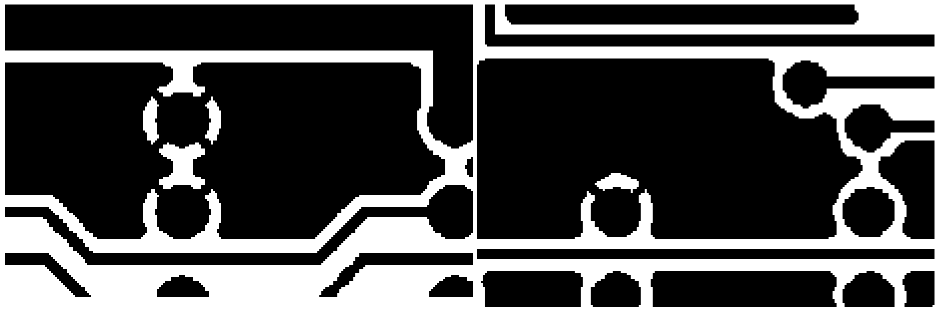
Tip: Pads on plane layers are checked for plating, a pad size on a plane layer that is greater than the drill size plus the drill oversize, or connection to a plane net.

Checking Same Layer Connectivity

Ensures that plane areas are continuous on the split/mixed plane layer. Plane areas of a particular net must have copper contact with each other without going to another layer.

1. **Tools** menu > **Verify Design** > select **Plane** check box > click **Setup**.
2. Click **Check clearance and connectivity.**

3. Select the **Same Layer Connectivity check box.**
4. Click **Ok**.
5. Click **Start** in the Verify Design dialog box to run the check.



Proper same layer continuity
The two planes are connected by thermal spokes and on the same layer.

Error condition
Two planes with the same net exist but are connected by a pad and a trace on a different layer.

Related Topics

Verify the Design

Troubleshooting Errors

Setting Up Wire Bond Checking

Use the Wire Bond Checking Setup dialog box to specify which wire bond rules to check during a Wire Bonds verification.

1. **Tools** menu > **Verify Design** > select **Wire Bonds** check box > click **Setup**.
2. Select the check box beside any types of checks you want to enable and then click OK.

Tip: Unchecked rules appear in the wire bond report as “Not Set.”

Restriction: The rule is not checked unless you select it and enter a value.

Related Topics

Wire Bond Rules Dialog Box

Verify the Design

Checking the Plane Connection for Continuity

When you run a continuity check in Verify the Design from the Tools menu, the net checks are connected if no errors are found.

Run the Plane check to see whether a pad exists in the connecting pad stack for the plane level, for example; is the pad size more than 0 or does the drill size exceed pad size? For links to SMD pads, it checks whether the pad-to-via connection connects to the plane.

Chapter 38
Dimensioning

Dimensioning Process

The Dimensioning Toolbar contains tools for dimensioning your design. The Dimensioning Toolbar is also available in the PCB Decal Editor.

1. Click the **Dimensioning Toolbar** button.
2. Set your preferences for dimensions on the Dimensioning tab of the Options dialog box. Give specific attention to the Layers where the dimensioning objects are added.

   Warning: Text and lines added to electrical layers will be fabricated in copper! Select a documentation layer for documentation objects.

   See also: Dimensioning Options
3. Before attempting to dimension a design object, set the selection filter to the type of object for dimensioning.

   See also: Using the Selection Filter
4. Based on the type of object to be dimensioned, select a dimensioning button.

   See also: Creating Dimensions
5. If you are creating multiple linear dimensions, different measurement styles are available. Right-click and click a measurement style.

   See also: Selecting a Dimension Measurement Style
6. Based on the dimension you are creating and line widths of extension lines and objects in the design, you may need to select a difference edge preference. Right-click and click on of the Edge Preferences.

   See also: Setting an Edge Preference
7. Unless you are dimensioning a single basic object, you may need to manually identify the points to dimension. Right-click and click one of the Snap Modes.

   See also: Snapping Dimensions Points
8. Select the object to dimension and then click to place the dimension.
9. When finished dimensioning, select the Select button to end dimensioning operations. The Select button in the standard toolbar closes the open toolbar.

Tip: You can also use the View > Clearance dialog box to add dimensions to your design.

Related Topics

View Clearance Dialog Box

Creating Dimensions

You can create dimensions automatically and manually with the dimensioning tools. If a design object is in a selected state when a dimensioning button is clicked, one item is dimensioned. After dimensioning the first item, you select the next data item, and then click the button again. Dimensioning modes are sticky, allowing you to select and dimension additional data items. To exit a dimensioning mode, click the Select button on the toolbar, or click **View > Toolbars > Dimensioning Toolbar** to close the toolbar.

- Automatic Dimensions Dimensioning with Auto Mode
- Vertical Dimensions
- Aligned Dimensions
- Rotated Dimensions
- Angular Dimensions
- Arc or Circle Dimensions
- Leader Line Dimensions

Automatic Dimensions Dimensioning with Auto Mode

Auto mode automatically establishes the orientation of newly added dimensions. If you select a line segment with Auto active, the system measures the length of the line segment and adds a dimension line parallel to the selected line. Auto is similar in functionality to Aligned mode except that Auto does not allow the selection of individual points. Snap mode is unavailable while you use Auto.

Auto dimensions arcs and circles in the same manner; click **Auto** and select the arc or circle to dimension. The advantage to using Auto mode is that you can create dimensions for line segments and/or arcs without having to switch modes.

To dimension with Auto mode:

1. **Dimensioning Toolbar** button > **Auto** button.
2. Select the line segment or arc you want to dimension. An image of the dimension is attached to and moves with the pointer to locate the dimension line.
3. Click to indicate the location for the dimension line.

Restriction: Snap Mode is unavailable for Auto dimensioning since this option only works with selected line segments.

Horizontal Dimensions

Horizontal mode creates dimensions with the dimension line always being in a horizontal orientation, X direction. Selected line segments that exist at an angle have their base distance measured. Line segments in a pure vertical orientation, Y direction, cause an error message to appear. The same orientation rules apply for points entered through the pointer.

To create new dimensions with Horizontal mode:

1. **Dimensioning Toolbar** button > **Horizontal** button.
2. Select a Snap Mode and Edge Preference, if necessary.
3. Select the line segment to dimension, or indicate two points. The dimension dynamically attaches to the pointer.
4. Click to indicate a location for the dimension line.

You can continue to dimension additional data items.

Vertical Dimensions

Vertical mode creates dimensions with the dimension line always being in a vertical orientation, Y direction. Selected line segments that exist at an angle have their height distance measured. Line segments in a pure horizontal orientation, X direction, cause an error message to appear. The same orientation rules apply for points entered through the pointer.

To create dimensions with Vertical mode:

1. **Dimensioning Toolbar** button > **Vertical** button.
2. Select a Snap Mode and Edge Preference, if necessary.
3. Select the line segment to dimension, or indicate two points. The dimension dynamically attaches to the pointer.
4. Click to indicate a location for the dimension line.

Vertical remains active, allowing you to dimension additional data items.

Aligned Dimensions

Aligned creates dimensions with the dimension line at the same angle as the selected line or points. Aligned is similar to Auto, but in addition to dimensioning selected lines, Aligned allows you to indicate two points that identify the measurement locations.

To create an aligned dimension:

1. **Dimensioning Toolbar** button > **Aligned** button.

2. Select a Snap Mode and Edge Preference, if necessary.
3. Select the line segment to dimension, or indicate two points. The dimension dynamically attaches to the pointer.
4. Click to indicate a location for the dimension line.

Aligned remains active so you can dimension additional data items.

Rotated Dimensions

Rotated mode creates dimensions similar to Aligned mode, but offers the additional flexibility of rotating the entire dimension object by a specified number of degrees. This feature is convenient when several dimensions exist in one location, because it allows you to offset some dimensions for easy viewing.

When dimensioning in Rotated mode, the system displays a prompt window specifying the current angle, with zero being in the positive Y axis (toward the top of the work area). You can type a different value to rotate and offset the entire dimension object. Positive values rotate counterclockwise; negative values rotate clockwise.

To create dimensions in Rotated mode:

1. **Dimensioning Toolbar** button > **Rotated** button.
2. Select a Snap Mode and Edge Preference, if necessary.
3. Select the line segment to dimension, or indicate two points.
4. Type the rotation value you want to use and click **OK**. The dimension dynamically attaches to the pointer.
5. Click to indicate a location for the dimension line.

Rotated remains active, allowing you to dimension additional data items.

Angular Dimensions

Angle mode measures and dimensions an angle in degrees. You can specify this angle by two line segments, a line segment and two points, or four points. These settings are controlled by the available Snap modes.

To create an angular dimension:

1. **Dimensioning Toolbar** button > **Angular** button.
2. Right-click and click a Snap Mode option, if necessary.
3. Select the lines to dimension, or indicate two points to define the angle. The angular dimension dynamically attaches to the pointer.

4. Click to indicate a location for the dimension line.

Angular remains active, allowing you to dimension additional data items.

Arc or Circle Dimensions

Arc creates dimensions that measure and document the size of arcs or circles, with the calculation being the radius or the diameter.

To dimension an arc or circle:

1. **Dimensioning Toolbar** button > **Arc** button.
2. Right-click and click an Edge Preference, if necessary.
3. Select the arc or circle to dimension. The radial dimension dynamically attaches to the pointer.
4. Click to indicate a location for the dimension line.

Arc remains active, allowing you to dimension additional data items.

Tip: Arc mode works best when the Options dialog box is set to **Manual Position** for text. This lets you create a leader line affect for documented arcs and circles.

Leader Line Dimensions

Leader creates a standard pointer to the selected information or point. It is simply a line with an arrow at one end and a user-specified text string at the other.

To create a leader line:

1. **Dimensioning Toolbar** button > **Leader** button.
2. Right-click and click a Snap Mode if you are creating the leader from a specific point, such as the midpoint of the selected line.
3. Select the line to dimension, or click to indicate a point for the arrow. A line with an orthogonal corner dynamically attaches to the pointer.
4. Click to add a vertex and anchor the leader line. Double-click to complete the leader line.
5. Type a text string for the leader line and click **OK**.

Related Topics

Dimensioning Process

Selecting a Dimension Measurement Style

In this Topic:

- Creating Chained Dimensions
- Creating Baseline Dimensions

Creating Chained Dimensions

Continue type dimensioning creates dimensions in a daisy-chained fashion. With **Continue** on, the last entered point becomes the new first point of subsequent dimensions.

To perform Continue type dimensioning:

1. Click the **Dimensioning Toolbar** button.
2. Click the appropriate dimensioning button on the Dimensioning toolbar, such as Horizontal, Vertical, or Aligned.
3. Right-click and click **Continue**. A check mark appears next to Continue, indicating that it is active.
4. Click to indicate the location of the first dimension point. The prompt "Continue. Enter second point" appears.
5. Click to indicate the location of the next dimension point.
6. Move the pointer to position the dimension line and click to indicate the first dimension object.
7. Repeat steps 4 and 5 for all points.
8. To end dimensioning, right-click and click **Continue** again. The check mark next to Continue is removed.

Creating Baseline Dimensions

Baseline dimensioning defines several measurements from the same starting point. To perform baseline dimensioning:

1. Click the **Dimensioning Toolbar** button.
2. Click the appropriate dimensioning button on the Dimensioning Toolbar, such as Horizontal, Vertical, or Aligned.
3. Right-click and click **Baseline**. A check mark appears next to Baseline, indicating that it is active.

4. Click to indicate the location of the baseline dimension point. The prompt “Baseline. Enter second point” appears on the Status Bar.
5. Click to indicate the location of the next dimension point.
6. Move the pointer to position the dimension line and click to indicate the first dimension object.
7. Repeat steps 4 and 5 for all points.
8. To end dimensioning, right-click and click **Baseline** again. The check mark next to Baseline is removed.

Setting an Existing Extension Line as the Baseline

You can set an existing extension line as the new baseline:

1. **Dimensioning Toolbar** button > **Select** button.
2. Select the extension line to act as the new baseline.
3. Right-click and click **Baseline**. A check mark appears next to Baseline, indicating that it is active.
4. Click a dimension button on the Dimensioning toolbar that matches the selected extension line orientation.
5. Click to indicate the location of the next dimension point.
6. Move the pointer to position the dimension line and click to indicate the first dimension object.
7. Repeat steps 4 and 5 for all points.
8. To end dimensioning, right-click and click **Baseline** again. The check mark next to Baseline is removed.

Related Topics

Dimensioning Process

Dimensioning Alignment Tool and Arrow Options

Setting an Edge Preference

The Edge Preference enables you to measure and dimension objects from the center of the drawn line or from either line edge. The default measures all lines from their physical centerline, regardless of the drawn line width. While this would be necessary for a part to part dimension, a dimension that documents the design rule spacing between two adjacent routes would need the edge preference set to Use Inner Edge.

1. **Dimensioning Toolbar** button > click a dimensioning style button.
2. Right-click and click an Edge Preference described in the table below. A check mark appears next to the selected mode.

Table 38-1. Edge Preference Choices

Edge Preference	Description
Use Centerline	Measures from the center of the object.
Use Inner Edge	Measures from the edge closest to the other point.
Use Outer Edge	Measures from the edge furthest from the other point.

Tips:

- This mode remains in effect until you select another. Also use the shortcut menu to set the snap mode.
- Line length measurements are also affected by the edge preference option. To ensure accuracy of standard dimensioning tasks, reset this option to Use Centerline, after using one of the alternate settings.

Related Topics

Dimensioning Process

Dimensioning in the *Concepts Guide*

Snapping Dimensions Points

A Snap mode lets you precisely identify points to dimension from by forcing the pointer to snap to a specified part of an object, such as the corner or center.

In this topic:

- Using a Snap Mode
- Adjusting Snap While Dimensioning

Using a Snap Mode

The system defaults to auto-snap mode. In this mode, both endpoints of a selected line segment or an entire arc are selected to identify the dimension points. You can select a more manual snap mode.

1. **Dimensioning Toolbar** button > click a dimensioning style button.
2. To disable auto-snap, right-click and click one of the snap modes listed in the table below. A check mark appears next to the selected mode.

Tip: This mode remains active until you select another one, or until you exit PADS Layout.

3. To re-enable auto-snap, right-click and click the snap mode to remove the checkmark from the currently-selected mode.

Restriction: Snap modes are unavailable for Auto and Arc type dimensioning styles. These styles only support the auto-snap mode because they apply to selected line segments and arcs, not selected points.

Adjusting Snap While Dimensioning

You can adjust snap mode while selecting dimensioning points. This capability can be important when dimensioning angles.

To adjust snap while dimensioning:

1. **Dimensioning Toolbar** button > click a dimensioning style button.
2. If needed, right-click and click a snap mode.

   Restriction: Snap mode is unavailable for Auto and Arc type dimensioning styles. These styles support the auto-snap mode only because they apply to selected line segments and arcs, not selected points.
3. Select the first dimensioning point.
4. If needed, right-click and click a snap mode.
5. Select the second dimensioning point.
6. If you selected the Angular dimensioning style, repeat steps 4-5 as needed to select the other side of the angle.

Snap Modes

The following table lists the snap modes and their descriptions.

Table 38-2. Snap Modes

Snap to:	Description
Corner	Snaps to the endpoints of a line segment or arc. A corner is where a design object changes direction.
Midpoint	Snaps to the center point of a line segment or arc. Circles are selected at the 180-degree point, left side, since arcs are drawn with the starting point on the right.
Any Point	Snaps to the closest point on a line segment, arc, or circle.
Center	Snaps to the center of the closest circle, arc, or pad.

Table 38-2. Snap Modes (cont.)

Snap to:	Description
Circle/Arc	Snaps to the radius point of an arc or circle.
Intersection	Snaps to closest point where two or more objects intersect.
Quadrant	Snaps to the closest orthogonal point, 0, 90, 180, or 270 degrees, on an arc or circle. Arcs must pass through an orthogonal point to be selected.
Do Not Snap	Snaps to any grid point.

Related Topics

Dimensioning Process

Selecting the Parent Dimensioning Object

To select the parent dimension object from a selected dimension element:

1. Select one of the items in the dimension object.
2. Right-click and click **Select Parent**. The entire dimension is selected.

Related Topics

Dimensioning Modes in the *Concepts Guide*

Moving Dimensions and Dimension Objects

You can move Dimensions or Dimension Objects using the Move command or by Dynamically Dragging Objects.

In this topic:

- Moving an Entire Dimension
- Moving a Dimension Object
- Moving Text to Its Default Location
- Dynamically Drag Objects
- Changing Lengths

Moving an Entire Dimension

To move the entire dimension:

1. Select the dimension object through multiple selection or by using the Select Parent command.
2. Right-click and click **Move**. The dimension object dynamically attaches to the pointer.
3. Click to indicate a location for the object.

The dimension object remains selected for additional moving, if required.

To move dimension objects using keyboard entered coordinates, use the Dimension Object Properties dialog box.

Moving a Dimension Object

You can move individual dimension objects, including: arrows (with their lines), text strings, extension lines, and leader segments.

To move an dimension object:

1. Select a dimension object.
2. Right-click and click **Move**.
3. Click to indicate the new location for the dimension object.

Tips:

- The new position of the text and the location of the arrows are determined by the current Option settings.
- If you select and move an extension line, the dimension line and the dimension text are both automatically modified to reflect the new measurement. Stretching the extension line without changing the measurement is performed with the Change Length command. This command works in the same manner as moving arrows, but works with selected extension lines.

Moving Text to Its Default Location

1. Click the dimensioning text to select it. If you cannot select the text, check the selection filter settings.
2. Right-click and click **Default Location**.

Tip: To add a new segment to an existing leader, right-click and click **Add Corner and Split**.

Dynamically Drag Objects

1. Select an object.
2. Place the pointer over the selected object, then click and hold the left mouse button to initiate Drag mode.
3. Move the mouse to the new location for the object and release the left mouse button.

Tip: The above sequence assumes that the **Drag** option is set in the Global tab of the Tools > Options dialog box to **Drag and Drop**.

Changing Lengths

With an extension line selected, the Change Length command adjusts the position of the dimension line and arrows without changing the actual measurement. When used, the dimension object is attached to, and moves along with, the pointer, such as when a new dimension is created.

1. Select the extension line for which you want to change the length.
2. Right-click and click **Change Length**. The dimension object dynamically attaches to the pointer.
3. Click to indicate the new location for the dimension.

Related Topics

Dimensioning Process

Resetting Dimension Measurements

Selecting the Parent Dimensioning Object

Deleting Dimensions

To delete a dimension element:

1. Select the dimension element to delete using one of the following methods:
    - Select one of the dimension's elements. Right-click and click Select Parent.
    - Use multiple selection to sequentially select each element belonging to the dimension.
    - Select the entire dimension with a selection rectangle.
2. Press **Delete**, or click **Delete** from the **Edit** menu.

Resetting Dimension Measurements

To reset the measurement text:

1. Click the dimensioning text to select it. If you cannot select the text, check the selection filter settings.
2. Right-click and click **Reset Measurement**.
3. Type the new measurement in the Text Value dialog box and click **OK**.

Specifying Missing Heights During Export to IDF

You use the Missing Height dialog box to specify missing heights when exporting the design to IDF. The Missing Height dialog box appears when the Geometry.Height attribute does not exist or is set to zero height for any part type and decal pair exported to IDF.

To specify a missing height during export to IDF:

1. In the Height box, type the package and mounting height for the part type and decal pair.

   Tip: If you specify the height as zero, the mechanical design system may prompt you to enter a height when you import the IDF file.
2. If you want to apply the height for all part type and decal pairs, select the For All Parts check box.
3. Click OK.

Result: Export to IDF resumes.

Related Topics

Missing Height Dialog Box

Exporting Part Height Information to IDF Files

Setting Up the CAM350 Link

Use the CAM350 dialog box to set options for, and to generate, a .cam output file.

1. **Tools** menu > **CAM350**.
2. In the Mode area, select from:
   - Create Files Only to simply produce the .cam file
     or
   - Create File and Launch CAM350 to start CAM 350 and load the resulting .cam file.

3. In the Layer Options area, select an option for the amount of PADS design detail to translate to the CAM350 database. Select from:

    - **CAM Documents (includes Nets and Parts)**—Translates CAM documents with part and net intelligence. Plot orientation, as specified in the CAM document, is not applied. This is useful for verifying net lists and DRCs in CAM350.

      Requirement: Before using this option, flood all copper pours and connect all split/mixed plane layers. Also, on the Tools > Options > Split/Mixed Plane tab, in the Save to PCB File area, click All Plane Data. This allows all pour data to be passed to CAM350.

    - **CAM Documents (Graphics only, Orientations applied)**—Translates CAM documents with plot orientation, including offset, rotation, mirroring, and scaling. This is useful for direct Gerber file translation to CAM350 where no component or net information is required.

      Requirement: Before using this option, if your PADS Layout photoplot output format is set to RS-274D, make sure your CAM350 default units (mils or inches) and CAM350 default precision are set to match the units for PADS Layout and the precision for PADS Layout File/CAM photoplot output, respectively.

      CAM350 Link uses the files pads3.arl (for English units) and pads3m.arl (for Metric units) with this option and when Gerber files are in RS274-D format. The *.arl files determine how the aperture report file (*.rep) is generated by CAM in PADS Layout will be interpreted by CAM350.

    - **CAD Layers -> CAM Layers (As-Is)**—Translates no CAM documents. Translates CAD layers as defined in the PADS Layout database. This is useful for legacy CAM processes that take advantage of CAD layer translation using the PADS-ASCII format import operation in CAM350.

      Requirement: Before using this option, flood all copper pours and connect all split/mixed plane layers. Also, on the Tools > Options > Split/Mixed Plane tab, in the Save to PCB File area, click All Plane Data. This allows all pour data to be passed to CAM350.

4. Arcs in polygon shapes, such as copper pour, are approximated using straight edges. In the Arc Approximation Tolerance box, type the minimum allowable distance between the actual arc path and the approximated straight-line segments. This appears in the appropriate design units, which you set on the Global tab.

5. In the CAM350 File Name box, type a .cam path and file name or click Browse to navigate to a location. The default file name and path are the same name as the current design file name and path.

Related Topics

Annotating DFF Errors

Chapter 39
CAM Output

Creating CAM Outputs to Manufacture Your PCB

This topic lists the typical gerber files, drill files, assembly coordinate files, and drawings you need to manufacture your PCB. Check with your manufacturer for a specific list of requirements. Click the links for the procedures to create each file type.

Procedure

For the Top Layer:

1. Create the Conductive Elements gerber-format file for the photo plotter. Choose between the following layer types:
    - Routing/Split Plane (gerber-format file)
    - CAM Plane (gerber-format file)
2. Create the Silkscreen gerber-format file for the photo plotter (gerber-format file)
3. Create the Solder mask gerber-format file for the photo plotter (gerber-format file)
4. Create the Paste mask gerber-format file for the photo plotter (gerber-format file)
5. Create the Assembly Drawing (non-gerber)

For Each Internal Layer:

- Create the Conductive Elements gerber-format file for the photo plotter. Choose between the following layer types:
    - Routing/Split Plane (gerber-format file)
    - CAM Plane (gerber-format file)

For the Bottom Layer:

1. Create the Conductive Elements gerber-format file for the photo plotter. Choose between the following layer types:
    - Routing/Split Plane (gerber-format file)
    - CAM Plane (gerber-format file)
2. Create the Silkscreen gerber-format file for the photo plotter (gerber-format file)

3. Create the Solder mask gerber-format file for a photo plotter (gerber-format file)
4. Create the Paste mask gerber-format file for a photo plotter (gerber-format file)
5. Create the Assembly Drawing (non-gerber)

For Each Drill Pair

1. Create the Drill Drawing with Drill Table (non-gerber)
2. Create the NC Drill file for the drilling machine (drill file)

Related Topics

Verifying a Gerber File

Creating a Custom CAM Output

You can create a custom CAM output. This setting has no presets. You start with no default assigned layers or objects and choose which items to include in the output.

Procedure

1. On the **File** menu, click **CAM**.
2. In the Define CAM Documents dialog box, click **Add**.
3. In the Add Document dialog box, type a Document Name.

   Tip: It's beneficial to add a description of your custom output in the document name.
4. In the Document Type list, click **Custom.**
5. In the Output File box, type a name for the file you are creating.

   Tip: An autogenerated name appears in the Output File box. This is the name of the file you might send to the manufacturer. Your manufacturer will find it helpful if this filename is related to its function and placement in the layer stackup.
6. In the Customize Document area, click the **Layers** button.
7. In the Select Items dialog box, choose which layers and layer items should appear in your output.

   Tip: Click the Preview button to check what will be included in the output.
8. Click **OK** to accept the changes and close the Select Items dialog box.
9. In the Customize Document area, click **Options**.
10. In the Plot Options dialog box, in the Positioning area, set the positioning options.
11. Click **OK** to close the Plot Options dialog box.

12. In the Output Device area, choose the device and update the Device Setup as needed.
13. Click **OK** to add the new file configuration to the Define CAM Documents dialog box.

    Tip: While you can click Run and create the output file from the Add Document dialog box, you probably want to add this configuration to the Define CAM Documents dialog box in case you need to generate the file again.

14. In the CAM Directory list, choose the folder where you want to save the output files. You can skip this step if you are sending it to your printer.
15. Select the document(s) you want to output in the Document name list, and then click **Run**.
16. Click **Save**.

    Tip: If you close the design without saving the design, you will lose any additions to the Define CAM Documents dialog box.

Results

Your file(s) appear in the C:\PADS Projects\Cam directory (or a subdirectory if you created one) if you are not sending the output to your printer.

Creating a Silkscreen Gerber-format File

You create the silkscreen gerber-format file for your manufacturer's photo plotter to produce the silkscreen top, or silkscreen bottom layer.

Procedure

1. On the **File** menu, click **CAM**.
2. In the Define CAM Documents dialog box, click **Add**.
3. In the Add Document dialog box, type a Document Name such as Silkscreen Top, or Silkscreen Bottom
4. In the Document Type list, click **Silkscreen** and in the Layer Association dialog box that appears, choose from the Top or Bottom side of the board.

    Tip: The layers may not be named Top or Bottom, but will have the names you chose for the top and bottom layer of your design in the Layers Setup dialog box**.**

5. In the Output File box, type a name for the file you are creating.

    Tip: An autogenerated name appears in the Output File box. This is the name of the file you will send to the manufacturer. Your manufacturer will find it helpful if this filename is related to its function. Name the file silkscreen_top.pho or silkscreen_bottom.pho.

6. In the Customize Document area, click the **Layers** button.

7. In the Select Items dialog box, choose which layers and layer items should appear in the silkscreen data.

   Tips:

   - If all your silkscreen elements have been added to the Silkscreen layer, you can remove other layers in the Selected list.
   - In a typical silkscreen, you want the Component outlines of the layer you are working on (Top Mounted or Bottom Mounted), as well as Ref. Des., and Outlines.
   - For large components, it's common to add attributes that add dots at pin 1 and numbers around the component to lessen the time it takes to locate a pin. If you've added such attributes on the Silkscreen layer, you should choose to display the Attributes from the Silkscreen layer.
   - Click the Preview button to check what will be included in the output.

8. Click **OK** to accept the changes and close the Select Items dialog box.
9. In the Customize Document area, click **Options**.
10. In the Plot Options dialog box, in the Positioning area, set the positioning options.

    Tip: It's important that each gerber-format file, that you output for your design, line up exactly with the others. You must use the same Positioning settings for each layer and you must ensure that objects on one layer are not offsetting the layer from the Justification setting. For example, you've chosen to justify the layers by the Top Left. And you've decided to include the board outline in all your outputs. The board outline will be justified at the top left of each output. But if you have a component that extends outside the board outline on the left side of the board, it's possible that your silkscreen layer will not be aligned to the other layers you've created, since the alignment is to the component outline on the left (and the board outline at the top).

11. In the Suppress area, you can suppress the display of some Reference Designators.

    Tip: For example, if you have a set of mounting holes that use the Ref Des prefix X, you can choose not to include those reference designators in the silkscreen.

12. Click **OK** to close the Plot Options dialog box.
13. In the Output Device area, click the **Photo button if not already selected**.
14. Click **Device Setup**. In the Photo Plotter Setup dialog box, make any necessary changes to the settings.
15. Click the **Advanced** button. In the Photo Plotter Advanced Setup dialog box, make any necessary changes to the settings.

    Tip: You may want to use the RS-274X format.

16. Click **OK** to close the Photo Plotter Advanced Setup dialog box.

17. Click **OK** to close the Photo Plotter Setup dialog box.
18. Click **OK** to add the new file configuration to the Define CAM Documents dialog box.

    Tip: While you can click Run and create the output file from the Add Document dialog box, you probably want to add this configuration to the Define CAM Documents dialog box in case you need to generate the file again.
19. Repeat these steps again to create the Silkscreen output for the opposite side of the board, if applicable.
20. In the CAM Directory list, choose the folder where you want to save the output files.
21. Select the document(s) you want to output in the Document name list, and then click **Run**.
22. Click **Save**.

    Tip: If you close the design without saving the design, you will lose any additions to the Define CAM Documents dialog box.

Results

Your file(s) appear in the C:\PADS Projects\Cam directory or a subdirectory if you created one.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Interpreting the Silkscreen Preview

When looking at the Silkscreen preview, the black lines represent the silkscreen artwork. You should typically see component outlines of the layer you are working on (Top Mounted or Bottom Mounted), any board identification or version text, company logo, as well as Reference Designators outside the component outlines renumbered (from the original netlist import) into an easily searchable pattern.
(You use two sets of reference designators, one set on the silkscreen layers for the silkscreen artwork positioned outside the components where they won't be hidden after the component are placed and soldered onto the board. And the other set on the assembly drawing layers centered on components for the assembly drawing. If you haven't built this second assembly drawing set of reference designators into your library decals, you can quickly add them to your design. See Generating a Second Set of Reference Designators for Assembly Drawings.)

Related Topics

Creating a Silkscreen Gerber-format File

Changing the Reference Designators of Multiple Components in ECO Mode (Autorenumbering)

Creating a Solder Mask Gerber-format File

You create the solder mask gerber-format file for your manufacturer's photo plotter to produce the solder mask top, or solder mask bottom layer.

Procedure

1. On the **File** menu, click **CAM**.
2. In the Define CAM Documents dialog box, click **Add**.
3. In the Add Document dialog box, type a Document Name such as Solder Mask Top, or Solder Mask Bottom
4. In the Document Type list, click **Solder Mask** and in the Layer Association dialog box that appears, choose from the Top or Bottom side of the board.

   Tip: The layers may not be named Top or Bottom, but will have the names you chose for the top and bottom layer of your design in the Layers Setup dialog box**.**
5. In the Output File box, type a name for the file you are creating.

   Tip: An autogenerated name appears in the Output File box. This is the name of the file you will send to the manufacturer. Your manufacturer will find it helpful if this filename is related to its function. Name the file soldermask_top.pho or soldermask_bottom.pho.
6. In the Customize Document area, click the **Layers** button.
7. In the Select Items dialog box, choose which layers and layer items should appear in the solder mask data.

   Tips:

   - You need solder mask openings for each pad of each component and perhaps each via too.
     - If you've added these openings in the pad stacks, and all your solder mask elements have been added to the Solder Mask layer, you can remove other layers in the Selected list.
     - If you haven't added these openings in the pad stacks, there is an easy way to get the openings that you need without editing all the pad stacks. You can add the top layer to the Selected list and select the Pads check box for that layer. This adds openings the exact size of the pads in the solder mask. You'll only need to oversize the Pads in the Plot Options.
   - Click the Preview button to check what will be included in the output. When looking at the preview, the white area will be the solder mask. Black areas are openings in the solder mask.
8. Click **OK** to accept the changes and close the Select Items dialog box.

9. In the Customize Document area, click **Options**.
10. In the Plot Options dialog box, in the Positioning area, set the positioning options.

    Tip: It's important that each gerber-format file, that you output for your design, line up exactly with the others. You must use the same Positioning settings for each layer and you must ensure that objects on one layer are not offsetting the layer from the Justification setting. For example, you've chosen to justify the layers by the Top Left. And you've decided to include the board outline in all your outputs. The board outline will be justified at the top left of each output. But if you have a component that extends outside the board outline on the left side of the board, it's possible that your silkscreen layer will not be aligned to the other layers you've created, since the alignment is to the component outline on the left (and the board outline at the top).

11. In the Over(Under)size Pads By box, type a value if you need to globally modify the pad size openings.

    Tip: If you're using the pad shapes of an outer layer to create the solder mask openings of the solder mask layer, you'll probably want to add a global oversize here.

12. Click **OK** to close the Plot Options dialog box.
13. In the Output Device area, click the **Photo button if not already selected**.
14. Click **Device Setup**. In the Photo Plotter Setup dialog box, make any necessary changes to the settings.
15. Click the **Advanced** button. In the Photo Plotter Advanced Setup dialog box, make any necessary changes to the settings.

    Tip: You may want to use the RS-274X format.

16. Click **OK** to close the Photo Plotter Advanced Setup dialog box.
17. Click **OK** to close the Photo Plotter Setup dialog box.
18. Click **OK** to add the new file configuration to the Define CAM Documents dialog box.

    Tip: While you can click Run and create the output file from the Add Document dialog box, you probably want to add this configuration to the Define CAM Documents dialog box in case you need to generate the file again.

19. Repeat these steps again to create the Solder Mask output for the opposite side of the board, if applicable.
20. In the CAM Directory list, choose the folder where you want to save the output files.
21. Select the document(s) you want to output in the Document name list, and then click **Run**.
22. Click **Save**.

Tip: If you close the design without saving the design, you will lose any additions to the Define CAM Documents dialog box.

Results

Your file(s) appear in the C:\PADS Projects\Cam directory or a subdirectory if you created one.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Control of Solder Mask and Paste Mask

Interpreting the Solder Mask Preview

When looking at the Solder Mask preview, the white area will be the solder mask. Black areas are openings in the solder mask. You should typically see solder mask openings for each pad of each component and any vias that are not tented (masked).

Related Topics

Creating a Solder Mask Gerber-format File

Creating a Paste Mask Gerber-format File

You create the paste mask gerber-format file for your manufacturer's photo plotter to produce the paste mask top, or paste mask bottom layer.

Procedure

1. On the **File** menu, click **CAM**.
2. In the Define CAM Documents dialog box, click **Add**.
3. In the Add Document dialog box, type a Document Name such as Paste Mask Top, or Paste Mask Bottom
4. In the Document Type list, click **Paste Mask** and in the Layer Association dialog box that appears, choose from the Top or Bottom side of the board.

   Tip: The layers may not be named Top or Bottom, but will have the names you chose for the top and bottom layer of your design in the Layers Setup dialog box**.**
5. In the Output File box, type a name for the file you are creating.

   Tip: An autogenerated name appears in the Output File box. This is the name of the file you will send to the manufacturer. Your manufacturer will find it helpful if this filename is related to its function. Name the file pastemask_top.pho or pastemask_bottom.pho.

6. In the Customize Document area, click the **Layers** button.
7. In the Select Items dialog box, choose which layers and layer items should appear in the paste mask data.

   Tips:

   - You need paste mask openings for each pad of each surface mount component.
     - If you've added these openings in the pad stacks, and all your paste mask elements have been added to the Paste Mask layer, you can remove other layers in the Selected list.
     - If you haven't added these openings in the pad stacks, there is an easy way to get the openings that you need without editing all the pad stacks. You can add the top layer to the Selected list and select the Pads check box for that layer. This adds openings the exact size of the pads in the paste mask. You can oversize or undersize the Pads (paste mask openings) in the Plot Options.
   - If all your paste mask elements have been added to the Paste Mask Top layer, you can remove other layers in the Selected list.
   - Click the Preview button to check what will be included in the output. When looking at the preview, the black areas are the paste mask openings which become the solder paste locations on the board.
8. Click **OK** to accept the changes and close the Select Items dialog box.
9. In the Customize Document area, click **Options**.
10. In the Plot Options dialog box, in the Positioning area, set the positioning options.

    Tip: It's important that each gerber-format file, that you output for your design, line up exactly with the others. You must use the same Positioning settings for each layer and you must ensure that objects on one layer are not offsetting the layer from the Justification setting. For example, you've chosen to justify the layers by the Top Left. And you've decided to include the board outline in all your outputs. The board outline will be justified at the top left of each output. But if you have a component that extends outside the board outline on the left side of the board, it's possible that your silkscreen layer will not be aligned to the other layers you've created, since the alignment is to the component outline on the left (and the board outline at the top).
11. In the Suppress area, you can suppress the paste mask openings for components.

    Tip: For example, if you have an edge connector, you don't want to add paste mask openings over its pads. If you're using the pad shapes of an outer layer to create the paste mask openings of the paste mask layer, you'll want to add the reference designator of the edge connector to the Suppress box to prevent it from getting solder paste in the manufacturing process.

12. In the Over(Under)size Pads By box, type a value if you need to globally modify the pad size openings.
13. Click **OK** to close the Plot Options dialog box.
14. In the Output Device area, click the **Photo button if not already selected**.
15. Click **Device Setup**. In the Photo Plotter Setup dialog box, make any necessary changes to the settings.
16. Click the **Advanced** button. In the Photo Plotter Advanced Setup dialog box, make any necessary changes to the settings.

    Tip: You may want to use the RS-274X format.
17. Click **OK** to close the Photo Plotter Advanced Setup dialog box.
18. Click **OK** to close the Photo Plotter Setup dialog box.
19. Click **OK** to add the new file configuration to the Define CAM Documents dialog box.

    Tip: While you can click Run and create the output file from the Add Document dialog box, you probably want to add this configuration to the Define CAM Documents dialog box in case you need to generate the file again.
20. Repeat these steps again to create the Paste Mask output for the opposite side of the board, if applicable.
21. In the CAM Directory list, choose the folder where you want to save the output files.
22. Select the document(s) you want to output in the Document name list, and then click **Run**.
23. Click **Save**.

    Tip: If you close the design without saving the design, you will lose any additions to the Define CAM Documents dialog box.

Results

Your file(s) appear in the C:\PADS Projects\Cam directory or a subdirectory if you created one.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Control of Solder Mask and Paste Mask

Interpreting the Paste Mask Preview

When looking at the Paste Mask preview keep in mind that, unlike solder mask, the paste mask is not an artwork layer - it is not a substance that is applied to the board. In this respect it is opposite to the solder mask layer. The black areas are in fact openings in the paste mask layer and are the locations where the solder paste will be applied to the board. You should typically see paste mask openings (solder paste locations) for each pad of surface mounted components.

Related Topics

Creating a Paste Mask Gerber-format File

Creating an Assembly Drawing

You create the assembly drawing for your manufacturer to reference when assembling the components on the board.

Procedure

1. On the **File** menu, click **CAM**.
2. In the Define CAM Documents dialog box, click **Add**.
3. In the Add Document dialog box, type a Document Name such as Assembly Top, or Assembly Bottom
4. In the Document Type list, click **Assembly** and in the Layer Association dialog box that appears, choose from the Top or Bottom side of the board.

   Tip: The layers may not be named Top or Bottom, but will have the names you chose for the top and bottom layer of your design in the Layers Setup dialog box**.**
5. In the Output File box, type a name for the file you are creating.

   Tip: An autogenerated name appears in the Output File box. This is the name of the file you will send to the manufacturer. Your manufacturer will find it helpful if this filename is related to its function. Name the file assembly_top.pho or assembly_bottom.pho.
6. In the Output Device area, choose the device and update the Device Setup as needed. Typically, you would send the Assembly drawings to your printer or plotter and not output them to a gerber file.

   Tip: When you activate a printer or plotter as your output, you have the option to apply different colors to items in your output document. For example, you could include traces and pads in your assembly drawing using a lighter gray color while your component outlines and reference designators remained black. This could provide more context when manually assembling the board.
7. In the Customize Document area, click the **Layers** button.
8. In the Select Items dialog box, choose which layers and layer items should appear in the paste mask data.

   Tips:

- You can add a set of reference designators to the assembly layers, and center those reference designators inside the component outline instead of using the silkscreen reference designators which will be scattered around the outside of components. This will make the assembly drawing easier to read. For the procedures, see Generating a Second Set of Reference Designators for Assembly Drawings.
- You may want to add the Part Types to the assembly drawing also.
- If you've added markers in attributes to mark pin 1, it might be beneficial to add these to your assembly drawing.
- If all your assembly elements have been added to the assembly layer, you can remove other layers in the Selected list.
- Click the Preview button to check what will be included in the output.

9. Click **OK** to accept the changes and close the Select Items dialog box.
10. In the Customize Document area, click **Options**.
11. In the Plot Options dialog box, in the Positioning area, set the positioning options.
12. In the Suppress area, you can suppress the display of some Reference Designators.

    Tip: For example, if you have a set of mounting holes that use the Ref Des prefix X, you can choose not to include those reference designators in the assembly. They're not a true component and you don't need the assemblers to think it needs to be populated with a component.
13. For the assembly bottom drawing, you may want to select the Mirror Image check box to mirror the image.
14. Click **OK** to close the Plot Options dialog box.
15. Click **OK** to add the new file configuration to the Define CAM Documents dialog box.

    Tip: While you can click Run and create the output file from the Add Document dialog box, you probably want to add this configuration to the Define CAM Documents dialog box in case you need to generate the file again.
16. Repeat these steps again to create the Assembly drawing for the opposite side of the board, if applicable.
17. In the CAM Directory list, choose the folder where you want to save the output files. You can skip this step if you are sending it to your printer.
18. Select the document(s) you want to output in the Document name list, and then click **Run**.
19. Click **Save**.

    Tip: If you close the design without saving the design, you will lose any additions to the Define CAM Documents dialog box.

Results

Your file(s) appear in the C:\PADS Projects\Cam directory (or a subdirectory if you created one) if you are not sending the output to your printer.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Interpreting the Assembly Drawing Preview

When looking at the Assembly Drawing preview, you typically see component outlines of the layer you are working on (Top Mounted or Bottom Mounted), as well as Reference Designators centered inside the component outlines renumbered (from the original netlist import) into an easily searchable pattern.
(You use two sets of reference designators, one set on the assembly drawing layers centered on components for the assembly drawing, and the other set on the silkscreen layers for the silkscreen positioned outside the components where they won't be hidden after the component placement. If you haven't built this second assembly drawing set of reference designators into your library decals, you can quickly add them to your design. See Generating a Second Set of Reference Designators for Assembly Drawings.)

Related Topics

Creating an Assembly Drawing

Changing the Reference Designators of Multiple Components in ECO Mode (Autorenumbering)

Creating a Routing/Split Plane Gerber-format File

Conductive element layers are separated into two types of outputs. Your layers are either routing/split plane layers, or CAM plane layers. You create the routing/split plane gerber-format file for your manufacturer's photo plotter to produce the conductive layer of your PCB.

Restriction

Before you actually create the gerber-format file for this conductive layer, you must ensure that all copper pour or plane areas are flooded and filled with copper.

Procedure

1. On the **File** menu, click **CAM**.
2. In the Define CAM Documents dialog box, click **Add**.
3. In the Add Document dialog box, type a Document Name.

Tip: It's beneficial to add not only the usage of the layer in the name, but also the placement of the layer in the board layer stackup. For example, Signal 2 - L6.

4. In the Document Type list, click **Routing/Split Plane** and in the Layer Association dialog box that appears, choose your layer from the available layers.

5. In the Output File box, type a name for the file you are creating.

   Tip: An autogenerated name appears in the Output File box. This is the name of the file you will send to the manufacturer. Your manufacturer will find it helpful if this filename is related to its function and placement in the layer stackup.

6. In the Customize Document area, click the **Layers** button.

7. In the Select Items dialog box, choose which layers and layer items should appear in the silkscreen data.

   Tips:

   - In a typical conductive elements output, you want all the conductive elements of one layer. You probably want Pads, Traces, Vias, Copper, and associated pin copper if it exists.
   - If you add drafting elements to this layer, like lines or text, they will be manufactured as copper. You might want to move these items to the silkscreen layer.
   - Click the Preview button to check what will be included in the output.

8. Click **OK** to accept the changes and close the Select Items dialog box.

9. In the Customize Document area, click **Options**.

10. In the Plot Options dialog box, in the Positioning area, set the positioning options.

    Tip: It's important that each gerber-format file, that you output for your design, line up exactly with the others. You must use the same Positioning settings for each layer and you must ensure that objects on one layer are not offsetting the layer from the Justification setting. For example, you've chosen to justify the layers by the Top Left. And you've decided to include the board outline in all your outputs. The board outline will be justified at the top left of each output. But if you have a component that extends outside the board outline on the left side of the board, it's possible that your silkscreen layer will not be aligned to the other layers you've created, since the alignment is to the component outline on the left (and the board outline at the top).

11. Click **OK** to close the Plot Options dialog box.

12. In the Output Device area, click the **Photo button if not already selected**.

13. Click **Device Setup**. In the Photo Plotter Setup dialog box, make any necessary changes to the settings.

14. Click the **Advanced** button. In the Photo Plotter Advanced Setup dialog box, make any necessary changes to the settings.

    Tip: You may want to use the RS-274X format.

15. Click **OK** to close the Photo Plotter Advanced Setup dialog box.

16. Click **OK** to close the Photo Plotter Setup dialog box.

17. Click **OK** to add the new file configuration to the Define CAM Documents dialog box.

    Tip: While you can click Run and create the output file from the Add Document dialog box, you probably want to add this configuration to the Define CAM Documents dialog box in case you need to generate the file again.

18. In the CAM Directory list, choose the folder where you want to save the output files.

19. Select the document(s) you want to output in the Document name list, and then click **Run**.

    Restriction: If your design has copper pour or plane areas that are not filled with copper, you must fill the areas before creating the gerber-format outputs. You will be prompted to fill them in order to create the output file.

20. Click **Save**.

    Tip: If you close the design without saving the design, you will lose any additions to the Define CAM Documents dialog box.

Results

Your file(s) appear in the C:\PADS Projects\Cam directory or a subdirectory if you created one.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Interpreting the Routing/Split Plane Preview

When looking at the Routing/Split Plane preview, you typically see in black - only the copper items located on the selected layer - all traces, vias, copper, copper pour areas, plane areas. If this is a component layer, you'll also see all the component pads and associated pin copper if it exists. The white areas are areas of no copper.

Related Topics

Creating a Routing/Split Plane Gerber-format File

Creating a CAM Plane Gerber-format File

Conductive element layers are separated into two types of outputs. Your layers are either routing/split plane layers, or CAM plane layers. You create the CAM plane gerber-format file for your manufacturer's photo plotter to produce the conductive layer of your PCB.

Procedure

1. On the **File** menu, click **CAM**.
2. In the Define CAM Documents dialog box, click **Add**.
3. In the Add Document dialog box, type a Document Name.

   Tip: It's beneficial to add not only the usage of the layer in the name, but also the placement of the layer in the board layer stackup. For example, GND CAM Plane - L3.
4. In the Document Type list, click **CAM Plane** and in the Layer Association dialog box that appears, choose your layer from the available layers.
5. In the Output File box, type a name for the file you are creating.

   Tip: An autogenerated name appears in the Output File box. This is the name of the file you will send to the manufacturer. Your manufacturer will find it helpful if this filename is related to its function and placement in the layer stackup.
6. In the Customize Document area, click the **Layers** button.
7. In the Select Items dialog box, choose which layers and layer items should appear in the silkscreen data.

   Tips:

   - In a typical CAM plane output, you probably want Pads, Vias, and Copper. Since this is a negative image plane, copper objects will show up as a cut out in the CAM Plane.
   - If you add drafting elements to this layer, like lines or text, they will be manufactured as copper. You might want to move these items to the silkscreen layer.
   - Click the Preview button to check what will be included in the output.

     Tip: CAM Planes are negative images in the work area and also in the CAM Preview. When viewing a routing/split plane layer document type the white areas are openings in the copper. But when viewing a CAM plane, the white area is copper.
8. Click **OK** to accept the changes and close the Select Items dialog box.
9. In the Customize Document area, click **Options**.
10. In the Plot Options dialog box, in the Positioning area, set the positioning options.

Tip: It's important that each gerber-format file, that you output for your design, line up exactly with the others. You must use the same Positioning settings for each layer and you must ensure that objects on one layer are not offsetting the layer from the Justification setting. For example, you've chosen to justify the layers by the Top Left. And you've decided to include the board outline in all your outputs. The board outline will be justified at the top left of each output. But if you have a component that extends outside the board outline on the left side of the board, it's possible that your silkscreen layer will not be aligned to the other layers you've created, since the alignment is to the component outline on the left (and the board outline at the top).

11. In the CAM Plane Layers area, set the settings as necessary.
12. Click **OK** to close the Plot Options dialog box.
13. In the Output Device area, click the **Photo button if not already selected**.
14. Click **Device Setup**. In the Photo Plotter Setup dialog box, make any necessary changes to the settings.
15. Click the **Advanced** button. In the Photo Plotter Advanced Setup dialog box, make any necessary changes to the settings.

    Tip: You may want to use the RS-274X format.
16. Click **OK** to close the Photo Plotter Advanced Setup dialog box.
17. Click **OK** to close the Photo Plotter Setup dialog box.
18. Click **OK** to add the new file configuration to the Define CAM Documents dialog box.

    Tip: While you can click Run and create the output file from the Add Document dialog box, you probably want to add this configuration to the Define CAM Documents dialog box in case you need to generate the file again.
19. In the CAM Directory list, choose the folder where you want to save the output files.
20. Select the document(s) you want to output in the Document name list, and then click **Run**.
21. Click **Save**.

    Tip: If you close the design without saving the design, you will lose any additions to the Define CAM Documents dialog box.

Results

Your file(s) appear in the C:\PADS Projects\Cam directory or a subdirectory if you created one.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Interpreting the CAM Plane Preview

When looking at the CAM Plane preview, you see a negative image (opposite image) compared to the Routing/Split Plane layers. The white area is the copper and the black objects - typically only vias or through hole pin locations are areas of no copper. Since this is a negative image, you place copper on this layer to create a copper void (an area of no copper). You should see these voids if you've used them.

Related Topics

Creating a CAM Plane Gerber-format File

Creating an NC Drill File

You can create an NC drill file for each drill pair in your design.

Procedure

1. On the **File** menu, click **CAM**.
2. In the Define CAM Documents dialog box, click **Add**.
3. In the Add Document dialog box, type a Document Name.

   Tip: It's beneficial to add not only the usage of the file in the name, but also the drill pair if using partial vias. For example, NC Drill L1-3.
4. In the Document Type list, click **NC Drill**.
5. In the Output File box, type a name for the file you are creating.

   Tip: An autogenerated name appears in the Output File box. This is the name of the file you will send to the manufacturer. Your manufacturer will find it helpful if this filename is related to its function and the drill pair it represents.
6. In the Customize Document area, click **Options**.
7. In the NC Drill Options dialog box, set the options for your file.

   Tip: If you are using partial vias, you can choose which drill pair this file represents by selecting the Partial Vias check box and selecting the Drill Pair.
8. Click **OK** to close the NC Drill Options dialog box.
9. In the Output Device area with the Drill button automatically selected, click **Device Setup**.
10. In the NC Drill Setup dialog box, make any necessary changes to the settings.
11. Click **OK** to close the NC Drill Setup dialog box.
12. Click **OK** to add the new file configuration to the Define CAM Documents dialog box.

Tip: While you can click Run and create the output file from the Add Document dialog box, you probably want to add this configuration to the Define CAM Documents dialog box in case you need to generate the file again.

13. In the CAM Directory list, choose the folder where you want to save the output files.
14. Select the document(s) you want to output in the Document name list, and then click **Run**.
15. Click **Save**.

    Tip: If you close the design without saving the design, you will lose any additions to the Define CAM Documents dialog box.

Results

Your file(s) appear in the C:\PADS Projects\Cam directory or a subdirectory if you created one.

Interpreting the NC Drill File Preview

When looking at the NC Drill File preview, you see drill locations, but this file is not meant to be viewed like the other output files. View the Drill Drawing instead.

Related Topics

Creating an NC Drill File

Creating a Drill Drawing with Drill Table

You can create a drill drawing for one or more drill pairs.

Procedure

1. On the **File** menu, click **CAM**.
2. In the Define CAM Documents dialog box, click **Add**.
3. In the Add Document dialog box, type a Document Name.

   Tip: It's beneficial to add not only the usage of the file in the name, but also the drill pair if using partial vias. For example, Drill Drawing L1-3.

4. In the Document Type list, click **Drill Drawing** and in the Layer Association dialog box that appears, choose one of the layers in your drill pair.

   Tip: If you are only using through hole vias, any layer will do. If you want to display the drill holes of multiple drill pairs in the same drawing, select one of the layers in your drill pair. You will add additional layers in a later step.

5. In the Output File box, type a name for the file you are creating.

Tip: An autogenerated name appears in the Output File box. This is the name of the file you will send to the manufacturer. Your manufacturer will find it helpful if this filename is related to its function and the drill pair it represents.

6. In the Customize Document area, click the **Layers** button.
7. In the Select Items dialog box, choose which layers to include.

   Tips:

   - If you've added dimensions to the Drill Drawing layer, you'll need to enable Lines and Text.
   - For designs that only use through holes, you'll only need one electrical layer. For designs that use partial vias, you'll need to add at least one layer that the partial via starts on, passes through, or ends on. Even though a layer of each drill pair appears in the list, you can use the Drill Drawing Options to restrict the drill holes that appear in the drawing.
   - Click the Preview button to check what will be included in the output.
8. Click **OK** to accept the changes and close the Select Items dialog box.
9. In the Customize Document area, click **Options**.
10. In the Plot Options dialog box, in the Positioning area, set the positioning options.
11. In the Preview window, notice the position of the magenta-colored rectangle in respect to the position of your board outline. This is the current location of the drill chart in the drill drawing. Determine where to position the drill chart with reference to the design origin. You will need to input a new X,Y location in the Drill Drawing Options dialog box to come in a later step.
12. Click the **Drill Symbols** button.
13. In the Drill Drawing Options dialog box, in the Drill Chart area, type new coordinates for the Location of the chart if necessary.
14. If using partial vias, select the **Show Through/Partial column** check box to allow filtering of the drill holes. This allows you to create multiple Drill Drawings and choose which drill holes you want to display. You are given access to check boxes for each drill symbol and you can turn those symbols on or off as needed.
15. You might need to **Regenerate** to update the table data.
16. Click **OK** to close the Drill Drawing Options dialog box.
17. Check your location of the Drill Chart in the Preview window. If needed, click the Drill Symbols button again to correct the location of the drill chart.
18. Click **OK** to close the Plot Options dialog box.
19. In the Output Device area, choose the device and update the Device Setup as needed.

20. Click **OK** to add the new file configuration to the Define CAM Documents dialog box.

    Tip: While you can click Run and create the output file from the Add Document dialog box, you probably want to add this configuration to the Define CAM Documents dialog box in case you need to generate the file again.

21. In the CAM Directory list, choose the folder where you want to save the output files. You can skip this step if you are sending it to your printer.

22. Select the document(s) you want to output in the Document name list, and then click **Run**.

23. Click **Save**.

    Tip: If you close the design without saving the design, you will lose any additions to the Define CAM Documents dialog box.

Results

Your file(s) appear in the C:\PADS Projects\Cam directory (or a subdirectory if you created one) if you are not sending the output to your printer.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Interpreting the Drill Drawing Preview

When looking at the Drill Drawing preview, you typically see drill locations with markers and a drill chart listing all the drill sizes - based on your assignments in the Drill Drawing Options dialog box. Dimensioning measurements for the board are sometimes also displayed on this drawing.

Related Topics

Creating a Drill Drawing with Drill Table

Verifying a Gerber File

Use Verify Photo to open .pho (gerber) files for viewing. The Verify Photo is more than just a Preview, it looks at the actual Gerber output for its data, rather than taking data from the PCB file before the actual photoplot is generated. This allows you to create a hardcopy paper printout of your Gerber files to perform visual inspections and keep records of your plot files.

Procedure

1. On the **File** menu, click **CAM**.

2. In the Define CAM Documents dialog box, click **Add**.
3. In the Document Type list, click **Verify Photo.**
4. **In the** *Photo plotter output file name* **dialog box, browse for the gerber file.**
5. Select the gerber file and click **Open**.

   Result: The path to the file appears in the Summary area.
6. Click the **Preview Selections** button.
7. View your gerber file in the CAM Preview window.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Creating Reusable Fabrication Notes

You can save your fab notes to the library for reuse.

Procedure

1. Create the individual line items of text.
   See also: Adding Free Text
2. Since text alone can't be saved to the library, add an underline to the heading of your fab notes using a 2D Line.
   See also: Creating a Drafting Object
3. Combine all the text with the line item.
   See also: Combining Line and Text Objects.
4. Save your notes to a library.
   See also: Saving a Drafting Item to a Library

Defining CAM Documents

Use the Define CAM Documents dialog box to define and store up to 250 CAM documents.

Tip: You must save the design file before the changes made to the CAM documents become part of the design file.

In this topic:

- CAM Documents Overview
- Adding a CAM Document Workflow

- Adding a CAM Document
- Editing a CAM Document
- Previewing a CAM Document
- Reporting Apertures of a Photo-Plot File(s)
- Deleting a CAM Document
- Reordering the List of CAM Documents
- Creating the Outputs
- Viewing CAM Document settings
- Selecting a Folder for the CAM Output Documents
- Saving the CAM Document Configurations
- Exporting and Importing CAM Document Configurations
- Listing the CAM Documents to File

CAM Documents Overview

CAM is an acronym for Computer-Aided Manufacturing. Using the CAM tools, you can produce not only “Gerber” or manufacturing outputs, but printouts and plots as well.

There is no default set of output type when you start a new design - the Define CAM Documents dialog box is empty. Different types of designs require unique sets of output files. A design with 14 layers requires a larger set of output files than a design with only 2 layers. You must define your own list of output-document configurations which you require for each design. Your configurations can be saved within the .pcb file, so each file has its own CAM Documents list. You can export and then import your document configurations to reuse with a similar design.

When you add a CAM document, you are adding a preset output configuration, like a script, which you run against your design to create the type of output document required. The preset (CAM document) contains the Document Type, visible objects, design positioning, and output device.

When you have a list of document configurations in the Define CAM Documents dialog box, you can quickly run the configurations against your current design and create the outputs, whether they are printouts or photo plots or both. You can print documents singly or in batch mode.

Restriction: You can create a maximum of 250 CAM Document configurations.

Adding a CAM Document Workflow

1. Define the properties of the document. You must set the CAM document name, document type (and fabrication layers if using CAM350 as a post processing tool).

   See also: Adding or Editing CAM Documents

2. Assign layers and set the visibility of items in the document.

   See also: Making Design Objects Visible in CAM Documents

3. Select plot options - set the positioning of the design in the document, the suppression of objects, and CAM Plane Layer options.

4. Assign an output type for the document - to file, to a plotter, or to a printer.

5. Once you have selected the output type, set up the output device.

   See also: Printing, Printing PostScript to a File, Setting Up the Photo Plotter Output

Adding a CAM Document

1. **File** menu > **CAM**.

2. Click the **Add** button to add a CAM Document to the list.

   Result: This opens the Add Document dialog box. Create a CAM Document configuration and return to the Define CAM Documents dialog box to save the configuration and Run the configuration against the design.

Editing a CAM Document

1. **File** menu > **CAM**.

2. Select a document in the CAM Documents list and click the **Edit** button. This opens the Edit Document dialog box where you can edit the document settings.

Previewing a CAM Document

You can preview the results of a CAM Document configuration before you run the configuration against your design.

1. **File** menu > **CAM**.

2. Select a document from the CAM Documents list and click **Preview** to preview the selected document. The CAM Preview dialog box appears.

   See also: Previewing CAM Documents

Reporting Apertures of a Photo-Plot File(s)

You can produce a report of the apertures used in a CAM Document.

Requirement: To produce an aperture report, you must have run the CAM Document(s) photo plot configuration against your design. If the CAM Document is set to print, or pen output, it will not produce the report.

1. **File** menu > **CAM**.
2. Select a CAM document in the CAM Documents list.
3. Click **Aperture Report**.
4. In the Aperture report file name dialog box, type a name for the report and then click **Save**.

   Result: The aperture report opens in the default text editor.

Deleting a CAM Document

1. **File** menu > **CAM**.
2. Select a document in the CAM Documents list and click the **Delete** button.

Reordering the List of CAM Documents

You can change the order of CAM documents in the list if you would like to organize them.

1. Select a document in the CAM Documents list.
2. Click the **Up** or **Down** buttons to move the selected document up or down in the list.

Creating the Outputs

You can run single or multiple CAM document configurations against your current design.

1. Select a document configuration in the CAM Documents list. You can also drag or Shift+click to select multiple adjacent documents. Select multiple non-adjacent documents using Ctrl+click.
2. Click **Run**.

Result: A prompt window verifies the list of documents to be created. The generated CAM outputs are send to print or plot or are written to a file in the CAM Directory.

When you run CAM on a layer that is specified as a mixed plane in the Layers Setup dialog box, the message “Split/mixed plane detected. Perform flood and DRC checks?” appears. This message appears only once when running CAM on multiple split/mixed plane layers.

- Click Yes to run the flood operation on the split/mixed plane, perform a Plane Check of Verify the Design, and generate split/mixed plane data for the layer. The Plane Check is run using the options of the Mixed Plane Setup dialog box.
- Click No to cancel the process. You can run Plane Check from Verify Design manually for more information on the errors.

Tips:

- CAM interprets pads and other objects differently when they are associated with copper in the PCB Decal Editor. **See also:** Working with Associated Copper
- For color print outputs, if text is combined with a 2D line or part of a dimension line, then a grayscale or preset color will be used for line items. Any free text on the board will use what is assigned for text in the Select Items dialog box.

Viewing CAM Document settings

A summary of the settings in any CAM Document configuration can be viewed in the Summary box.

- Select a document in the CAM Documents list and view the settings in the Summary box.

Selecting a Folder for the CAM Output Documents

When you run a CAM Document and the output is a file, it is saved in your CAM directory. By default, CAM directories are located under the path \PADS Projects\Cam\default which is called from the CAMDir entry of powerpcb.ini file.

- To browse for the desired directory and create a new path that will be saved with the design, select <Create> in the CAM directory list.

Saving the CAM Document Configurations

You can save the CAM documents you add to the CAM Documents list. The software will be saved into the current software session but not yet saved into the .pcb file.

Requirement: You must save the .pcb design file for the changes made to the CAM documents to become part of the design file.

- Click the **Save** button.

Exporting and Importing CAM Document Configurations

You can reuse your CAM document configurations with other similar designs by exporting them and importing them into another design.

- Click **Export** to saves the configuration to a separate file. If you name the file default.cam, it is used as the default CAM configuration for each .pcb file.
- Click **Import** to recall an exported configuration to use as the CAM configuration for the .pcb file.

Listing the CAM Documents to File

You can save a list of the CAM documents, which can also be printed and saved with your design documentation.

1. Click **Listing**.
2. You are prompted with the Listing File Name box to provide a name for the listing. Type a name and then click **Save**.

   Result: The listing is saved and the file opens in your default text editor.

Adding or Editing CAM Documents

Use the Add or Edit Document dialog box to define a CAM document.

In this topic:

- Add or Edit a CAM Document
- Selecting an Output Device
- Using TrueLayer Associations

Add or Edit a CAM Document

Define a new CAM Document or edit an existing CAM document using the Add Document/Edit Document dialog box.

Tip: If you use the TrueLayer option, see the Using TrueLayer Associations section for information on how TrueLayer affects layers and CAM documents.

1. **File** menu > **CAM** > **Add** button or select a document and click **Edit**.

   Result: Depending on your action, the Add Document dialog box or the Edit Document dialog box appears. The process for using either dialog box is the same.

2. In the Document Name box, type a document name that describes the plot type and output device; for example, Silkscreen Top Layer - to Print. If you were outputting this document to a file, this is not the filename to which it would be saved; you define that in the Output File box.

3. In the **Document Type** list, select a default plot configuration. When you select a document type, the program automatically generates a stock set of layers and items to plot. You can use these or customize them. You can view a summary of the settings of the default document type in the Summary box.

   Exception: If during your selection of a Document Type you are prompted to select a Layer Association and the layer does not exist in the list, you may need to revisit your Layers Setup. Click Set Layers for a shortcut to the Layers Setup dialog box; otherwise you would need to close all CAM dialog boxes to access the Setup menu.

4. In the Output File box, accept the default name, or type your preferred name for the output file.

5. If you will be using CAM350 for post processing, select the layer that you will use to fabricate the printed circuit board from the **Fabrication Layer** list. The Fabrication Layer is only used during conversion to the CAM350 database using **CAM350 Link**.

6. Click an output device. For plots and drill drawings, select the printer, pen plotter, or photo plotter. The drill device is always selected for NC Drill Output.

   - Click **Device Setup** to open a dialog box that lets you set options associated with the selected device type, Printer Setup, Pen Plot Setup, Photo plot Setup, or NC Drill Setup.

7. If you approve of the default layers and items chosen by PADS Layout, do nothing further. However, you can use the following controls to modify the output further:

   - Click **Layers** to open the Select Items dialog box. You can modify which layers and items appear in this document.

   - Click **Options** to gain access to one of the options dialog boxes listed below, depending on the plot type with which you're working:

Button	**Set up**
Plot Options	Printer, Pen and photo plots
Drill Drawing Options (available from the Plot Options dialog box only when the Drill Drawing document type is selected)	Drill drawings
NC Drill Options	NC drill output

- Click **Assembly** to open the Select Assembly Variant dialog box. You can choose whether to base CAM output on an Assembly Variant and choose the Assembly Variant to use. You cannot edit an Assembly Variant in CAM.

Restriction: The Assembly button is unavailable until you create an assembly variant.

See also: Using the Assembly Variants Dialog Box

8. Click **Preview Selections** to preview Layers and Options settings and check the settings against your expectations.
9. Click **Run** to produce the document you just defined.

   Alternative: Or click OK to load the document configuration into the Define CAM Documents dialog box.
10. If you want to save your current settings as the defaults for this CAM document type and output device, click **Save As Defaults**.

    Tip: If you manually build your own drill table and want to use this drill table as the default drill table, click Save As Defaults.
11. When you finish, click **OK** to accept your changes.

Selecting an Output Device

Basically, you can create either printouts or files to send to the manufacturer. You can print the file to make sure it's okay, and then send the file to the manufacturer. The only difference is the type of output device you select in the Add or Edit Document dialog box.

There are four types of output devices. They're listed as Print, Pen, Photo, and Drill. Easily confused are the two plotter types. One is commonly found in the workplace - a pen plotter; and the other is a photo plotter used by manufacturing. So, you can print or plot your outputs on paper or you can send a photo plot file to manufacturing. There is another type of output - an NC Drill file. This is a file sent to manufacturing like those for the photo plotter, except you would input it into a computerized drilling machine to drill all the vias.

Using TrueLayer Associations

By default, when you flip a component, the component attributes flip with it. For example, you flip a component from one side to the other in the design. Although the reference designator attribute is visible and located on the top silkscreen layer, it automatically flips to the bottom silkscreen layer. You are using TrueLayer association. This feature also depends on you associating layers correctly in the Layer Setup.
You can override the TrueLayer functionality by applying a command line switch to the software on startup.

See also: Start-up Options

Making Design Objects Visible in CAM Documents

When you are adding a CAM document, you can use the Select Items dialog box to define which layers and items should appear in a particular document. You can also define colors for layers and items.

In this topic:

- Selecting Layers and Layer Objects to Plot
- Working with Associated Copper
- Selecting Non Layer-Set Objects to Plot
- Applying Colors to Objects
- Applying Colors to Individual Nets
- Previewing the Settings

Selecting Layers and Layer Objects to Plot

You can enable layers and layer objects to be included in your CAM Document.

1. **File** menu > **CAM**.
2. Click **Add** to create a new CAM document, or select a document from the Document Name list and click **Edit** to edit the selected document.
3. Click **Layers**.
4. Select the layers you want displayed in the Available section.
5. Click **Add** to add the layers to the Selected box.

   Tip: The layers you select are not saved as defaults when you click Save As Defaults on the Add Document dialog box.
6. With the layer selected in the Selected list, select the objects you want in the output. If you don't select any objects, nothing from that layer will be displayed in the CAM document.

Exceptions:

- CAM does not generate output for attribute labels unless you choose to make the labels visible in the design. To display the labels, use the Show list in the Labels Properties dialog box.
- Selecting the **Test Points** check box displays all test points (component pins and vias) on the specified layer. By default, test points are located on the bottom side of the board.

When creating a solder mask document of the top side of your design, only test points marked as Top Access in the Properties dialog box will display in the CAM document. For solder mask documents, test points are added by default. This unmasks test points - specifically test point vias. Typically, vias are covered in solder mask which appears white when previewing the CAM document.

Working with Associated Copper

CAM interprets pads and other objects differently when they are associated with copper in the PCB Decal Editor. Associating copper shapes and open copper is one of the methods used to create hard breakouts in decals. Interpretation is as follows:

- Terminals are interpreted as vias.
- Closed copper shapes are interpreted as pads.
- Open copper (a path drawn with copper) is interpreted as a trace.

Tip: Using the Vias, Pads, and Traces check boxes along with those in the Pins with Associated Copper area will give you total control over what appears in your CAM document.

Additional object visibility options are available in the Pins with Associated Copper area if you have decals that were created with associated copper.

1. In the Pins with Associated Copper section, select the **Advanced Selection** check box to enable the selection of the object check boxes. These options allow for selection of associated copper items independently of regular options for pads, vias and traces.
2. Select the **Pads** check box to display pads with associated copper.
3. Select the **Open Copper** check box to display the open copper that is associated to pins.
4. Select the **Filled Copper** check box to display the closed copper shapes that are associated to pins.

See also: Associating Copper with Terminals

Selecting Non Layer-Set Objects to Plot

You can also enable objects to be included in your CAM document which are not set per layer.

- In the **Other** area, select the check box beside an item to include in your CAM document. Choose from the board outline (and board cut outs), connections, plated slotted holes, and nonplated slotted holes.
- In the **Component Outlines** area, select the check box beside an item to include in your CAM document. Choose from top or bottom mounted component outlines.

Applying Colors to Objects

You can apply colors to CAM document objects if you are outputting to a printer or a pen plotter. The color palette only shows colors that the printer or plotter can output. If your device can only print grayscale, the palette will show grayscale.

- In the Selected Color area, click a color in the palette and click the box beside a design object. You can assign colors to the items in the Other, Component outlines, and Items on Primary areas.

  Requirement: You must select the check box of a design object before the object color swatch appears.

Tip: For monochrome output devices (printers/plotters), the only color available is black. You can save your selected colors (for items you select under Other, Items on Primary, and/or Component outlines) for each document type and use these settings as the defaults for new documents of that type. For more information, see Adding or Editing CAM Documents.

Applying Colors to Individual Nets

You can use the colors you have assigned to nets in the View Nets dialog box when you print.

- Select the **Color By Net** check box to use the View Nets colors in the output.

Previewing the Settings

- Click the Preview button to preview the settings you make in the Select Items dialog box.

See also: Previewing CAM Documents

Related Topics

Defining CAM Documents

Applying the Over(Under)size Value to All Layers

You can extend the value of the Over(Under)Size value to non-electrical layers using an attribute applied to the PCB level of the attribute hierarchy.
In PADS2007.1 or later, the attribute is automatically added to all designs when opening the design or when importing an ASCII file. Opened designs, created with version PADS2007 have the attribute value set to No to only apply the Over(Under)size value to electrical layers. Opened designs created with any other version have the attribute set to Yes to apply the Over(Under)size value to all layers.

Procedure

1. In the design, select an object (for example, a component, component pad, or net), right-click and click **Attribute.**
2. In the **Attributes for** list, click PCB PCB.
   Tip: This displays any attribute of the PCB at the PCB level of the attribute hierarchy.
3. The CAM.Apply Oversize To All Pads attribute appears in the attribute list.
4. Use an attribute value of Yes to apply the Over(Under)size value to all layers or use No to accept the default of applying the value only to electrical layers.

Result

- Is the Over(Under)size value not being applied correctly? There is a hierarchy of settings that apply to Solder Mask and Paste Mask layers, and the Over(Under)size value has the lowest priority. For more information, see Control of Solder Mask and Paste Mask.
- Is the attribute not listed when viewing the PCB attributes at the PCB level of the hierarchy? Add the attribute with the correct value.

Setting Drill Drawing Options

Use the Drill Drawing Options dialog box to set drill drawing legend and marker parameters.

In this section:

- Creating a Drill Drawing Workflow
- Activating and Positioning the Drill Chart
- Specifying Drill Markers
- Sorting Data in the Drill Data Table
- Modifying Drill Table Entries
- Saving Your Selections

Creating a Drill Drawing Workflow

1. Click **Add** in the Define CAM Documents dialog box.
2. In the Add Document dialog box, type a name in the Document Name box.
3. Select **Drill Drawing** in the Document Type list.
4. Select a Layer Association and click **OK**.
5. Click **Options**.

6. In the Plot Options dialog box, click **Drill Symbols**.
7. Use the controls in the Drill Chart area to enable, and define the position and size of the drill chart.

   Tip: The representation of the Drill Chart (and Drill Data) values depends upon how you set your design unit options (mils, metric, or inches).

Activating and Positioning the Drill Chart

You can automatically add a drill chart to your drill drawing. The default location of the drill chart is at the origin (0,0) of your design which may overlap design objects. You can alter the default location of the drill chart in your drill drawing.

1. In the Drill Chart area, select the **Draw Chart** check box to includes the legend in the plot.
2. In the **Letter Height** box, set the height of letters in design units.
3. In the **Chart Line Width** box, set the width of the chart lines in design units.
4. In the **Location** boxes, set the X,Y location of the drill chart in design units. The location is not saved when you click the Save As Defaults button.

Tip: You can click OK and use the Preview in the Plot Options dialog box to check the location against the board outline. The board outline is blue and the drill chart outline is pink.

Restriction: The drill chart is only output in the drill drawing CAM document. It is not embedded in the design.

Specifying Drill Markers

Use the controls in the Drill Symbol Markers area of the Drill Data area to specify how you want markers drawn. A symbol preview box enables you to view the graphical image of a selected drill symbol (for example, the plus sign, +).

1. In the **Height (Letter)** box, set the letter height of the drill symbol marker.
2. In the **Line Width** box, set the line width for the drill marker symbol and for all text.
3. In the **Height (Symbol)** box, set the drill marker symbol height.

Sorting Data in the Drill Data Table

You can populate and then sort the data in the drill drawing table by drill size or quantity of the holes, and then by plating type.

Tip: Populate data in the drill drawing table by manually adding entries or by gathering information from the design database with Augment or Regenerate. If you have not saved any new default settings and if no data previously existed in the table, Augment or Regenerate populates the table in ascending order of size and then plated holes before non-plated holes.

To Sort the Data in the Drill Drawing Table

1. Click on the appropriate column (Size, Quantity, or Plated) in the drill drawing table.
2. Click another column heading (Size, Quantity, or Plated) to also sort by that column.

The final sorting order is determined by the order in which the columns are selected for sorting.

For example, clicking the Size column first determines whether the data appears in numerical ascending or descending order. If you then click the Plated column, the list is reordered with plated or non-plated holes first while maintaining the ascending or descending numerical sorting order that you previously selected. If you then click the Quantity column, the Quantity sorting order overrides the previous selections.

Example

Table 39-1 shows the result of a sort by size (ascending order) and then by plated type.

Table 39-1. Example Sort Result

Symbol	Size	Quantity	Plated	Tolerance
+	0.0135	35	Yes	+0.000/0.0135
X	0.02	125	Yes	+/-0.003
Rectangle	0.037	62	Yes	+/-0.003
Diamond	0.048 X 1.020	44	Yes	+/-0.003
Rectangle +	0.056	8	Yes	+/-0.003
Rectangle X	0.072	4	Yes	+/-0.003
Diamond +	0.11	6	No	+/-0.005
Circle +	0.156	4	No	+0.005/-0.002

Sorting and Augment or Regenerate

The Augment button adds actual design drill sizes to a pre-existing default set of drill sizes. The Regenerate button clears all previous data, including any pre-existing defaults, and replaces it with the actual design data. Both Augment and Regenerate populate data in the following order of sorting precedence:

- If you have sorted data before clicking Augment or Regenerate, the appropriate data is formatted in the order currently specified by the open CAM document or the settings from the current editing session.

- If you have not sorted in the current session, but you have saved a set of sorting defaults, clicking Augment or Regenerate populates the table in accordance with your saved defaults.

Modifying Drill Table Entries

To modify the drill table entries shown in the Drill Data area:

1. To modify the symbol associated with a specific drill entry, double-click the target symbol entry in the Symbol column and select a symbol from the list of available drill symbols. The next time you click on the entry, the symbol preview window in the Drill Symbol Markers area reflects the symbol change.

   Restriction: Symbol usage is exclusive; you cannot save your changes if you create duplicate symbol assignments.

2. To specify or change the tolerance value for a specific drill entry, double-click the target tolerance entry in the Tolerance column and type a tolerance value.

   Tip: The Tolerance column gives you the flexibility of entering a text string of up to 32 characters, which enables you to add a fabrication note along with your tolerance value. As a general rule, be careful to type suitable tolerance values. You can copy and paste tolerance data between text fields, but you cannot perform multiple cell copy functions.

Restriction: You cannot edit data in the Size, Quantity, or Plated columns.

Saving Your Selections

If you manually build your own drill table, or you want to reuse the contents of the table as the defaults for all CAM drill drawings you can save the contents as the default.

- Click **Save As Defaults**

  Tip: The drill chart location is not saved when you click Save As Defaults.

Result

This creates a *cam.defaults* file located in the following location (default installation):
C:\MentorGraphics\<version>PADS\SDD_HOME\Settings

Related Topics

Adding or Editing CAM Documents

Defining CAM Documents

Assembly Variants

Substitute a Component for Assembly Variants

The Variant/Substitute dialog box appears when you choose to substitute a component in an assembly variant. When you substitute a component, the substitution is referred to as the active component. The original component that you substituted is referred to as the default. The default component is what exists in the base option and the raw database.

- Substituting Components
- Interpreting the Component Status
- Displaying Substitution Differences
- Previewing a Variant

Substituting Components

1. **Tools** menu > **Assembly Variants**.
2. There are several ways to begin substituting a component:
   - Click an Assembly Variant in which you want to substitute the component from the Variant **Name** list, select the component name to change, and click **Substituted** in the **Status** area.

     Or
   - Select a component in the multicolumn list, and click **Substitute** in the **Status** area.

     Or
   - Click **Substitute** from the **Verb Mode** list, and select a component in the Layout Editor.
3. Double-click a Part Type in the **Active** column of the multicolumn list in the Variant/Substitute dialog box. The multicolumn list displays the attributes (Value) of the component you are substituting, its Default value, and its Active value (the substitution).
4. In the list, select a value you want to use for the substitution.
5. Click **OK** to complete the substitution.
6. Click **OK** to save and apply the substitutions to the Assembly Variant.

To choose a different part type from a library click **Browse**.

Restriction: You cannot substitute the Family, Number of Pins, Number of Gates, or Signals Pins. This information is updated when you choose a different part type.

Interpreting the Component Status

The Status area shows the status of the component:

Current	The state before you click Substituted or Verb Mode.
New	The state after you click Substituted or Verb Mode.

For example, if a component is Installed and you want to Substitute it, Current displays Installed because the component was installed in the variant. New reads Substituted because you are creating a new substitution for the component.

Displaying Substitution Differences

You can display only the values of the Default and Active component which differ, in the multicolumn list. This includes items that you manually change and items that change because you select a different part type.

- Select the **Show Difference** check box.

Previewing a Variant

You can preview your substitutions.

- Click the **Preview** button. For more information, see Previewing Assembly Variants.

Using the Assembly Variants Dialog Box

Use the Assembly Variants dialog box to create a new variant, review or edit a variant, preview variants, delete variants, and create reports for variants.

In this topic:

Creating Assembly Variants

To create Assembly Variants:

1. **Tools** menu > **Assembly Variants**.
2. Type the name of the new variant, up to 26 characters, in the **New variant's name** box.
3. Click **Create**. A new variant is created that contains all the items in the Base Option.
4. Use the **Display** list to filter the items to view in the multicolumn list.
5. Uninstall, or substitute your design components.
6. When you finish defining the new Variant, click **OK** or **Apply** to save the changes to the new Variant. The Base Option is also updated based on the new variant.

You can continue to create new Variants.

Installing, Uninstalling, or Substituting Variant Design Components

You can install, uninstall, or substitute variant design components using two different methods.

Using the Multicolumn List

You can modify the component status using the multicolumn list and the Status area.

Restriction: Because you cannot directly modify the Status of the Base Option which is based on the other variants; modifying the Status is unavailable when you view the Base Option in the multicolumn list.

1. Select a variant in the Name list.
2. In the multicolumn list, select the items you want to uninstall from the new Variant and click **Not Installed**.

   Tip: Click the column header to sort the multicolumn list.
3. In the multicolumn list, select the items you want to substitute and click **Substituted**. The Variant/Substitute dialog box appears for each item.
4. Make any substitutions you want and click **OK**. You return to the Assembly Variants dialog box.

Using the Design Area/Layout Editor

You can also modify the component status using the Layout Editor. With Verb Mode you can decide what action to take and perform it on components in the Layout Editor (outside the dialog box).

1. Select a variant in the Name list.
2. In the Manager area, select **Install**, **Uninstall**, or **Substitute** in the Verb Mode list.
3. Click outside the dialog box. The dialog box remains open, but it is not active.
4. In the design area, select the components, one at a time, to which to perform the action.

   Tip: When using the Substitute Verb Mode, the Variant/Substitute dialog box appears after each component you select.
5. When you finish selecting components or want to change the Verb Mode, click inside the dialog box. The multicolumn list in the dialog box updates to reflect the actions you performed on items.

Tip: You cannot modify the BaseOption.

Modifying Assembly Variants

By modifying an Assembly Variants, you can, for example, choose to uninstall or substitute a component that is Installed in the variant you click in the Name list.

To change assembly variants:

1. **Tools** menu > **Assembly Variants**.
2. In the **Name** list, click the variant for which you want to view the status.
3. To view all components in the Assembly Variant click **All** in the **Display** list. The multicolumn list reflects the status of components for that variant.
4. In the multicolumn list, click any components you want to install, uninstall, or substitute for the variant.

   Tip: Click the column header to sort the multicolumn list.
5. Click **Installed**, **Not Installed**, or **Substituted**.

   When you click Substituted, the Variant/Substitute dialog box appears for each variant that you select to substitute.

   If you change the status of a component from Substituted to Installed, the variant uses the Default component.

   If a component is already substituted in the variant and you want to change the substitution values, select the component in the multicolumn list and click **Edit** in the Status area. The Variant/Substitute dialog box appears.
6. Make any substitutions in the Variant/Substitute dialog box and click **OK**. You return to the Assembly Variants dialog box.
7. Click **OK** or **Apply**.

Modifying Assembly Variants by Component

This topic describes modifying an Assembly Variants when you click Components in the Type list. If the component selected in the Name list is Installed in Options, you can uninstall it from or substitute it in Options.

1. **Tools** menu > **Assembly Variants**.
2. Click **Components** from the Type list.
3. In the **Name** list, click the component whose status you want to view.
4. Click **All** in the Display list to view the status of the component in all variants.
5. In the multicolumn list, select the variants from which you want to install, uninstall, or substitute the component.

   If a component is already substituted in the variant and you want to change the substitution, select the variant in the multicolumn list and click **Edit** in the Status area. The Variant/Substitute dialog box appears.

   Tip: Click the column header to sort the multicolumn list.
6. Click **Installed**, **Not Installed**, or **Substituted**.

   When you click Substituted, the Variant/Substitute dialog box appears.

   If you change the status of a component from Substituted to Installed, the variant uses the Default component.
7. Make any substitutions in the Variant/Substitute dialog box and click **OK**. You return to the Assembly Variants dialog box.
8. Click **OK** or **Apply** to save the changes to the variant. Substituted items are removed from the Base Option.

Deleting Assembly Variants

1. **Tools** menu > **Assembly Variants**.
2. In the **Name** list, click the variant which you want to delete.
3. Click the **Delete** button.

Previewing Assembly Variants

You can display Installed components in all of your Assembly Variants. You can view statistics about Assembly Variants, such as where component location, how components relate spatially to other components in the Assembly Variant, and what components you have created substitutes for.

To display, to not display, or to display components in a different color:

1. **Tools** menu > **Assembly Variants**.
2. Select the variant to preview in the **Name** list.
3. Click **Preview**. The Preview for <variant> dialog box appears displaying an assembly drawing of the variant.
4. Click **Variants** to open the Preview/Option dialog box. Use the Preview/Option dialog box to change the appearance of your preview. The multicolumn list box indicates whether objects of a status are currently visible in the preview window for a variant.
5. Double-click in the cell of the component in the Assembly Variant for which you want to change visibility.
    - Click **No** to make the components invisible.
    - Click **Yes** to make the components visible.
    - Click **Color** to choose a display component color. The Colors dialog box appears. Click a color in which to display selected components and click **OK**. You return to the Preview/Option dialog box appears. The multicolumn list reflects a Yes status for the component. If it has a color, it is visible.
6. Click **OK**. You return to the Preview for Variant dialog box. The preview area reflects the visibility status you set.

Creating Assembly Variant Parts Lists

You can create parts lists based on assembly variants.

- Click the **Report** button.

For more information, see Creating a Report Using an Assembly Variant.

Creating Assembly Variant Assembly Drawings

You can create assembly drawings from assembly variants. For more information, see Adding or Editing CAM Documents and Selecting an Assembly Variant for Assembly Drawings.

Selecting an Assembly Variant for Assembly Drawings

You can create assembly drawings of your assembly variants by selecting a variant in the Select Assembly Variant dialog box.

1. In the Add Document dialog box, click **Assembly**.

   Restriction: This button is unavailable unless you select the Assembly document type.

2. In the Select Assembly Variant dialog box, click the **Use Assembly Variant** check box to use an assembly variant as your design input for the assembly drawing.

   Tip: Clear the check box to use all parts in the database, known as the raw database.

3. In the Name list, select a variant.

Related Topics

Defining CAM Documents

Setting CAM Preview Options

Use the CAM Preview Setup dialog box to invert, show plot orientation, or overlay multiple CAM Documents - you can change the preview attributes of all documents.

Tip: You must have at least one CAM document defined to use preview setup.

In this topic:

- Overlaying CAM Documents
- Inverting a CAM Document
- Showing Plot Orientation of a CAM Document

Overlaying CAM Documents

You can overlay multiple CAM documents by enabling the visibility of documents.

- Select the **Visible** check box of each desired document.

Inverting a CAM Document

You can invert the view of a CAM document. You might want to invert the view of CAM Plane layers since they are negative layers and will appear different than all other layers.

- Select the **Inverse** check box of a document.

Showing Plot Orientation of a CAM Document

You can scale and orient the preview data as defined by the CAM document plot options.

- Select the **Orient** check box of a document.

Related Topics

Defining CAM Documents

Previewing CAM Documents

During CAM document operations, use the CAM Preview dialog box to preview your output. It displays all items that will appear in the final CAM document and their position within the plot paper extents.

Tip: You can also use the CAM Preview dialog box after you use Verify the Design from the Tools menu to perform fabrication checking and you want to preview the CAM document associated with the layer for a selected error.

In this topic:

- Controlling the View
- Viewing a Different Document
- Viewing Multiple Documents

Controlling the View

There are many methods of controlling the view in the CAM Preview dialog box.

- In the Zoom area, use the following buttons:
    - **Board button**—Fits the contents of the selected document into the preview window.
    - **Extents button**—Fits the world, or the whole coordinate system, into the preview window.
    - **Workspace button**—Fits the current screen view into the preview window.
- You can also alter the view with the mouse:
    - Right-click to zoom out by a factor of 2, centered on the pointer location.
    - Click to zoom in by a factor of 2, centered on the pointer location.
    - Click and drag to draw a zoom rectangle centered on the pointer location. Drag the pointer towards the top of the screen, diagonally from the indicated pointer locations to zoom in. Drag the pointer towards the bottom of the screen, diagonally from the indicated pointer locations to zoom out.
- If the preview doesn't appear properly, click **Refresh**. This occurs when you remove the application focus from the CAM Preview dialog box. For example, switch back to the preview window from another application and you will need to click Refresh.

Viewing a Different Document

You can switch to previewing any of the CAM Documents you have saved in your session.

- In the Documents list, select a document to preview.

Viewing Multiple Documents

You can invert, show plot orientation, or overlay multiple CAM Documents.

- Click the **Setup** button. The CAM Preview Setup dialog box appears and enables you to change CAM document attributes.

Related Topics

Defining CAM Documents

Printing

This section describes the following methods to print or plot your design:

- Printing to a Windows Printer
- Printing PostScript to a File

Printing to a Windows Printer

You can print CAM Documents on your printer.

Tip: Test the link between Windows and the printer by clicking **Print Topic** from the file menu in this Help window.

1. **File** menu > **CAM**.
2. Select the default folder to which to output your CAM files from the CAM Directory list. If you click **<Create>**, you can create a new folder. The default folder is \PADS Projects\Cam\default.
3. Click **Add** in the **CAM Documents** area. The Add Document dialog box appears.
4. From **Document Type**, click the document type you want to use (for example, Silkscreen).
5. From the Layer Association dialog box that appears, select the layer you want to use and click **OK**. All associated items, tracks, and vias are added to the Summary area.
6. Under **Output Device**, click the **Print** button. This indicates the output device. All printing is controlled using the Windows printer properties. For information on setting up printers and defining printer properties, see the Microsoft Windows Help (on Windows 2000, go to **Start** and click **Help**; on Window XP, go to **Start** and click **Help and Support Center**).

7. Click **Device Setup**. The Print Setup dialog box appears. In this dialog box, you can choose a printer, set printer properties such as resolution and number of copies, and set document properties such as orientation. For more information on this dialog box, see the Microsoft Windows online help.

   When you finish setting printer and document properties, click **OK**.

8. Under **Document Name**, type a meaningful name for the document, such as one that indicates the document type, the type of output, and the layer. Click **OK**. Your document is added to the CAM Documents list in the Define CAM Documents dialog box.

9. Click the **Layers** button to display the Select Items dialog box. Set the layers to output information on and click **Add**.

10. Establish any color changes. Click the color to use from the **Selected Color** area, and turn on the check boxes of the objects to assign to the color. The new color appears to the right of the object. Continue to assign/redefine the colors by clicking the color and the check box of the objects.

11. When you finish setting options, click **OK**. You return to the Add Document dialog box.

12. To print, click **Run**. The message "Do you wish to generate the following outputs?" appears.

13. Click **Yes** to print.

Related Topics

Printing PostScript to a File

Defining CAM Documents

Printing PostScript to a File

PADS Layout supports PostScript printing to a file through use of the Windows printer properties. For information on setting up printers and defining printer properties, see the Microsoft Windows Help (on Windows 2000, go to **Start** and click **Help**; on Windows XP, go to **Start** and click **Help and Support**).

You must set up your printer to print to a file before you create a PostScript file.

In this topic:

- Setting the Printer to Print to a File
- Printing a CAM Document to a File

Setting the Printer to Print to a File

1. Locate printer information based on your platform:
   - Using your Windows **Start menu**, locate your **Printers and Faxes** control panel.
2. Right-click the PostScript printer you want to use to print to a file and then click **Properties**. The Properties dialog box appears.
3. Click the **Ports** tab.
4. Under **Print to the Following Port**, in the **Port** column, select the **FILE:** check box, which has the description **Print to File**.

   Tip: This procedure works with local printers. If you are using network printers, you may not have access rights to select or change the port information.
5. Click **OK**. You can now print to a file.

Printing a CAM Document to a File

1. Complete the steps above to set up the printer to print to a file.
2. Click **CAM** from the **File** menu. The Define CAM Documents dialog box appears.
3. Select the default folder to which to output your CAM files from the CAM Directory box. If you click **<Create>**, you can create a new folder. The default folder is \PADS Projects\Cam\default.
4. Click **Add** in the CAM Documents area. The Add Document dialog box appears.
5. From **Document Type**, click the document type you want to use.
6. From the Layer Association dialog box that appears, select the layer you want to use and click **OK**. All associated items, tracks, and vias are added to the Summary area.
7. Under **Output Device**, click the **Print** button.
8. Click the **Device Setup** button to open the Print Setup dialog box. In this dialog box, you can choose a printer and set printer properties. For more information on this dialog box, see the Microsoft Windows Help.

   Click **OK** when you finish setting document and printer properties.
9. Under **Document Name**, type a meaningful name for the document and Click **OK**. Once you are satisfied with your CAM document definitions and have finished setting printing options, you can print as you normally would.

   See also: Printing

   Tip: For best results, make sure that you have a design loaded before generating the output file.

10. From the Add Document dialog box, click **Run**. The message "Do you wish to generate the following outputs?" appears.
11. Click **Yes** to print to a file. The Print to File dialog box appears (as long as you have selected a printer that is defined to print to a file).
12. Use the dialog box to specify a name (and, optionally, a path) for the output file. The default path is \My Documents\PADS Projects\Cam\default, but you may have changed this in step 3 above.
13. Click **OK**. The printer file is created.

Related Topics

Defining CAM Documents

Setting Up the Photo Plotter Output

Use the Photo Plotter Setup dialog box to set up photo plotting and send your output to a plotter

In this topic:

- Maintaining the D-Code list
- Maximum Aperture Count
- Automatic D-Codes
- Manual D-Codes
- Setting Aperture Dimensions

Maintaining the D-Code list

D-Codes contain the apertures required for the Photo Plotter. The D-Code list can be created automatically or you can maintain it manually. The D-Code list contains all defined apertures.

Tip: Before you generate your CAM files, regenerate your list of apertures.

Maximum Aperture Count

- In the Aperture Count box, type a value for the maximum aperture count.

  Restriction: When you click RS-274-X format in the Photo Plotter Advanced Setup dialog box, the aperture count is set to 989 and is unavailable.

Automatic D-Codes

- To automatically add apertures as you add information to the design, select the **Augment on-the-fly** check box.
- To automatically generate D-codes for items in the design file that are not in the current list, click **Augment.**

  Tip: Augment adds oval and rectangular flash apertures for orthogonal orientations only.
- To clear the current D-code list and automatically add D-codes for all items in the design, click **Regenerate.**

  Tip: Regenerate adds oval and rectangular flash apertures for orthogonal orientations only.

Manual D-Codes

- To add a new D-code to the D-Code list, click **Add**, type a value excluding the "D" prefix, and click **OK**.
- To delete a D-code, select a code in the D-Code list, and click **Delete**.

Setting Aperture Dimensions

You can customize the aperture dimensions of a selected D-Code.

- To draw lines and flashed items with the same aperture, select the **Same Aperture for Flashes/Lines** check box.
- To set a flash aperture, click a **Flash** shape or to set a draw aperture, click a **Line** shape. Line shapes are unavailable if the Same Aperture for Flashes/Lines check box is selected.
- To specify a width for square, rectangle, and oval shapes or the outer diameter of round and thermal shapes, type a value in the **Width** box. This box is unavailable if a value is not appropriate for the specified shape.
- To specify a height for oval and rectangular shapes, type a value in the **Height** box. This box is unavailable if a value is not appropriate for the specified shape.
- To specify an inner diameter for annular ring shapes, type a value in the **Inner Diam** box. This box is unavailable if a value is not appropriate for the specified shape.
- To specify the width used for shapes to be filled, type a value in the **Fill Width** box. An example shape is a non-orthogonal pad. Larger widths decrease photo plot time but provide a less precise approximation of the shape.

- To specify the width used for hatching of system texts, type a value in the **Texts Fill Width** box. Larger widths decrease photo plot time but provide a less precise approximation of the text.

Related Topics

Defining CAM Documents

Analyzing CAM Documents

Once you create your CAM documents, you can analyze them using DFM Analysis.

Accessing

- **Tools** > **DFM Analysis** > **Start Analysis**

Tip: To view help on DFM Analysis, press F1 with the dialog box open.

Setting up DFM Analysis

You can set up the optons you want to use while analyzing your CAM documents.

Accessing

- **Tools** > **DFM Analysis** > **Setup**

Tip: To view help on DFM Analysis, press F1 with the dialog box open.

Using the CAM Plus Assembly Machine Interface

The CAM Plus command generates computer-aided manufacturing (CAM) output files that are compatible with a variety of automatic assembly and pick-and-place machines. Before you use CAM Plus prepare an information file called part.def.

In this topic

- Creating a Part Definition File
- Setting the CAM Plus Options
- Interpreting Error Messages and Troubleshooting

Creating a Part Definition File

You must manually create a part.def file in order to use the CAM Plus option. Every part that is used in the design must be listed in the part.def file. By default, the part definition file is read from the \Libraries folder. For more information, see the Part Definition File topic in the *Concepts Guide*.

Setting the CAM Plus Options

1. **File** menu > **CAM Plus**
2. In the Part Definition Filename box, type **part.def**. The file is read from the \Libraries folder by default.
3. In the Setup area:

   a. Select a side of the board from the **Side** list.

   b. Select the type of parts to include in the report from the **Parts** list. Choose from SMT, ThruPin, All, and Masked. Masked parts are those that are assigned to the machine selected format as an insert class.

   c. Select the **Read Part Definition** check box to add the additional information contained in the Part Definition File, part.def, to the parts contained in a design.

   This information defines the insertion class for all parts. Read Part Definition scans the Part Definition File for information about the parts in the database. When an exact match is found between a part type name in the database and a part type name in the definition file, the information combines to provide the manufacturing output.

   d. Select the **Read Value Definition** check box to read the Value attributes for each part in the PADS Layout design and append the Value attribute to the part type name when matching each part type in the Part Definitions file, part.def.

   For example, an R1/4W part type with Value attribute 100K could have an entry in the Part Definitions file as follows:
   R1/4W{100K},ins=un6241,bodydiam=200,leaddiam=30,anvil=2

   e. Select the **Verify File** check box to produce an ASCII verification file.

   This ASCII file is stored in the \PADS Projects\Cam\<board_file_name> folder. This file contains 2D line data that describes the path for the inserted parts. The name of the file is the name of the interface program created, with the .asc extension. You can read this file into PADS Layout with the ASCII In command. It states the insertion path as 2D lines on layer 19. If a Part_Num value for a part is not found, Part_Num is set to Missing.

   f. Select the **Batch Part Def. File** check box to run all of the outputs with a single command.

   For each program you run, an output file is produced with the name suffix bt or bb, for example dym318bt.smt or un6241bb.put. If this file already exists, the message “Overwrite existing file (Y/N)?” appears. Click either Yes or No. If you click Yes, the file is overwritten and placed in the CAM subfolder. Each of the parts in the selected category is added to the file. A report called insert.lst lists each part and the machine that inserts it. The Batch command does not create a Verify file.

4. In the Geometry area:

    a. Type a value for the Board Offsets to define the offset of the machine's location dowel with regard to the 0,0 system origin board.

    These offset values convert the design coordinates to the machine origin. Allowable values are from 0 to 10 inches. Offset values are in inches; for example, 1250 is 1.25 inches. You may need to define a new Board offset for each machine.

    b. Type values for the Step/Repeat values to define whether to treat the board as a single design when creating the output program file, or to insert a number of boards simultaneously.

    When you insert a number of boards, you can define the number of steps in the X and Y direction and the step and repeat interval to use, as shown in Table 39-2. CAM Plus uses current design units for the offset and step and repeat values. Each machine has its own units type to which the data is always converted regardless of the current design units. CAM Plus generates assembly program files for inserting parts on all boards.

Table 39-2. X and Y Step and Count Options

Option	Description
X Count	Number of copies in X direction. The maximum is 20.
Y Count	Number of copies in Y direction. The maximum is 20.
X Step	The step distance in the X direction between the origin of each board. The maximum is 10 inches.
Y Step	The step distance in the Y direction between the origin of each board. The maximum is 10 inches.

The default is no step and repeat, equivalent to a step of 1 in X and Y.

5. In the Output Format list, select the machine format.

    Files are produced for all parts of a selected class: masked, through hole, SMT, top, bottom, and so on. All parts in the class are included in this output file, whether or not their insert class is defined as belonging to the specific machine.

6. In the Universal Tooling and Universal Axial Output lists select the desired settings if applicable.

    Restriction: Universal-specific instructions are available when you select Universal machine formats or check Batch Part Def. File.

7. Click **Run**.

8. Examine the Status Messages box for the current state of output.
9. Navigate to *C:\PadsProjects\CAM\Newfolder(Design Name)* to examine the output file.

Interpreting Error Messages and Troubleshooting

CAM Plus produces and prints error messages in the file padscim.err.

Related Topics

CAM Plus Assembly Machine Interface in the *Concepts Guide*

Chapter 40 Using Scripts

Use the Sax Basic Script and Editor features to design, develop, and run scripts that add to, replace, enhance, or customize existing PADS Layout features.

- Creating Scripts
- Managing Scripts
- Debugging Scripts
- Accessing Help on the Basic Language

Creating Scripts

You can create scripts to simplify redundant activities.

The following descriptions are included in this topic:

- Creating a Script
- Inserting an Automation Statement
    - Using the Object and Procedure Lists
    - Using the ActiveX Automation Members Dialog Box
- Setting the Next Statement
- Showing the Next Statement
- Saving the Script

Creating a Script

To create a script:

1. **Tools menu** > **Basic Scripts** > **Basic Script Editor**.

   In PADS Layout and PADS Logic, the SAX Basic Engine dialog box appears.

2. Click the **New** button.

Inserting an Automation Statement

You can add automation statements to the bottom of a script automatically.

Using the Object and Procedure Lists

Use the Object and Procedure lists to select and insert a statement. These lists contain the most commonly used statements.

1. Click the **Object** list and click an object type. The Object list shows all the objects for the current module. The (General) object groups all of the procedures that are not part of any specific object.
2. Click the **Procedure** list and click a non-bold procedure to insert. The Procedure list shows all the procedures for the current object. Selecting a procedure that is not bold inserts the proper procedure definition for that procedure.

The statement appears at the bottom of the script.

Using the ActiveX Automation Members Dialog Box

- In the Basic Script Editor, right-click and click **Debug** > **Browse**.

Use the ActiveX Automation Members dialog box to select and insert a statement. This dialog box contains an extensive list of statements.

Tip: If the pointer is on any line in the script other than the bottom line, the line is overwritten.

Setting the Next Statement

You can force a particular line in a script to run next. You can only select statements in the current subroutine or function.

To set the next statement:

1. In the Basic Script Editor, put your cursor on the line you want to run next.
2. Right-click and click **Debug** > **Set Next Statement**.

An instruction pointer appears next to the selected line. This line, and only this line, will run next. If you go to other parts of the script, you can return to this line by clicking Show Next Statement.

Showing the Next Statement

- In the Basic Script Editor, right-click and click **Debug** > **Show Next Statement**.

An instruction pointer indicates the next statement to run. Pausing a running script or setting a statement to run next sets the next statement. You can locate the set statement from anywhere in the script.

Saving the Script

1. In the Basic Script Editor, click the **Save** button.

2. Enter a file name, if necessary, and then click **Save**.

Related Topics

- Basic Scripting in the *Concepts Guide*

Running Scripts

You can run an existing script using Run. Run also resumes the playback of a paused script. When you run a script, you cannot use the mouse in the workspace.

The following descriptions are included in this topic:

- Running a Script
- Pausing a Running Script
- Stopping a Running Script

Running a Script

You can run a script from the Basic Scripts dialog box, or you can run a run a script from the Basic Script Editor. Running it from the Editor gives you options for pausing and stopping the script.

From the Basic Scripts dialog box

1. **Tools** menu > **Basic Scripts** > **Basic Scripts**.
2. Select a script from the list.

   If the script you want to run does not appear in the list, click **Load File** and browse for the script to load.
3. Click **Run**. Scripts are compiled when you run them. You cannot run multiple scripts at the same time.

   If the selected script has a compiling error, it automatically opens in the Basic editor for correction.

From the Basic Script Editor

1. In the Basic Script Editor, open a script file.
2. Right-click and click **Macro** > **Run**.

Alternative: On the Basic Script Editor toolbar, click the Start/Resume button.

Pausing a Running Script

When running a long script, you may need to pause it to perform some other design activity.

- In the Basic Script Editor, right-click and click **Macro** > **Pause**.

  Alternative: On the Basic Script Editor toolbar, click the Pause button.

Tip: If you paused the script, you can also use Run, Step Over, or Step to Cursor to resume running the script. Right-click and select **Run** to resume running the script.

Stopping a Running Script

You can stop a running script at any time. However, you cannot resume running a script once you have stopped it. When you click Run, the script starts from the beginning.

- In the Basic Script Editor, right-click and click **Macro** > **End**.

  Alternative: On the Basic Script Editor toolbar, click the Stop button.

  Tip: While a script is running, a third command, Stop *scriptname*, appears on the Basic Scripting submenu (on the Tools menu) along with Basic Editor and Basic Scripts. Click Stop *scriptname* to halt the execution of the currently running script.

Related Topics

- Basic Scripting in the *Concepts Guide*

Managing Scripts

The following descriptions are included in this topic:

- Opening an Existing Script
- Managing Open Scripts
  - Opening #uses Modules
  - Closing an Open Script
  - Closing all Open Scripts
  - Viewing a Particular Script
- Editing a Script
- Editing a User Dialog Box
- Finding an Automation Statement
- Printing a Script
- Saving a Script
- Watching a Variable

Opening an Existing Script

Scripts are created in and stored in script files that have a .bas extension. The default location for .bas files is C:\PADS Projects. To open an existing script:

1. In the Basic Script Editor, click the **Open** button.

2. Select the script and then click **Open**.

You can have up to nine scripts open at the same time.

Managing Open Scripts

The commands on the Sheet submenu provide script management methods. Since you can have up to nine scripts open at the same time, you can open #uses, close sheets, close multiple sheets, and choose scripts to view and edit.

Opening #uses Modules

#Uses modules are Basic scripts that are called from within other scripts. To open these secondary scripts:

- In the Basic Script Editor, right-click and select **Sheet** > **Open Uses**.

The #uses modules called in the script appear as script sheets in the Basic Script Editor. They are assigned a numbered tab and you can edit or run them.

Closing an Open Script

- In the Basic Script Editor, right-click and select **Sheet** > **Close**.

Alternatively, you can double-click the script's numbered tab in the gutter.

Closing all Open Scripts

- In the Basic Script Editor, right-click and select **Sheet** > **Close All**.

Viewing a Particular Script

If you have multiple scripts open, you can view a particular open script. You can have up to nine scripts open at the same time.

To view a particular script:

- Right-click and select **Sheet**. Then click the script you want to view from the list of open scripts on the submenu. Alternatively, you can click the script's numbered tab in the gutter.

Editing a Script

You can copy or cut selected text from the Basic Script Editor to the Clipboard. You can also paste a selection from the Clipboard into the text window. You can also paste text from the Clipboard into other applications.

To copy or cut and paste text in a script:

1. In the Basic Script Editor, select the text you want to copy or cut.

2. Right-click and click **Edit** > **Copy** or **Cut**.
3. Right-click and select Edit > **Paste** to paste the script text. Your selection is pasted in the Output window at the insertion point.

   Alternative: Click the Copy, Cut, and Paste buttons on the Basic Script Editor toolbar.

Editing a User Dialog Box

A UserDialog is defined by a Begin Dialog...End Dialog block. To graphically edit a user dialog:

1. In the Basic Script Editor, put your cursor in a UserDialog block of the script.
2. Click the **Edit UserDialog** button.

   See also: *Sax Basic Editor On Line Help* (C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\sbe5_000.hlp)

Finding an Automation Statement

If you are working with a long script, you can search for particular statements.

1. In the Basic Script Editor, click the **Object** list and select an object type. The Object list shows all the objects for the current module. The (General) object groups all of the procedures that are not part of any specific object.
2. Click the **Procedure** list and select a bold procedure. The Procedure list shows all the procedures for the current object. Selecting a procedure that is bold locates the procedure in the script.

The statement appears in the Basic Script Editor.

Printing a Script

To print a Basic script:

1. Open the script in the Basic Script Editor.
2. On toolbar, click the **Print** button.

Saving a Script

1. In the Basic Script Editor, click the **Save** button.
2. Enter a file name, if necessary, and then click **Save**.

Watching a Variable

Quick Watch shows the value of the expression under the cursor in the immediate window.

- Right-click and click **Quick Watch**.

Alternative: In the Basic Script Editor, click the Quick Watch button.

See also: *Sax Basic Editor On Line Help* (C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\sbe5_000.hlp)

Related Topics

- Basic Scripting in the *Concepts Guide*

Debugging Scripts

When running a script, you can run it step-by-step or to a certain location in the script. To perform these debugging tasks, insert breakpoints in the script at the points at which you want the script to stop.

This topic discusses the following:

- Setting or Removing the Breakpoints
- Removing All Breakpoints in the Script
- Debugging the Scripts
- Correcting Run-time Errors

Setting or Removing the Breakpoints

The ability to set or remove breakpoints is useful when you debug a script. If the Basic engine encounters a breakpoint when running a script, it pauses the script.

To set a breakpoint in a script:

1. Place the cursor on the line to which to add a breakpoint.
2. On the Basic Script Editor toolbar, click the **Toggle Breakpoint** button.

   Alternative: In the Basic Editor, right-click and click **Debug** > **Toggle Break**.

Result: This action inserts a breakpoint at the current cursor location. A breakpoint marker appears in the gutter area.

When the Basic engine encounters a breakpoint while running a script, it pauses the script. The next line in the script is marked with the instruction pointer.

Debugging the Scripts

Once breakpoints are inserted, you can debug scripts using the following tasks.

To run a single line of the script:

- On the Basic Script Editor toolbar, click the **Step over** button.

To perform a subroutine call on the current line:

- On the Basic Script Editor toolbar, click the **Step into** button.

  Alternative: In the Basic Script Editor, right-click and click **Debug** > **Step Into**.

To return from the subroutine to the point from which it was called:

- On the Basic Script Editor toolbar, click the **Step out** button.

To run a script to a point:

- In the Basic Script Editor, right-click and click **Debug** > **Step to cursor**.

To continue the execution from the current point:

- On the Basic Script Editor toolbar, click the **Run** button.

  Alternative: In the Basic Script Editor, right-click and click **Macro**> **Run**.

Removing All Breakpoints in the Script

- In the Basic Script Editor, right-click and click **Debug** > **Clear All Breaks**.

This removes all breakpoints in the script.

Correcting Run-time Errors

If run-time errors occur, the script debugger switches to step-by-step mode and displays a detailed message on the status bar. The instruction pointer is set on the line that produced the error. After fixing the error, you can resume running the script.

Related Topics

- Basic Scripting *in the Concepts Guide*

Accessing Help on the Basic Language

While writing or running scripts, you can access Help that provides information and a sample script using the Basic language statements.

To access the Basic Script Editor Help:

- Select or click in an item in color in the edit area of the Basic Script Editor and then press **F1**.

  Help appears for the current statement.

See also: Sax Basic Editor Help. In the Basic Script Editor, right click, point to Help, then click Editor Help.
(*C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs\sbe5_000.hlp*)

Chapter 41
OLE in PADS Layout

PADS Layout object embedding capabilities allow design engineers to insert other files or other applications as linked or embedded objects within a PADS Layout design. You can insert a Microsoft Word document, a Microsoft Excel spreadsheet containing a Bill of Materials, video or audio clips, and so forth. PADS Layout does not need to understand the format of the inserted object; PADS Layout communicates with the application that created the file and that source application tells PADS Layout what information to display and how to display it.

Restriction: The insertion of PADS Logic, Layout or Router files as OLE objects in other files (including other PADS files) is not supported. Any PADS Logic, Layout or Router file inserted in another file will not behave properly and cannot be edited within the "container" application (Visual Editing).

Inserting OLE Objects in PADS Layout

You can insert a linked or embedded object into your design.

Procedure

1. **Edit** menu > **Insert New Object**. The Insert Object dialog box appears.
2. Click whether to Create New or Create from File. Create New inserts a new OLE object. Create from File inserts an existing file as an OLE object.
3. If you clicked Create New, click the type of OLE object you want to create. If you clicked Create from File, click the file you want to insert as an OLE object.
4. If you clicked Create from File, and you want to link the inserted object to the original file, click **Link**. If you choose not to link the object, it is an embedded object.
5. If you want to display the linked or embedded object as an icon, click **Display as Icon**.
6. Click **OK** to insert the linked or embedded object in PADS Layout.

See also: Object Linking and Embedding in the *Concepts Guide*

Tip: Once you insert an object into a PADS Layout design, you can export it to an .ole file and import it into other designs.

Embedding a Text Document

Using OLE, you can embed a text document in your design to more quickly add multiple lines of text since the Text tool on the Drafting toolbar only allows single lines of text.

Embedding is recommended over linking since the embedded document will reside inside the .pcb file and can't get lost or accidentally deleted as an external file.

You can see a sample of this in the preview.pcb sample design. See the Notes section below the board outline. Double-click the text to activate the Microsoft Word document.

Restrictions

- OLE objects can only be printed. They cannot be plotted by a pen or photo plotter.
- Plot OLE objects must be enabled in the Plot Options dialog box to appear in the printout, but they will never be visible when viewing the Print Preview.
- OLE objects can only be printed using a zero plot orientation.

Procedure

- Follow the instructions in **Inserting OLE Objects in PADS Layout.**

  Tip: You can resize the object within your design when the object is active for editing.

Related Topics

Adding Free Text

Selecting OLE Objects

Select and manage OLE linked or embedded objects just like you manage nontext Word items. Click on the OLE object to select it.

Some limitations exist for selecting OLE objects:

- You cannot select more than one OLE object at a time.
- You cannot use area select to select OLE objects.
- Commands apply to selected OLE objects only, even if you also select PADS Layout objects. OLE objects have selection priority over PADS Layout components.

Tip: To select PADS Layout items under an OLE object, move the OLE object.

When you select an OLE object:

- Right-click to access a shortcut menu that lists all commands that you can apply to the OLE object.
- The object is selected just as a nontext object in Word is selected, using a rectangular area with sizing handles to indicate that it is selected. (Sizing handles are small, black squares that appear at the corners and along the sides of a rectangular area surrounding a selected object.) OLE objects do not use the PADS Layout method of highlighting to indicate selection state.

Editing OLE Objects in PADS Layout

Editing an OLE linked or embedded object is similar to editing PADS Layout objects. You can cut, copy, and paste OLE objects using the Cut, Copy, and Paste commands from the Edit menu.

Tip: You cannot cut, copy, or paste OLE objects when in the PCB Decal Editor.

You can perform the following edits on an OLE linked or embedded object:

- Copy an OLE Object
- Cut an OLE Object
- Paste an OLE Object
- Change the Background Color of an OLE Object
- Move an OLE Object
- Size an OLE Object
- Delete an OLE Object

Warning: Undo and Redo do not effect OLE objects. Since you cannot Undo or Redo actions performed on OLE objects, use care when modifying OLE objects.

Copy an OLE Object

To copy an OLE object:

1. Select the OLE object to cut.
2. Click **Copy** from the **Edit** menu.

Warning: Undo and Redo do not effect OLE objects. Since you cannot Undo or Redo actions performed on OLE objects, use care when editing OLE objects.

Cut an OLE Object

To cut an OLE object:

1. Select the OLE object to cut.
2. Click **Cut** from the **Edit** menu.

Warning: Undo and Redo do not effect OLE objects. Since you cannot Undo or Redo actions performed on OLE objects, use care when editing OLE objects.

Paste an OLE Object

Editing an OLE linked or embedded object is similar to editing PADS Layout objects. You can cut, copy, and paste OLE objects using the Cut, Copy, and Paste commands from the Edit menu.

Tip: You cannot cut, copy, or paste OLE objects when in the PCB Decal Editor.

To copy an OLE object:

1. Cut or copy the OLE object to place it in the Windows paste buffer.
2. Click **Paste** from the **Edit** menu.

The item is pasted into your design.

Warning: Undo and Redo do not effect OLE objects. Since you cannot Undo or Redo actions performed on OLE objects, use care when editing OLE objects.

Change the Background Color of an OLE Object

To change the background color of an OLE object:

1. Select the OLE object.
2. Right-click and click **White Background**. A check next to the command indicates that the object will use a white background.

Move an OLE Object

Move OLE linked or embedded objects just as you resize nontext objects in Word.

To move an OLE object:

1. Select the object.
2. Click and hold the left mouse button.
3. Move the pointer to move the object.
4. Release the mouse button once the object is in the correct location.

Size an OLE Object

Resize OLE linked or embedded objects just as you resize nontext objects in Word.

To size an OLE object:

1. Select the object.
2. Click and hold the left mouse button on one of the sizing handles. Sizing handles are small, black squares that appear at the corners and along the sides of a rectangular area surrounding a selected object.
3. Move the pointer; the object changes size according to the pointer movements.
4. Release the mouse button when the object is sized correctly.
5. Small, black squares that appear at the corners and along the sides of a rectangular area that surround a selected object.

Delete an OLE Object

You can delete OLE linked or embedded objects.

To delete an OLE object:

1. Select the OLE object to delete.
2. Click **Delete** from the **Edit** menu, or press **Delete**. You are asked to confirm the deletion.
3. Click **Yes** to delete the object.

Warning: Undo and Redo do not effect OLE objects. Since you cannot Undo or Redo actions performed on OLE objects, use care when editing OLE objects.

Tip: To delete all OLE objects in the design, click Delete All OLE Objects from the Edit menu.

Editing OLE Links

You can edit the link of a linked OLE object. Editing the link allows you to update the link, open the original object source, change the original object source, or break the link with the object source to make an embedded OLE object. You can also choose to update the object automatically or using a manual command.

To edit the link of an OLE object:

1. Click **Links** from the **Edit** menu. The Links dialog box appears.
2. Click the link you want to edit from the list.

3. Click the options you want to use or modify.
4. Click **Close** to close the Links dialog box. You cannot cancel changes you make in the Links dialog box.

Editing an OLE Object's Content

You can edit an object's content within PADS Layout (known as in-place visual editing), or in a separate window. In either case, you edit its contents as you normally would using all of the source application's commands and tools.

In this topic:

- In-place Visual Editing in PADS Layout
- Separate Window Editing In PADS Layout

Tip: See the documentation for the source application for more information on displaying, selecting, deleting, and saving PADS Layout objects in container applications.

In-place Visual Editing in PADS Layout

Visual Editing occurs when the source application for a linked or embedded OLE object opens within PADS Layout. You can also edit an OLE object by opening the source application and editing the object in the environment in which it was created.

To edit an object within PADS Layout, double-click on the object.

Click outside of the object to deactivate visual editing. Updates are automatically reflected in the object.

Restriction: You cannot edit embedded PADS programs within PADS Logic with visual editing.

Tips:

- Linked objects cannot be edited in place: they open in a separate window for editing.
- If the container application does not support in-place visual editing, the object will open in a separate window.

Separate Window Editing In PADS Layout

To edit an embedded object outside of the container application:

- Select the PADS Layout object. Click **Edit**, point to **PADS Layout Object**, and click **Edit**.

- Ctrl+double-click the OLE object to edit in the source application. The source application opens and you edit the object.

To update the object in the container application:

- Click **Update Document** from the **File** menu. This forces a redraw of the object.
- Set an option for PADS Layout OLE objects in the Global tab of the Options dialog box. When you click Update on Redraw, the object in the container application will update whenever you perform a redraw in the separate editing window.

  Turn this option off for best performance.

To return to the container application, click **Exit** from the **File** menu and Return to <host>.

Tip: If you want to save the object you edit in the separate window, you can click Save Copy As from the File menu. The object is really a copy of the original, and this command lets you save this copy. You cannot open other files, create new files, or save original designs in the separate window.

Saving OLE Objects

Linked and embedded objects are automatically saved as part of your design when you save a PADS Layout design. If you want to save OLE objects separately, use Export to save the objects in an .ole file. You can then use Import to import them into other designs.

Procedure

1. **File** menu > **Export**.
2. Click **OLE Files (*.ole)** from the **Save as Type** list.
3. Type a name for the OLE file you are saving.
4. Click the location for the file.
5. Click **Save**.

Chapter 42 Troubleshooting

In this chapter:

Warning: Test Point Locked Dialog Box

When you modify clusters that contain test points; modify vias, pins, or jumper pins that are locked test points; or modify routes that are connected to locked test points, a Warning dialog box appears. This warning dialog box performs different functions depending on whether you are modifying vias, pins, or routes. The following topics discuss your options for handling locked test points during each of the following circumstances:

- Modifying a jumper pin that is a locked test point
- Modifying a pin that is a locked test point
- Modifying a via that is a locked test point
- Modifying a route attached to a locked test point
- Moving, dispersing, or aligning a component, cluster, or union with a locked test point
- Move Sequential to move components, unions, or clusters with a locked test point

Recovering from Database Problems

When PADS Layout encounters an unrecoverable problem, a warning stating the error code and the message “Please read the on-line help topic Recovering From Database Problems” appears.

The following topics outline what to do when you have a database error and how you can recover or restore lost data:

Database Errors During Normal Editing Operation

Database Error Check During File Loading, ASCII, or DXF Import

Verifying and Restoring Lost Data

To Test Database Integrity Using ASCII

Comparing Netlists Between a Current and an Original Database

To Restore Data if Differences Exist Between the Damaged and the Original Database

To Test Database Integrity Using ASCII

Related Troubleshooting Topics

Crash Detection, BMW and BLT

Database Error Check During File Loading, ASCII, or DXF Import

Whenever you load a file, it undergoes a database integrity check. PADS Layout verifies that values and limits for the database are within an acceptable range.

If you encounter problems one of the following happens:

- A dialog box appears asking you to enter an acceptable value.
- You are prompted to confirm an automatic database correction routine.
- The message “Fatal Database Error” appears. In this case, you can still recover your data. See Recovering from Database Problems for instructions.

The database correction routine works in two ways:

- Automatically modifying or inserting data to correct a design.
- Deleting erroneous data from a design.

After you use entered data or the automatic correction routine to fix errors, run a set of interactive checks on the new database. These interactive checks should include, but are not limited to:

- Clearance checks
- Comparing netlists
- Check continuity
- Check tie plane

For instructions on how to run these checks see Verify the Design. This data verification indicates any differences between the corrected database and the original database.

Database Integrity Check During Normal Use

To run the same database integrity check that occurs during file loading type **I**, for integrity check, and press Enter. PADS Layout verifies that the values in the database are within an acceptable range. If problems are encountered, a dialog box appears and either asks you to enter an acceptable value or to confirm an automatic database correction routine. For a description of the process and corrective action, see "Database Error Check During File Loading, ASCII, or DXF Import."

Database Errors During Normal Editing Operation

During normal use, PADS Layout constantly monitors the design in memory. If a serious error occurs, the program saves the design to the backup file and a "Fatal Database Error" message appears. After an error, do the following to preserve the file:

1. Copy the backup file, Layout.pcb, and the three sequential backups, Layou1.pcb, Layou2.pcb and Layou3.pcb, to new names or to a new location on your hard drive.
2. Restart PADS Layout.
3. Load the original Layout.pcb backup file. Depending on the error you may be able to reload the file and continue.
4. If you cannot load Layout.pcb, try to load the sequential backup files and use the file closest to the time when the error occurred.

Once the file is loaded, follow the steps in "To Test Database Integrity Using ASCII."

If you can recreate the error, call Technical Support. Inform the technician of:

- The steps to reproduce the problem.

- Your software and hardware configurations.
- Your registration number.
- Whether you can send the design file for troubleshooting.

To Restore Data if Differences Exist Between the Damaged and the Original Database

If the part or connection information differs at all after running Compare Netlist, you can restore lost data using one of two methods.

If data loss is minimal:

1. Open the ECO toolbar and use the **Route**, **Add Pin Pair**, **Add Part**, **Rename Net**, and **Rename Part** commands to correct the design manually.
2. After data is restored, compare netlists to ensure that no differences exist between the corrected and the original design.
3. If the Differences Report indicates no differences, run the other PADS Layout Verify the Design routines to ensure that spacing, continuity, and plane net data are correct.

If data loss is extensive:

- Use the ECOGEN program to restore data. ECOGEN runs both inside Windows and as a standalone DOS program. ECOGEN compares a new netlist and an old netlist of a design. The comparison creates a file containing commands to update the new version of the design so that it contains the same part and connection data as the older version.

To Correct the Registration File

The registration file, powerpcb.reg, defines all registry keys required for the proper registration of PADS Layout OLE components. Also, other programs acting as clients access the PADS Layout OLE Automation server through the registry file. This file is automatically created by the Installation program and saved, for reference, in the same folder as the powerpcb.exe.

Warning: Never change the contents of this file. Incorrect PADS Layout registry entries may result, making PADS Layout unable to run.

If the registry file becomes corrupted, you can correct it. To correct the registry:

1. Open the Windows Explorer by clicking **Start**, pointing to **Programs**, and clicking **Windows Explorer**.
2. Double-click on the powerpcb.reg file. Explorer passes this file to the Windows Registry where the powerpcb.reg entries are updated, or refreshed.

   See the “Troubleshooting” topic in the PADS Layout Automation Server Help for more information.

To Test Database Integrity Using ASCII

You can save an ASCII version of your binary file from PADS Layout. If you suspect that your database contains errors or corrupt data, you can run integrity checks on the database using the ASCII format.

When you perform an ASCII Import, the database integrity check runs to determine if your database contains proper values. A second check occurs when PADS Layout actually reconstructs the binary data in memory. If data is missing, incomplete, or incorrect, the ASCII In routines will report it.

To run the integrity check, first create an ASCII file:

1. Click **Export** from the **File** menu. The Export dialog box appears.
2. Type a name for the ASCII file.
3. Click **Save**.

4. In the ASCII Out dialog box, click **Select All**, set the units to **Basic**, and select the correct format. Click **OK**.

Note: You cannot export the Perform 6 ASCII format from PADS Layout.

When the ASCII export is complete, save your current .pcb design and create a new file. In the new file:

1. Click **Import** from the **File** menu. The Import dialog box appears.
2. Select the filename of the ASCII file you just created.
3. Click **OK**. If no errors are found during loading your database is stable.

   If problems exist, an error file is generated. See "Database Error Check During File Loading, ASCII, or DXF Import."

Chapter 43
SPECCTRA Link

In this section:

Passing Maximum Number of Vias to SPECCTRA

You can set up the maximum number of vias in PADS Layout, and the SPECCTRA Link automatically passes them to SPECCTRA.

Warning: If the maximum number of vias rule is not supported by the set of licensed SPECCTRA options you have enabled, SPECCTRA may ignore this rule or even disable autorouting.

SPECCTRA Output File Location and Router Settings

Use the Setup SPECCTRA Finish dialog box to specify output file locations and to set instructions for the SPECCTRA router regarding actions performed when routing is completed, such as running the mitering pass, running re-cornering, and insertion of test points.

To specify output file settings and SPECCTRA router instructions for import to PADS Layout from the Setup SPECCTRA Finish dialog box:

1. **O**n the **File** menu**,** click **Export.**
2. In the File Export dialog box, in the Save as type list, click **SPECCTRA Files (*.do).**
3. Browse to overwrite a file or type a new file name. Click **Save.**
4. In the SPECCTRA Link dialog box, click **DO File**.
5. In the SPECCTRA Do File dialog box, click **Finish**.
6. To indicate the wires file, type or browse to the location in the **Wires File** box.

7. To indicate the routes file, type or browse to the location in the **Routes File** box.
8. To indicate the session file, type or browse to the location in the **Session File** box.
9. Select the options you want for test points installed by SPECCTRA in the Test Points area.

   See also: Passing DFT Audit Settings to SPECCTRA in the *Concepts Guide*, “Testpoint” topic in the *SPECCTRA Help*
10. Select a miter conversion type from the **Miter** area.
11. Select a recornering option from the **Recorner** area.
12. To remove crossover and clearance violations, click **Delete Conflicts**.
13. To eliminate notches and remove extra bends, click **Critic**.
14. To add extra space if there is room, click **Spread** and type the spread value in the **Extra** box.
15. Select the type of data you want to include in the report from the **Reports** area.
16. To indicate the report file, type or browse to the location in the **Report File** box.
17. Click **Apply**. The appropriate lines are added to your .do file at the last pointer location.

    Tip: To remove a line, select it in the .do file in the Editor area and press Delete on your keyboard.

Related Topics

SPECCTRA Help

Interface

Loading In and Out of SPECCTRA Automatically

When you start SPECCTRA from within PADS Layout, the SPECCTRA Link dialog box enables you to load in and out of SPECCTRA automatically.

Alternative: If you start SPECCTRA independently of PADS Layout, use the Stand-alone SPECCTRA Link dialog box to load in and out of SPECCTRA manually.

To load your design into and out of the SPECCTRA router in batch mode follow these steps:

1. On the **File** menu, click **Export.**
2. In the File Export dialog box, in the Save as type list, click **SPECCTRA Files (*.do).**
3. Browse to overwrite a file or type a new file name. Click **Save.**

4. In the SPECCTRA Link dialog box, type or browse for the .DO file you want to use.
5. To create or edit the .DO file, click **DO File**.

   See also: Creating or Editing a .do File
6. To set the SPECCTRA automatic startup information, click **Setup**.

   See also: Setting the SPECCTRA Automatic Startup Information
7. To specify output files (routes file and session file), type in their locations on the Setup SPECCTRA Finish dialog box. To access it, click **DO File** on the SPECCTRA Link dialog box, then click **Finish** on the SPECCTRA Do File dialog box and enter the file locations.

   See also: SPECCTRA Output File Location and Router Settings
8. To set options for sending via keepout information, passing advanced rules, setting a trace arc translation mode, or returning unused pins net, click **Options**.

   See also: Setting SPECCTRA Options
9. Click **Continue**. The design loads into SPECCTRA and the router runs in batch mode.

   When translation completes, the modified design file appears on the screen, loaded into a new PADS Layout session (unless Launch Mentor session is cleared in the Setup dialog box). The Link starts a new session so it does not interrupt any current sessions. If you run the router overnight, you can close the PADS Layout session to save memory.

Loading In and Out of SPECCTRA Manually

If you start SPECCTRA independently of PADS Layout, you can manually control the SPECCTRA interface. This gives you more control over how to use SPECCTRA. You can import from and export to SPECCTRA, and you can set instructions for the SPECCTRA router regarding actions performed when routing is completed (for example, running the mitering pass, running re-cornering, and inserting test points).

Alternative: If you start SPECCTRA from within PADS Layout, you use the SPECCTRA Link dialog box to load in and out of SPECCTRA automatically.

To run SPECCTRA manually:

1. If you are in PADS Layout, save the .pcb file.
2. Use Windows Explorer to navigate to your PADS program files directory, and double-click **pads2sp.exe**. The SPECCTRA Link dialog box appears.

   Example:
   C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\pads2sp.exe
3. To set up the SPECCTRA automatic startup information, click **Setup**.

See also: Setting the SPECCTRA Automatic Startup Information

4. To translate the .pcb file to SPECCTRA format click **To SPECCTRA**. The To SPECCTRA dialog box appears.

   See also: Translating Design Data from PADS Layout to SPECCTRA

5. To run SPECCTRA, click **Start**, point to **Programs**, point to **SPECCTRA**, and click **SPECCTRA**.

   Tip: You can also launch SPECCTRA by checking the Startup SPECCTRA check box on the To SPECCTRA dialog box.

6. Use the File operations in SPECCTRA to load the translated design (.dsn) file.

7. When you're finished with SPECCTRA, to run the Link again to translate the output back to a .pcb design file, click **From SPECCTRA**.

   See also: Translating Design Data from SPECCTRA to PADS Layout

8. Load the new .pcb file into PADS Layout.

Translating Design Data from PADS Layout to SPECCTRA

Use the To SPECCTRA dialog box to translate a .pcb design file into a SPECCTRA design file.

1. Use Windows Explorer to navigate to your ...\SDD_HOME\Programs directory, and double-click **pads2sp.exe**.

2. In the SPECCTRA Link dialog box, click **To SPECCTRA**.

3. To indicate the file to send to SPECCTRA, type or browse to the location in the **PCB File** box.

4. To indicate the design file (.dsn) that SPECCTRA inputs, type or browse to the location in the **Design File** box.

5. To indicate the .do file to send to SPECCTRA, type or browse to the location in the **DO File** box. The .do file is the script file that controls SPECCTRA operation.

6. To indicate the output file (.did) that SPECCTRA creates, type or browse to the location in the **Did File** box. This file serves as an input .do file in a subsequent SPECCTRA session.

7. To start SPECCTRA after the batch conversion is complete, click **Startup SPECCTRA**.

8. To create or edit the .DO file, click **DO File**.

   See also: Creating or Editing a .do File

9. To set options for sending via keepout information, passing advanced rules, setting a trace arc translation mode, or returning unused pins net, click **Options**.

   See also: Setting SPECCTRA Options

Translating Design Data from SPECCTRA to PADS Layout

Use the From SPECCTRA dialog box to translate design data modified by SPECCTRA back into a .pcb design file.

1. Use Windows Explorer to navigate to your PADS program files directory, and double-click **pads2sp.exe**.
2. In the SPECCTRA Link dialog box, click **From SPECCTRA**.
3. To indicate the routing information file that is returned by SPECCTRA after processing, type or browse to the location in the **SPECCTRA Routes** box. Include the command to write this file after autorouting at the end of the .do file.
4. To indicate the placement and routing information file, type or browse to the location in the **Session File** box. You do not need to supply this file name if you did not use any of the SPECCTRA placement capabilities.
5. To indication the original (source) .pcb file, type or browse to the location in the **Original PCB File** box.
6. To indicate the file to be created from the SPECCTRA file, type or browse to the location in the **New PCB File** box.
7. To set options for sending via keepout information, passing advanced rules, setting a trace arc translation mode, or returning unused pins net, click **Options**.

   See also: Setting SPECCTRA Options

Setting SPECCTRA Options

The Options dialog box appears when you click the Options button on the SPECCTRA Link dialog box, the TO SPECCTRA dialog box (stand-alone), or the FROM SPECCTRA dialog box (stand-alone).

This dialog box controls options for sending via keepout information, passing advanced rules to SPECCTRA, setting a mode for trace arc translation, and returning the unused pins net from SPECCTRA.

See also: Unused Pins Net in the *Concepts Guide*

1. On the **File** menu, click **Export.**

2. In the File Export dialog box, in the Save as type list, click **SPECCTRA Files (*.do).**
3. Browse to overwrite a file or type a new file name. Click **Save.**
4. In the SPECCTRA Link dialog box, click **Options**.
5. To send via keepout areas from your decals to SPECCTRA, select the layer you want from the **Layer Containing Via Keepout Shapes** list.

   Tip: PADS Layout fully supports keepouts in the Layout Editor. The preferred method to create a via keepout is to define keepouts in your decals.

6. To pass default Selected Layer and Selected Via rules to SPECCTRA, click **Pass Default Advanced Rules**. These rules require the Advanced Rules option in SPECCTRA. Turn this option on if you have this SPECCTRA option; otherwise, leave this option off.

   See also: PADS Layout to SPECCTRA Rules Conversion *in the Concepts Guide*.

7. Select from one of three modes to perform trace arc translation:

   To Single Segment—Replaces each trace arc with a single segment. This is the default mode.

   To Multiple Segments—Replaces a trace arc with multiple segments. The original trace arc is divided into smaller arcs (equal to approximately 5 degrees) and then each smaller arc is replaced by a single segment. The result is a polyline of multiple segments instead of the arc.

   To Quarter Arcs (QARCs)—Breaks existing arcs into quarter arcs and other segments. (Quarter arcs are arcs whose start and end points are exactly 0–90, 90–180, 180–270, and 270–360 degrees.) The quarter arcs are translated to the SPECCTRA QARC structure. The remaining parts of arcs are translated to polylines.

8. To return unused pin and fanout information to PADS Layout, click **Return UNUSED_PINS routing to PADS Layout**. Type the name of the net in the PADS Layout design that will contain the unused pins. Provide a new name if you do not want to use the default.

   Clear this option to ignore unused pin and fanout information when returning to PADS Layout.

Tip: SPECCTRA names the unused pins net +UNUSED_PINS+ while previous versions named it *UNUSED_PINS*. The SPECCTRA Link interprets both names.

The maximum netname length in PADS Layout is 47 characters. You can use any alphanumeric characters except for brackets{ }, asterisks *, or spaces.

Setting the SPECCTRA Automatic Startup Information

Use the SPECCTRA Setup dialog box to set SPECCTRA automatic startup information.

1. On the **File** menu, click **Export.**
2. In the File Export dialog box, in the Save as type list, click **SPECCTRA Files (*.do).**
3. Browse to overwrite a file or type a new file name. Click **Save.**
4. In the SPECCTRA Link dialog box, click the **Setup** button.
5. To indicate the executable needed to run SPECCTRA, type or browse to the location in the **Executable** box.
6. To indicate the password file needed to run SPECCTRA, type or browse to the location in the **Password/Server** box.

   Tip: For teal key node-locked or floating licensing, point to your license server in the standard port@host format. For example, 7508@myserver.
7. To indicate the SPECCTRA message output file, type or browse to the location in the **Message Output** box.
8. To indicate the SPECCTRA status file, type or browse to the location in the **Status** box.
9. To indicate the SPECCTRA color mapping file, type or browse to the location in the **Color Mapping** box.
10. To disable the SPECCTRA graphic display and make SPECCTRA run faster, click **No Graphics**.
11. To close SPECCTRA after it processes the .do file commands, click **Quit After Do**.
12. To delete all prerouted traces before entering SPECCTRA, click **No Preroutes**.
13. To ignore copper without net assignments, which have no net association in SPECCTRA, click **Don't Strip Orphan Shapes**.
14. To convert one-inch square, or smaller, polygons to simple rectangles, click **Simplify Polygons**.
15. Select the licensing type you want: Floating, Node-locked with Flexid Key, or Node-locked with SSI key.
16. To reload the routed design back into PADS Layout after it processes the .do file commands, click **Launch PADS Layout session**.

Passing Keepouts to SPECCTRA

You can define various types of keepouts in PADS Layout, such as placement, trace, and via keepouts. The SPECCTRA Link passes them to SPECCTRA automatically: placement keepout (as place_keepout), trace keepout (as wire_keepout), and via keepout (as via_keepout).

Do Files

Creating or Editing a .do File

The .do file is an editable batch script file, which controls SPECCTRA operation. You can add or edit command lines in a .do file. When you start the editor, the .do file you specify in the SPECCTRA Link dialog box is read for editing.

To create or edit a .do file:

1. On the **File** menu, click **Export.**
2. In the File Export dialog box, in the Save as type list, click **SPECCTRA Files (*.do).**
3. Browse to overwrite a file or type a new file name. Click **Save.**
4. In the SPECCTRA Link dialog box, do one of the following:
    - To create a new file, type the name of the new .do file in the **DO File Name** box and click **DO File**.
    - To edit an existing file, type or browse to the name of the .do file in the **DO File Name** box and click **DO File**.
5. The contents of the **Setup** and **All** areas change according to the options you select in the Setup area. Select the options you want and then click **Apply to Editor** to write the information to the .do file. The commands appear in the Editor area.
    - **Order** -change the original net ordering and control whether nets are routed in daisy-chain or starburst fashion.

      See also: “Order” and “Choosing Starburst or Daisy-Chain Wiring” topics in the *SPECCTRA Help*
    - **Action** - control wire rerouting, net routing, and the availability of connections, vias, and layers for autorouting.

      See also: “Protect/Unprotect”,” “Fix/Unfix,” and “Select/Unselect” topics in the *SPECCTRA Help*
    - **Cost** - control routing costs and override the autorouter internal cost table.

      See also: “Cost,” “Limit,” “Tax,” and “Using Standard Autorouting Commands” topics in the *SPECCTRA Help*

      If you click Cost in the Cost area and Layer in the All area, options in the Type area appear.

      See also: “Type,” “Length,” “Way (Cost),” and “Way (Limit)” topics in the *SPECCTRA Help*

- **Test Point Rule** - control passing DFT Audit test point placement options to SPECCTRA for its test point placement routine.

  See also: Passing DFT Audit Settings to SPECCTRA in the *Concepts Guide* and the “Testpoint” and “Testpoint Antennas” topics in the *SPECCTRA Help*.

Table 43-1. SPECCTRA Link Dialog Box Options

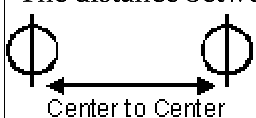
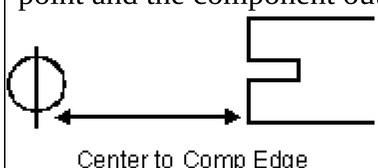
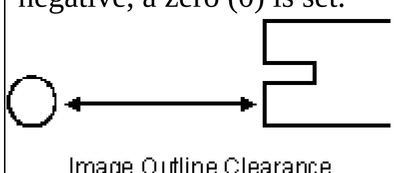
Option	Function
Insert Test Points	Allows insertion of test points and makes the options in the Test Points area available.
Allow Points at Pins	Allows placement of test points on component pins. There is no equivalent in DFT Audit; test points are always allowed on pins.
Allow Antennas	Allows antennas. Set a length. There is no equivalent in DFT Audit.
Max Length	Sets the length restriction for antennas. The default maximum length is negative one (-1), no length restriction.
Center to Center	The distance between the centers of test points. 
Center to Comp Edge	The distance between the center of the test point and the component outline. 
Image Outline Clearance	The clearance between the component outline and the test point carrier (via or component pin). If the Image Outline Clearance is negative, a zero (0) is set. 
Test Side	Searches the specified side for test point placement.
Use Via	Uses vias as test points.

Table 43-1. SPECCTRA Link Dialog Box Options (cont.)

Option	Function
Grid X, Y	The test point grid. **See also:** Grid Options

- **All area** - Limits actions to certain selected objects. The content of this area changes depending on the options you select in the Setup area.
- **Only area** - Limits actions to certain selected objects.

6. Use the **Routing** area to set fanout rules: direction, pin type, and maximum length, and to set the **Bus** direction and enter the number of **passes** for each type.
7. To autoroute your design based on how your design is converging, click **Smart Route**. For more information see the *SPECCTRA Help*.
8. To write the commands to the .do file click **Apply to Editor**. The command appears in the Editor area.
9. Click **Startup** to set the startup information.

   See also: Setting the SPECCTRA Automatic Startup Information
10. Click **Finish** and set the parameters in the Setup SPECCTRA Finish dialog box.

    See also: SPECCTRA Output File Location and Router Settings
11. Click **Apply** to save the parameters.
12. Click **Save As** to save the file and enter a name for the file. This is optional.
13. Click **Continue** to start the conversion and loading process.

Setting up SPECCTRA .do File Startup Options

Use the Setup SPECCTRA Startup dialog box to include a line referencing previously entered routes, saved in Wires or Best Save files, in your .do file. SPECCTRA refers to these files upon startup. You can also include the name of the status file and parameters for Via at SMD, Seed Via, and Seed Via minimum distance.

For details on these files and functions see the SPECCTRA Design Language Reference PDF file spdlr.pdf in the SPECCTRA group.

1. On the **File** menu, click **Export.**
2. In the File Export dialog box, in the Save as type list, click **SPECCTRA Files (*.do).**
3. Browse to overwrite a file or type a new file name. Click **Save.**
4. In the SPECCTRA Link dialog box, click **DO File**.

5. In the SPECCTRA DO File dialog box, click **Startup**.
6. To indicate the wires file, type or browse to the location in the **Wires File** box.
7. To indicate the status file, type or browse to the location in the **Status File** box.
8. To indicate the Best Save file, type or browse to the location in the **Best Save** box.
9. To allow vias at SMD pads, select **On** in the Via at SMD area. To ensure that the via is on a grid point select **On** in the Grid area. To ensure that the via fits within the pad, select **On** in the Fit area.
10. To break up two-pin connections that are larger than a certain length, click **Seed Via** and type the length value in the **Distance** box.
11. Click **Apply**.

Related Topics

SPECCTRA help

Working with Split Planes

SPECCTRA and Split/Mixed Planes

If you plan to use SPECCTRA to fanout or otherwise route your power nets to their associated plane area polygons, read this important information before you define your plane polygons in PADS Layout.

During the development of the split/mixed plane features in PADS Layout, certain operational details were discovered about the way that SPECCTRA routing commands respect plane polygons. To achieve proper attachment of power nets to plane polygons in SPECCTRA, use one of the following split plane definition procedures. Following these steps will ensure the highest possible quality routing results from SPECCTRA.

- If you typically define split planes after you route your designs in SPECCTRA, see Defining Split Planes After Routing in SPECCTRA.
- If you typically define split planes before you route your designs in SPECCTRA, see Defining Split Planes Before Routing in SPECCTRA.

Split/mixed plane layers without routing, named copper, or plane polygons are translated as power layers in SPECCTRA. Power layers are not considered routing layers by SPECCTRA; therefore, SPECCTRA cannot route on these layers. This minimizes the layer count passed to the router. For example, you can route a four-layer design with two power layers in a SPECCTRA configuration licensed for two routing layers.

SPECCTRA regards the entire plane as the area in which to connect component pins to all plane nets. The SPECCTRA fanout and route commands connect SMD component pins by routing short traces from the pins to vias to satisfy a connection to the plane.

For best results:

- Perform a multipass fanout of power pins before you execute multiple route passes by inserting the number of fanout passes in the command. For example, change “fanout (pin_type power)” to “fanout 5 (pin_type power).”
- Select the proper fanout options as defined in Creating or Editing a .do File.
- Avoid assigning design rules to nets that are associated with split/mixed plane layers. The design will not open in SPECCTRA if design rules are present. The SPECCTRA Link automatically removes or ignores design rules associated with split/mixed plane layers.

Related Topics

Creating or Editing a .do File

Defining Split Planes Before Routing in SPECCTRA

When you can define your split planes before routing, you can use the advanced functionality of PADS Layout and SPECCTRA to quickly route your split plane designs. Use the following steps to take advantage of this functionality:

1. Assign layers as split/mixed layer type: Before you can add split plane nets and data to a layer, you must assign the layer as split/mixed.
   See also: Layers Setup dialog box, Plane Layer Nets Dialog Box

   For best results, limit the assignment of split/mixed to internal, embedded plane players. Assigning external routing layers as split/mixed layers may produce unexpected results.

2. Define the plane polygons: Create split plane polygons for each net assigned to the plane layer using the steps described in “Creating a Plane Area” and “Auto Separate.”

   With PADS Layout you can define overlapping plane polygons, but Plane Connect detects overlaps and automatically adjusts the plane fill area to eliminate overlaps. SPECCTRA does not offer features to automatically adjust overlapping polygons and may produce unexpected results when large numbers of overlapping polygons are translated.

   For best results:

   - Define your planes using the Auto Separate command only. This eliminates the possibility of overlapping plane areas or plane area cutouts.

- If you prefer to create planes as polygon rectangles and circles, create the smaller polygons first and then add polygons around them rather than creating a larger polygon and embedding a smaller polygon within it.

3. Route the design in SPECCTRA: Once you assign the proper layer and net data for the plane layer, you can transfer the design to SPECCTRA.

   See also: Loading In and Out of SPECCTRA Automatically

4. Flood the split/mixed plane polygons: After routing is completed in SPECCTRA and the design returns to PADS Layout, flood the plane polygons.

5. Verify plane net continuity: After flooding the planes, verify the continuity of the plane nets. This process scans the plane polygons and route data and reports portions of nets disconnected from the plane polygons. To do this, see "Verifying the Design", "Setting Up Plane Checking", "Creating a Plane Area" and "Setting Up Plane Checking."

Defining Split Planes After Routing in SPECCTRA

When your design process involves splitting planes after you route the design in SPECCTRA, use the following steps in PADS Layout:

1. Assign layers as Split/Mixed Plane Layer Type: Before you can add split plane nets and data to a layer, you must assign the layer as split/mixed.
   See also: Layers Setup dialog box, Plane Layer Nets Dialog Box

   For best results:

   - Do not define a plane area polygon for the split/mixed layer prior to autorouting in SPECCTRA. SPECCTRA considers that the entire plane layer belongs to all nets and provides short fanout traces for SMD pins connected to the plane nets.

   - Limit the assignment of split/mixed to internal, embedded plane layers. Assigning external routing layers as split/mixed layers may produce unexpected results.

2. Route the design in SPECCTRA: Once you assign the proper layer and net data for the plane layer, you can transfer the design to SPECCTRA.

   See also: Loading In and Out of SPECCTRA Automatically

3. Define the plane polygons: Create split plane polygons for each net assigned to the plane layer using the steps described in "Creating a Plane Area and "Auto Separate."

   Tip: If you want to pass this design to SPECCTRA again after you define split/mixed planes, see the information below. Also see Routed Traces on PADS Layout Split/Mixed Plane Layers in the *Concepts Guide*.

   With PADS Layout you can define overlapping plane polygons, but Plane Connect detects overlaps and automatically adjusts the plane fill area to eliminate overlaps. SPECCTRA, however, does not offer features to automatically adjust overlapping polygons. SPECCTRA may produce unexpected results when large numbers of overlapping polygons are translated.

For best results:

- Define planes using Auto Separate only. This eliminates the possibility of overlapping plane areas or plane area cutouts.
- If you prefer to create planes as polygon rectangles and circles, create the smaller polygons first and then add polygons around them rather than creating a larger polygon and embedding a smaller polygon within it.

4. Flood the split/mixed plane polygons: After routing is completed and the design returns to PADS Layout, flood the plane polygons.
5. Verify the plane net continuity: After flooding the planes, verify the continuity of the plane nets. This process scans the plane polygons and route data and reports portions of nets disconnected from the plane polygons. To do this, see “Verifying the Design”, “Setting Up Plane Checking”, “Creating a Plane Area” and “Setting Up Plane Checking.”

Chapter 44
Crash Detection, BMW and BLT

Crash Detected Dialog Box

The Crash Detected dialog box opens at a crash and allows you to save a report of the PADS environment as well as pertinent files into a compressed PADS Dump File. You can then submit this file to Mentor Customer Support. You can attach feedback to this report, and optionally, the BMW media and project files.

Accessing

- This dialog box is inaccessible unless the software crashes and crash detection is enabled in the software .ini file.

Enabling/Disabling the Crash Detected Dialog Box

This functionality is turned off by default and is controlled by a switch in the .ini file.

- If no switch exists in the .ini file or if, in the [General] section of the .ini file, the switch exists with a value of 0 (zero), then crash detection is turned off. No report is created of the environment at the time of the crash.
- If, in the [General] section of the .ini file, the *CrashDetection* switch exists with a value of 1, then crashes are detected and the Crash Detection dialog box appears.

BMW and BLT

BMW (Basic Media Wizard) and BLT (Basic Log Test) are tools that you can use to record and play back PADS Logic, PADS Layout and PADS Router sessions. They are particularly useful as a means of supplying information to PADS Technical Support engineers trying to identify and resolve any problematical behavior you may encounter.

If you report problematical behavior for one of the PADS tools to PADS Technical Support, Tech Support engineers may ask you to use BMW to record *session playback media documenting the actions that caused the problem*. PADS tech support engineers can then replay the session with BLT to help them identify and resolve the problem.

Creating Session Playback Media With BMW

To create session playback media, BMW session logging must be enabled when the problem occurs. If session logging was not enabled when you encountered the problem, you must recreate the actions that caused the problem in a new session with session logging enabled.

Also, depending on whether the problem you want to document caused the PADS tool to crash, you can create session playback media based on either the current or the immediately previous PADS tool session.

The following table specifies which of the procedures described below you must use to create session playback media.

Was logging enabled?	Did the PADS tool crash?	Then use this procedure.
Yes	No	Creating Session Playback Media For a Normal Session
No	No	Creating Session Playback Media For a Normal Session
Yes	Yes	Automatically Creating Session Playback Media for a Crashed Session
No	Yes	Manually Creating Session Playback Media For a Crashed Session

Creating Session Playback Media For a Normal Session

Use this procedure when the session you are recreating did not cause a PADS tool crash.

To create session playback media from the current session:

1. Start the PADS tool.
2. Type the modeless command **BMW ON** and press **Enter to enable session logging**. Logging remains enabled for this and all future sessions until you disable it with the **BMW OFF** command.

   Tip: You *must* enable session logging *before* you open the file.
3. Open the file in which you encountered the problematical behavior.
4. Perform the series of actions that produced the problematical behavior. The series of actions, as well as changes to the board or to the configuration, are stored in the Session Log Files for the current session.
5. Type the **BMW modeless command** and press **Enter.**
6. In the Media Wizard dialog box, click **Create Media from Current Session**.

7. Type your initials in the **User Initials** box. (They are included in the playback media filenames to identify the files as yours.)

8. To delete all entries in the session log file between the first Open and the last Save command, click **Delete Actions Before Last Save**. You can do this to eliminate any actions you may have performed before beginning the series of actions that produced the problematical behavior. This makes it easier for the Tech Support engineer to identify the problem.

9. Click **OK** to create the Session Media Files.

Automatically Creating Session Playback Media for a Crashed Session

Use this procedure when the session you are recreating caused the PADS tool to crash, and none of the listed restrictions applies.

Restrictions:

- This procedure works only if:
  - The previous (crashed) session started with BMW logging already enabled.
  - Logging remained enabled throughout the session.
- This procedure does not give useful results if any additional instance of the PADS tool ran concurrently (for any period) with the previous (crashed) session.

To automatically create session playback media from the previous session:

1. Start the PADS tool.
2. Type the modeless command **BMW ON** and press **Enter to enable session logging**. Logging remains enabled for this and all future sessions until you disable it with the **BMW OFF** command.

   Tip: You *must* enable session logging *before* you open the file.
3. Open the file in which you encountered the problematical behavior.
4. Perform the series of actions that produced the problematical behavior. The series of actions, as well as changes to the board or to the configuration, are stored in the Session Log Files for the current session.
5. After the crash, restart the PADS tool. A dialog box is displayed asking if you want to save media files for the crashed session. Click **Yes to create the** Session Media Files.

Manually Creating Session Playback Media For a Crashed Session

Use this procedure when the session you are recreating caused the PADS tool to crash, and the automatic procedure described in Automatically Creating Session Playback Media for a Crashed Session cannot be used due to one of the restrictions listed in that section.

To manually create session playback media from the previous session:

1. Start the PADS tool.
2. Type the modeless command **BMW ON** and press **Enter to enable session logging**. Logging remains enabled for this and all future sessions until you disable it with the **BMW OFF** command.

   Tip: You *must* enable session logging *before* you open the file.
3. Open the file in which you encountered the problematical behavior.
4. Perform the series of actions that produced the problematical behavior. The series of actions, as well as changes to the board or to the configuration, are stored in the Session Log Files for the current session.
5. After the crash, restart the PADS tool.
6. Type the shortcut **BMW** and press **Enter.**
7. In the Media Wizard dialog box, click **Create Media from Previous Session**.
8. Type your initials in the User Initials box to identify your session playback media files.
9. Click **OK** to create the Session Media Files.

Session Log Files

Whenever BMW session logging is enabled, two sets of session log files are maintained in the \PADS Projects folder. These logs record actions performed in the current session, and in the immediately previous session.

BMW names these files as follows:

Current Session Log Files	**Previous Session Log Files**
<*pads_tool*>_Next.log	<*pads_tool*>_NextBak.log
<*pads_tool*>_Next.reg	<*pads_tool*>_NextBak.reg
<*pads_tool*>_Next.ini	<*pads_tool*>_NextBak.ini

Tip: *These files are dynamic*; each time you start a session, the current session log files are renamed as the previous session log files, and new current session log files are created. The contents of the old previous session log files are lost.

Whenever you elect to create session media files for a session, the appropriate set of these log files is saved in a permanent location, as described in Session Media Files.

Note: You may see a log file named <*pads_tool*>_Session.log listed in the \PADS Projects folder. This file is unrelated to the session playback media created by BMW.

Session Media Files

Each time you create session playback media, BMW creates a new session media folder in the \PADS Projects folder, and copies into it:

- The .pcb or .sch file for which you are recording the session
- The Session Log Files for the session. BMW then renames these files based on the session media folder name.

The session media folder is named <month><day><initials><sequential letter>, where:

- <month><day> is the date.
- <initials> are letters you type in the Media Wizard dialog box to personalize the media files.
- <sequential letter> is a letter automatically assigned to sequence the directories created on a specific date.

Example: \PADS Projects\0530jsb represents a session media folder created on May 30, using the initials js, and that was the second session media folder created on that day.

When creating the session playback media, the following files are written to the session media folder:

<*folder_name*>.log	The session log
<*folder_name*>.reg	The program-specific settings in the Windows registry
<*folder_name*>.ini	The configuration file (.ini file)
<*folder_name*><*sl*>.pcb (if needed) <*folder_name*><*sl*>.sch (if needed) Where <*sl*> is a sequential letter for multiple files.	Files related to <*folder_name*>.log. These files are automatically renamed and added to the session media folder.

Replaying Session Playback Media with BLT

Use BLT to replay session playback media created by BMW.

To replay a session:

1. Type the **BLT** modeless command, and press **Enter**.
2. Select the session playback media from the Media Directories list and click **OK**.

Result: The session is replayed.

Tip: To personalize the media folder and session playback media file names, select a media session from the Media Directories list, type the new name into the New name box, and then click **Rename**.

The /BMW Command Line Switch

If you want BMW to automatically prompt you to create media from the previous session each time you start a PADS tool, open the PADS tool using the /BMW command line switch. Or use /BMW-*xx* (where *xx* represents your initials, which are used in folder and file names to identify them as yours).

Restriction: When you use BMW as a command line option, it only creates media of the previous session; use the BMW modeless command to create media of your current session.

Chapter 45
PADS Layout GUI Reference

The section contains information on all of the GUI elements in PADS Layout.

Add Archive to Vault Dialog Box

Use the Add Archive to Vault dialog box to configure an Archive Navigator archive and save it to the vault.

Tip: From this dialog box you can also access the Templates dialog box, where you can define customized archive templates.

Accessing

- **Working Folder** view > **Add Archive to Vault button**

Figure 45-1. Add Archive to Vault Dialog Box

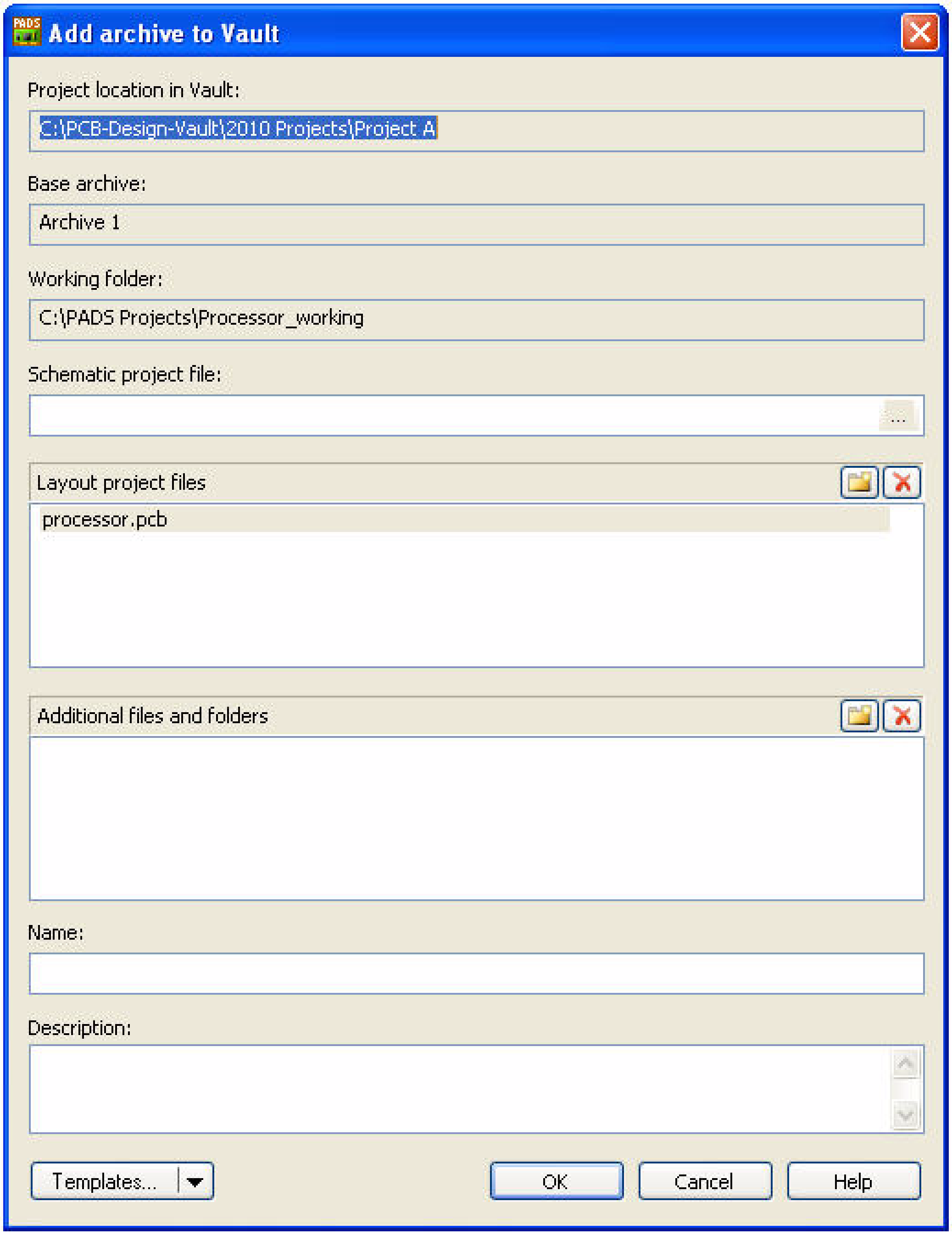


Table 45-1. Add Archive to Vault Dialog Box contents

Name	Description
Project location in Vault	The project in the vault view tree where the new archive will be created.
Base archive	The existing archive that the new archive will be based on, that is, the archive currently restored to the working folder. **Tip:** In the vault view, the archive currently restored to the working folder is shown in green.
Working folder	The pathname of the current working folder.
Schematic project file	Specifies the DxDesigner .prj file to be stored in the new archive.
Layout project files	Specifies the PADS Layout .pcb files to be stored in the new archive.
Additional files and folders	Specifies other files and folders to be stored in the new archive.
Name	Specifies a description attribute for the new archive. **Tip:** Create a name you can search for with the Find in Vault tool.
Description	Specifies a description attribute for new archive. **Tip:** Create a name you can search for with the Find in Vault tool.
Templates	Click the down arrow to select from a list of choices: **Select**—Opens the Templates dialog box, where you can select a fileset configuration for the new archive from a list of existing templates. **New**—Opens the Templates dialog box, where you can create a new archive configuration template. **Edit**—Opens the Templates dialog box, where you can edit a template.

Related Topics

Adding an Archive to the Vault

Add BGA Pin Labels Dialog Box

Use the Add BGA Pin Labels dialog box to label the die part's substrate bond pad. Usually labels match the name of the BGA pin to which they are connected.

Tips:

- You can select the pads individually, by group, or by die part.
- Selecting a die part lists all pins in the Connection multicolumn list.
- The substrate bond pad of the selected die part is highlighted in the **Die Pin#** column. The BGA pin labels are listed in the BGA Pad column.

Accessing

- **BGA Toolbar button** > **Wire Bond Diagram button** > select a **BGA pin**

Figure 45-2. Add BGA Pin Labels Dialog Box

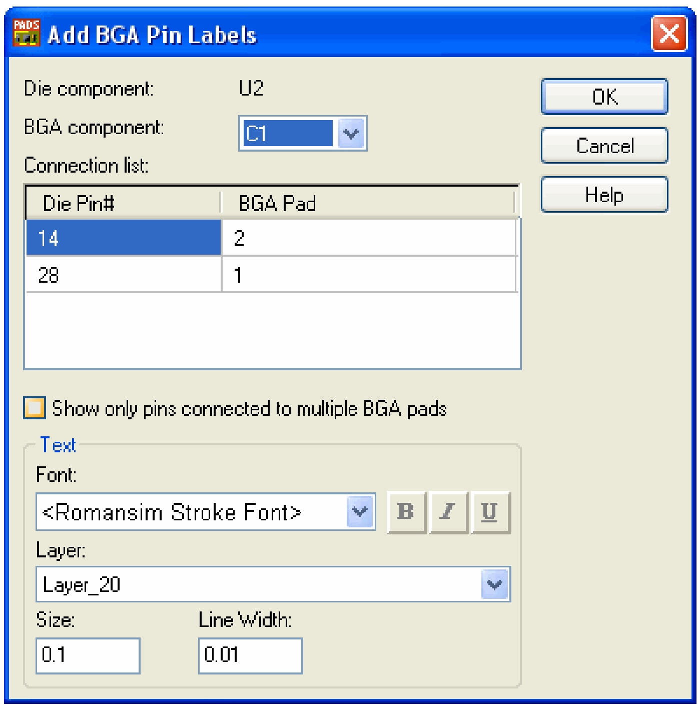


Table 45-2. Add BGA Pin Labels Dialog Box contents

Name	Description
Die Component	Displays the component name.
BGA Component	Lists the BGA components available.
Connection list	• **Die Pin #**—Displays the substrate bond pad of the selected die part. • **BGA Pad**—Lists the BGA pin labels. **Tip:** Double-click to edit the label.

Table 45-2. Add BGA Pin Labels Dialog Box contents (cont.)

Name	Description
	Opens the Pads for Die Pin Dialog Box. **Restriction:** Available only after double-clicking in the BGA Pad cell of the Connection list.
Show only pins connected to multiple BGA pads	Specifies to display only die pins that are connected to multiple BGA pin pads in the **Connection** list.
Font	The fonts available to you. **Tips:** • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.
Layer	The layers available to you on which to place the text.
Size	Specifies the size of the font. **Size (pts):** This is font size in points and appears for system fonts **Size (mils):** This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters. 
Line Width	Specifies the line width for stroke fonts only. 

Related Topics

To Add a BGA Pin Label

Add/Edit CAM Document Dialog Box

Use the Add or Edit Document dialog box to define a CAM document. The only difference between the two dialog boxes is that the Edit CAM Document dialog box opens populated with the information from the selected document, while the ADD CAM Document dialog box opens empty.

Accessing

- **File Menu > CAM > Add button**

  or

- **File Menu > CAM > Select a document name > Edit button**

Figure 45-3. Add/Edit Document Dialog Box

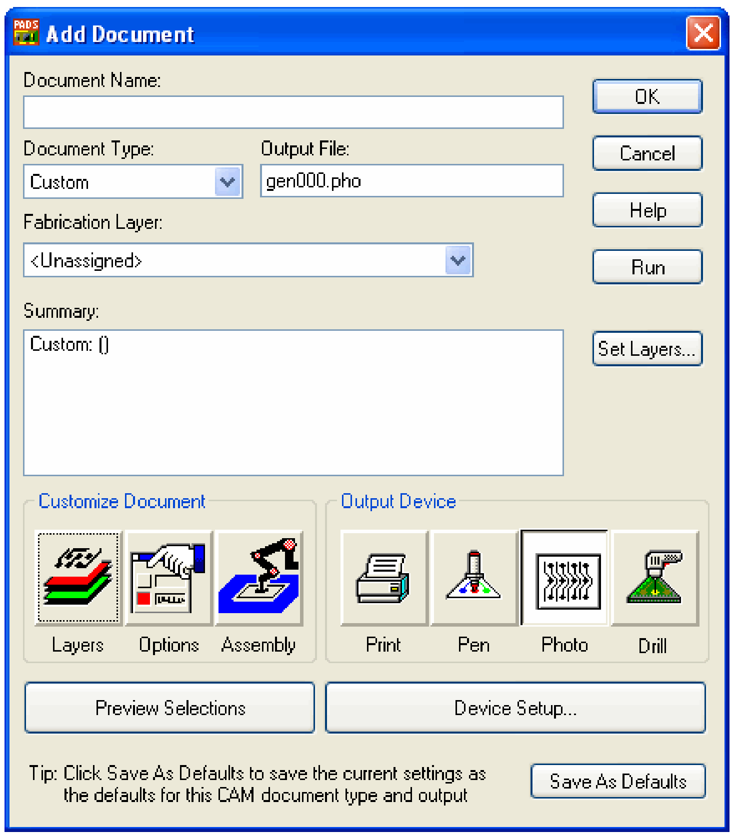


Table 45-3. Add/Edit Document Dialog Box contents

Name	Description
Document Name	The name of the CAM document.

Table 45-3. Add/Edit Document Dialog Box contents (cont.)

Name	Description
Document Type	When you add a CAM document, you must select the type of CAM document that you want to create. Each CAM document type has a specific set of options. **Tip:** The default settings of the CAM document types may not suit your decal layer usage. Always verify the document in the CAM Preview window to ensure that all necessary design elements are displayed for the type of document you are generating. • **Custom**—Create your own definition for a document. Use the Select Items Dialog Box to specify which objects appear in the document. • **CAM Plane**—Document CAM Plane layers. CAM planes are always solid copper planes in PADS Layout. To improve visibility in the work area, CAM planes are negative images and take on the background color of PADS Layout. In the CAM Preview window options, you can invert the negative image of the CAM document. By default, conductive elements appear white in the CAM Preview window. Layer 25 (125 in extended layer mode) is frequently used for CAM plane objects where added copper is the opposite - not copper (negative layer). Layer 25 can also be used for defining the oversize of thermals and antipads (an old method), however, the CAM Plane custom thermal setting in the Plot Options dialog box is a better method. Use the Select Items Dialog Box to add layer 25 if needed. • **Routing/Split Plane**—Document layers that contain plane areas and/or routing. Use this type for Split/Mixed and No Plane layers. Conductive elements appear black in the CAM Preview window. • **Silkscreen**—Document top and bottom silkscreen layers. Layer 20 (120 in extended layer mode) is frequently used for placement/nudge outlines as an alternative to the silkscreen layer outline. Use the Select Items Dialog Box to add layer 20 if needed. Silkscreen items appear black in the CAM Preview window. • **Paste Mask**—Document top and bottom paste mask layers. Black areas are paste locations. White is the paste mask. Paste is only used for surface mount components. Pads of through-hole components are not shown. • **Solder Mask**—Document top and bottom solder mask layers. Black areas are solder locations. White is the solder mask.

Table 45-3. Add/Edit Document Dialog Box contents (cont.)

Name	Description
Document Type (con't)	• **Assembly**—Document assembly drawings. Component outlines may have been created on the top layer, silkscreen layer, and/or layer 20. Use the Select Items Dialog Box to add the layers you need. Assembly items appear black in the CAM Preview window. • **Drill Drawing**—Document drill locations. A drill table is added automatically and appears at a location specified in the Drill Drawing Options dialog box. • **NC Drill**—Generate an NC Drill file. This document is not intended for viewing. It contains the x and y location and drill size of each hole required in the design. • **Verify Photo**—View and verify any existing CAM document. The document to be viewed can be from a different design. Verify Photo supports the macros, aperture selection, and fill area commands of PADS Layout Gerber outputs. Verify Photo plots can only process RS-274-X Gerber files created by PADS Layout.
Output File	Specifies the name of the CAM output file. Accept the default name, or type your preferred name for the output file.
Fabrication Layer	The layer on which you will be using CAM350 for post processing. **Tip:** The Fabrication Layer is only used during conversion to the CAM350 database using **CAM350 Link**.
Summary	Displays a summary of the selected CAM document.
Layers button	Opens the Select Items Dialog Box.
Options button	Opens a dialog box where you can set plot options depending on the plot type with which you're working: • Plot Options—Printer, Pen and photo plots • Drill Drawing Options—Drill drawings **Restriction:** Available from the Plot Options Dialog Box only when the Drill Drawing document type is selected. • NC Drill Options—NC drill output
Assembly button	Opens the Select Assembly Variant Dialog Box.
Print button	Specifies that you want the document to go to a printer when you click Run.
Pen button	Specifies that you want to use the Pen Plotter as your output when you click Run.
Photo button	Specifies that you want to use the Photo Plotter as your output when you click Run.
Drill button	Specifies that you want to want to use the Drill

Table 45-3. Add/Edit Document Dialog Box contents (cont.)

Name	Description
Preview Selections button	Opens the CAM Preview Dialog Box.
Device Setup button	Opens a dialog box where you can set options associated with the selected device type, Printer Setup, Pen Plot Setup, Photo plot Setup, or NC Drill Setup.
Save As Defaults button	Specifies to save your current settings as the defaults for this CAM document type and output device. **Tip:** This option is useful if you manually build your own drill table and want to use this drill table as the default drill table.
Run button	Produces the document you just defined.
Set Layers button	Opens the Layers Setup Dialog Box.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Add Chamfered Path Dialog Box

The Add Chamfered Path dialog box opens upon selection of the Chamfered Path - copper shape. You set Chamfered Path parameters before you add the copper to the design.

Accessing

- Drafting Toolbar > **Copper** button > Right-click > **Chamfered Path**

  Tip: You may need to click OK to switch off Design Rule Checking.

Figure 45-4. Add Chamfered Path Dialog Box

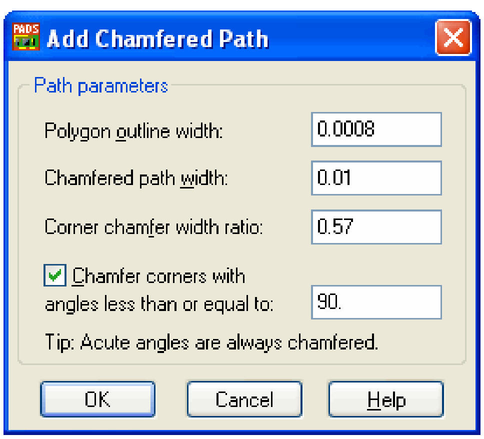


Table 45-4. Add Chamfered Path Dialog Box contents

Name	Description
Polygon outline width	Specifies the width value for the copper outline. **Tip:** Since copper is created with an outline and a fill, you can specify a very narrow outline width to achieve very sharp corners. Decrease the value for sharper corners and increase the value for more blunt corners. All corners are rounded with a radius equal to one half of the outline width.
Chamfered path width	Specifies the width value for the overall width of the copper path.
Corner chamfer width ratio	Specifies the ratio of the chamfered corner width to the chamfered path width. If the ratio is 1.0, the width of the chamfered corner is the same as the chamfered path. Reduce the ratio for a more narrow chamfered corner.
Chamfer corners with angles less than or equal to	Specifies an angle between 90 and 180 degrees as the upper limit beneath which all angles are chamfered. Outside corners less than 90 degrees are always chamfered. **Tip:** Click to clear the check box to only chamfer angles less than 90 degrees.

Related Topics

Creating Copper Chamfered Paths

Setting Chamfered Path Parameters

Add Class Tasks Dialog Box

See: Add Net Tasks/Add Class Tasks Dialog Box

Add/Edit Command Dialog Box

Use the Add Command dialog box to create commands that you can then use as selections on menus or as buttons on toolbars.

Accessing

- **Tools menu** > **Customize** > **Commands tab** > **New or Edit button**

Figure 45-5. Add Command Dialog Box

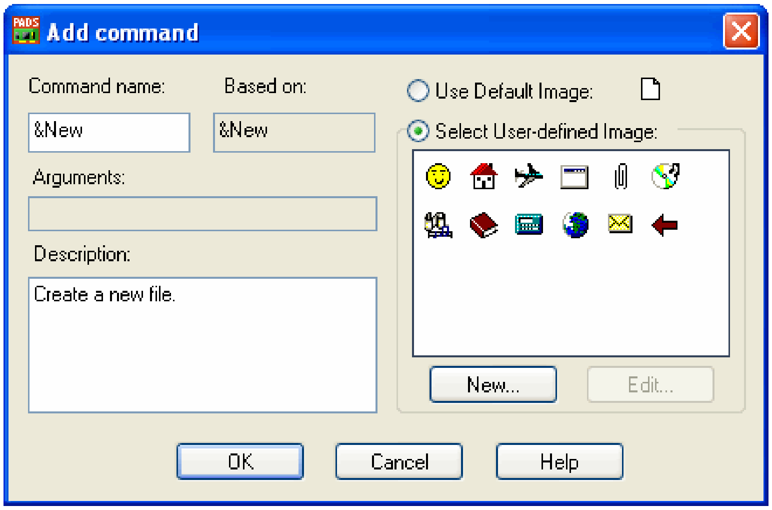


Table 45-5. Add Command Dialog Box Contents

Name	Description
Command name	The name of the new command. **Tip:** Type an ampersand before the letter you want to use as the Alt keyboard shortcut.
Based on	The command on which you want to base the new command.
Arguments	Any arguments for the new command. Tip: Use a space to separate arguments. If an argument contains a space, enclose the argument in quotation marks (““). **Restriction:** PADS Router only.
Description	Lists what the new command does.
Use Default Image	Use the recommended image.
Select User-defined Image	Select or create your own image to associate with the new command.
New	Open the Edit Button Image Dialog Box.
Edit	Open button in the Edit Button Image Dialog Box.

Related Topics

Creating a Custom Command

Creating a Custom Menu

Add Component Bond Pad Dialog Box

Use the Add Component Bond Pad dialog box to create new bond pads in your design.

Restriction: This information applies only to the BGA toolkit.

Accessing

- **BGA Toolbar** button > **Wire Bond Editor button** > **click the BGA** > **Right-click** > **Add CBP**

Figure 45-6. Add Component Bond Pad Dialog Box

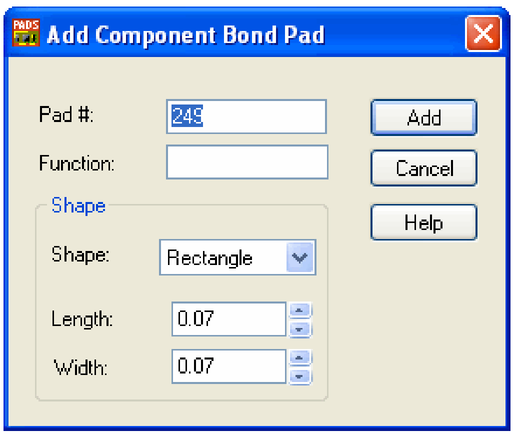


Table 45-6. Add Component Bond Pad Dialog Box contents

Name	Description
Pad #	Assigns a number to the currently selected component bond pad. By default, it is the same number as the substrate bond pad to which it is connected.
Function	Defines the function of the currently selected bond pad.
Shape list	Assigns a shape to the currently selected bond pad. Select **Rectangle** or **Oval**.
Length	Assigns a physical length, in current design units, for the currently selected bond pad.

Table 45-6. Add Component Bond Pad Dialog Box contents (cont.)

Name	Description
Width	Assigns a physical width, in current design units, for the currently selected bond pad.
Add button	Dynamically attaches the substrate bond pad and wire bond to the cursor for placement.

Related Topics

To Add Component Bond Pads

Add Die Parts Dialog Box

Use the Add Die Parts dialog box to import die parts from Library IQ into PADS Layout.

Restriction: This information applies to only the BGA toolkit.

Accessing

- **BGA Toolbar** button > **Add Die Parts** button

Figure 45-7. Add Die Parts Dialog Box

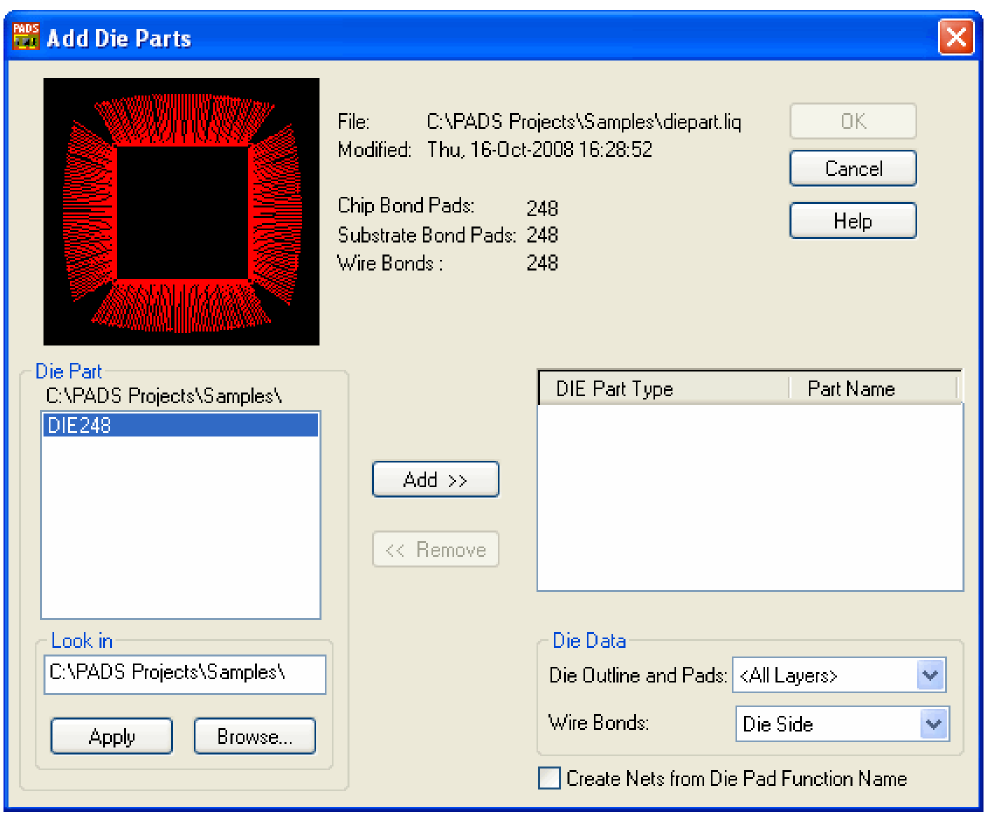


Table 45-7. Add Die Parts Dialog Box contents

Name	Description
Preview Area	Displays the Library IQ die part selected in the Die Part list.
File	The die part filename.
Modified	The last modification date.
Chip Bond Pads	The number of chip bond pads in the die part.
Substrate Bond Pads	The number of substrate bond pads in the die part.
Wire Bonds	The number of wire bonds in the die part.
Flip Chip	The Flip Chip identification for the die part.
Die Part list	Lists all Library IQ die parts in the current folder.
Look in	Displays the die part search folder. **Tip:** Click Browse to search for the die part folder.
Apply button	Sets the die part search folder.
Add >> button	Adds the selected Library IQ die parts to the DIE Part Type table.
<< Remove button	Removes the selected DIE Part from the DIE Part Type table.
Die Part Type column	The Library IQ die part to add.
Part Name column	The reference designator that is automatically assigned to the die part type. To change the reference designator, double-click on the part name and type a new name.
Die Outline and Pads	Sets the layer on which the die outline and pads appear. Select a layer from the list.
Wire Bonds	Sets the layer on which the wire bonds appear. Select a layer from the list.
Create Nets from Die Pad Function Name	Automatically creates nets for pins with matching pin name when you add die parts to PADS Layout. The net name matches the pin function.

Related Topics

To Add Die Parts from LIQ

Add Drafting Dialog Box

See: Drafting Properties Dialog Box

Add Free Text Dialog Box

Use the Add Free Text dialog box to add free text; text not belonging to another object. When you add text, there is an invisible bounding rectangle around the text itself.

Restriction: Text can only be added one line at a time. See the following topic for a tip on saving multiple lines of text to the library for reuse, Creating Reusable Fabrication Notes.

Accessing

- **Drafting Toolbar** button > **Text** button

Figure 45-8. Add Free Text Dialog Box

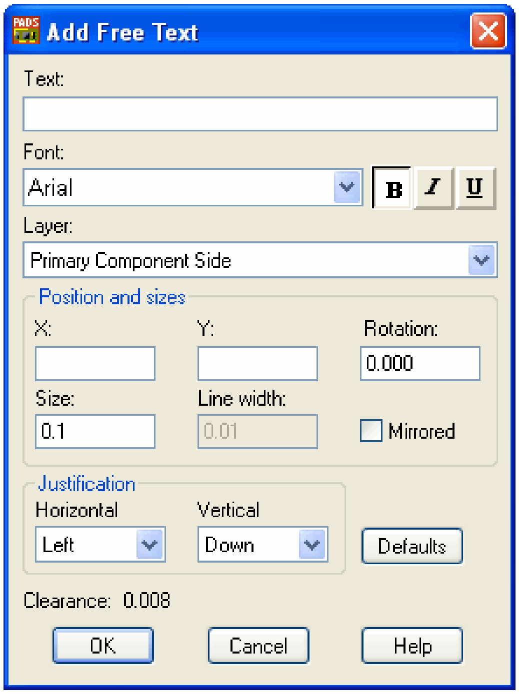
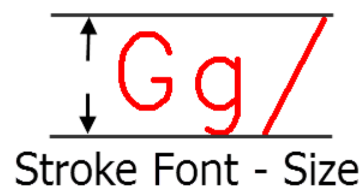


Table 45-8. Add Free Text Dialog Box contents

Name	Description
Text	The text string you want to use. **Tip:** There is a maximum of 128 characters per text string.
Font	The fonts available to you. **Tips:** • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.

Table 45-8. Add Free Text Dialog Box contents (cont.)

Name	Description
Layer	The layers available to you on which to place the text.
X,Y	Places the decal label in a specified location.
Rotation	Specifies the rotation angle of the label.
Size	Specifies the size of the font. **Size (pts):** This is font size in points and appears for system fonts **Size (mils):** This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters. 
Line Width	Specifies the line width for stroke fonts only. 
Mirrored	Flips the label - text is considered readable from the bottom side of the board.
Horizontal, Vertical	Sets the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change. **Tips:** • For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**. • Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.
Defaults	Restores the PADS Layout default settings.
Clearance	Specifies clearance values between the text and objects around it.

Related Topics

Adding Free Text

Add Net Tasks/Add Class Tasks Dialog Box

You can run specific electrodynamic checks on nets or classes.

Accessing

- **Tools** menu > **Verify Design** > **High Speed** check > **Setup button** > **Add Nets** button

  or

- **Tools** menu > **Verify Design** > **High Speed** check > **Setup button** > **Add Classes** button

Figure 45-9. Add Net Tasks/Add Class Tasks Dialog Box

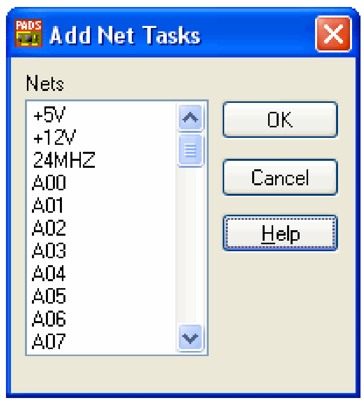


Table 45-9. Add Net Tasks/Add Class Tasks Dialog Box contents

Name	Description
Nets/Classes list	Lists the nets or classes available in the design.

Related Topics

Adding Nets or Classes for Specific High-Speed Checks

Add Net to Class Dialog Box

Use the Add Net to Class dialog box to create a collection of nets to which you can assign a common set of design rules.

Accessing

- Select one or more nets > **Right-click** > **Make Class**

Figure 45-10. Add Nets to Class Dialog Box

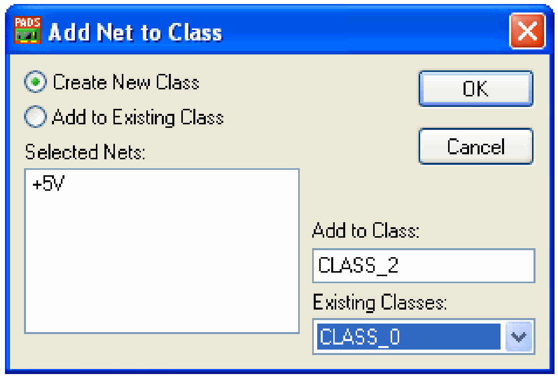


Table 45-10. Add Nets to Class Dialog Box

Name	Description
Create New Class	Specifies to make a new class for the selected nets.
Add to Existing Class	Specifies to add the selected nets to an existing class.
Selected Nets	Lists all nets selected in the design.
Add to Class:	Specifies the name of the new class. **Tip:** This is unavailable if Add to Existing Class is selected.
Existing Classes:	Lists the classes available in which to add the selected nets.

Related Topics

Creating Class Design Rules

Add New Attribute to Library Dialog Box

Use the Add New Attribute to Library dialog box to set name and value properties when adding new attributes to libraries.

Accessing

- **File** menu > **Library** > Library Manager dialog box > **Attr Manager** button > **Add Attr** button

Figure 45-11. Add New Attribute to Library Dialog Box

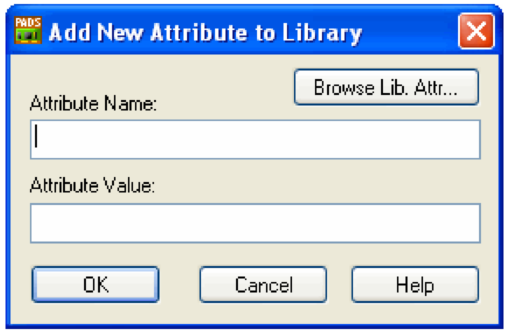


Table 45-11. Add New Attribute to Library Dialog Box Contents

Name	Description
Browse Lib. Attr	Opens the Browse Library Attributes Dialog Box.
Attribute Name	The name of the new attribute.
Attribute Value	The value of the new attribute.

Related Topics

Adding an Attribute to Multiple Library Items

Library Manager Dialog Box

Part Information Dialog Box, Attributes Tab

Add New Decal Label Dialog Box

Use the Add New Decal Label dialog box to create attribute labels for decals.

Accessing

- **Tools** menu > **PCB Decal Editor** > **Drafting** button > **Add New Label** button > select a decal

Figure 45-12. Add New Decal Label Dialog Box

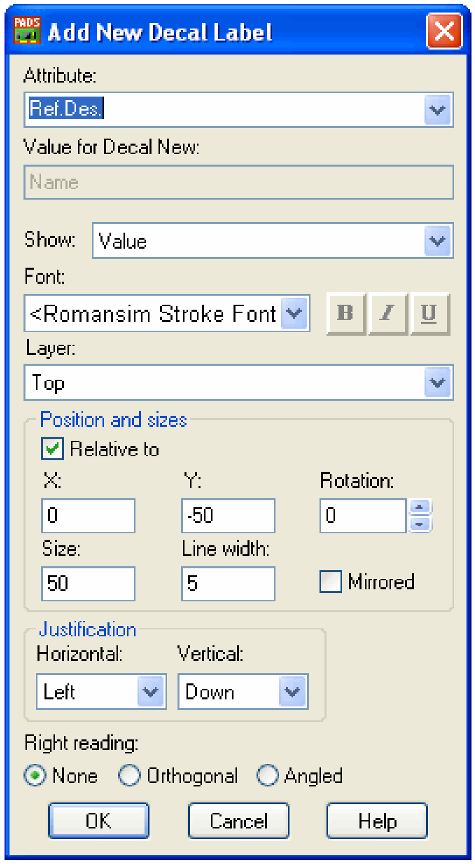


Table 45-12. Add New Decal Label Dialog Box Contents

Name	Description
Attribute	The attributes available to you. If you are creating labels for jumpers, Reference Designator is the only available attribute. **Tip:** Hidden attributes do not appear in the Attribute list unless the attribute was selected for the label before it was set as a hidden attribute.

Table 45-12. Add New Decal Label Dialog Box Contents (cont.)

Name	Description
Value for	The value of the selected attribute. **Tips:** • Unavailable if you selected Reference Designator or Part Type from the Attribute list, if the attribute is read-only, or if you open the Properties dialog box for labels of different attribute types. However, if the labels you select belong to attributes of the same type, you can edit this box. • If the attributes have different values, the box is blank. You can type a new value in the box to apply it to all of the selected attribute labels and their parent objects. • Value is also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.
Show	Controls the visibility of the label. • **None**—Turns visibility off. • **Value**— Displays only the label value. • **Name and Value**—Displays the name and value. • **Full Name and Value**—When labeling a structured attribute, displays the full structured name and value. **Tip:** Labels are invisible regardless of this setting unless you use the Display Colors Setup Dialog Box to change the color of labels to a color different from that of the background.
Font	The fonts available to you. **Tips:** • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.
Layer	The layers available to you.
Relative to	Places the label at the X and Y location relative to the component or the jumper. If you clear this check box, the label is placed at the X and Y location relative to the design origin.
X,Y	Places the decal label in a specified location.
Rotation	Specifies the rotation angle of the label.

Table 45-12. Add New Decal Label Dialog Box Contents (cont.)

Name	Description
Size	Specifies the size of the font. **Size (pts):** This is font size in points and appears for system fonts **Size (mils):** This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters. 
Line Width	Specifies the line width for stroke fonts only. 
Mirrored	Flips the label - text is considered readable from the bottom side of the board.
Horizontal, Vertical	Sets the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change. **Tips:** • For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**. • Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.
Right reading	Controls whether labels are readable (from left to right, or from bottom to top if the label is rotated). Click the **None**, **Orthogonal**, or **Angled** button to indicate the direction of reading you want.

Related Topics

Creating Attribute Labels in the PCB Decal Editor

Add New Part Label Dialog Box

Use the Add New Part Label dialog box to create attribute labels, part type labels, and reference designator labels for components or jumpers.

Tips:

- Reference designator is the only label available for use when you are creating labels for jumpers.
- If you don't set visibility information, default positions are used.

  See also: Label Defaults
- Unlike free text, when you add a label, there is *no* invisible bounding rectangle around the label itself.

Accessing

- **Select a part >** Right-click **> Add New Label**

Figure 45-13. Add New Part Label Dialog Box

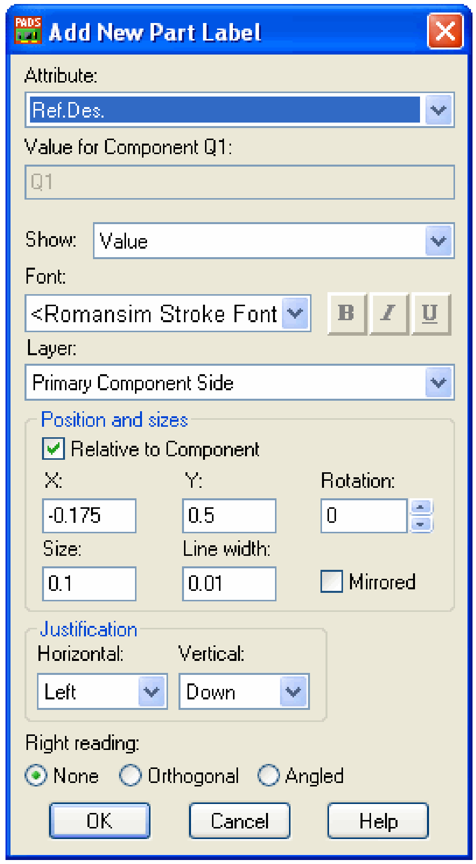


Table 45-13. Add New Label Dialog Box Contents

Name	Description
Attribute	The attributes available to you. If you are creating labels for jumpers, Reference Designator is the only available attribute. **Tip:** Hidden attributes do not appear in the Attribute list unless the attribute was selected for the label before it was set as a hidden attribute.

Table 45-13. Add New Label Dialog Box Contents (cont.)

Name	Description
Value for	The value of the selected attribute. **Tips:** • Unavailable if you selected Reference Designator or Part Type from the Attribute list, if the attribute is read-only, or if you open the Properties dialog box for labels of different attribute types. However, if the labels you select belong to attributes of the same type, you can edit this box. • If the attributes have different values, the box is blank. You can type a new value in the box to apply it to all of the selected attribute labels and their parent objects. • Value is also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.
Show	Controls the visibility of the label. • **None**—Turns visibility off. • **Value**— Displays only the label value. • **Name and Value**—Displays the name and value. • **Full Name and Value**—When labeling a structured attribute, displays the full structured name and value. **Tip:** Labels are invisible regardless of this setting unless you use the Display Colors Setup Dialog Box to change the color of labels to a color different from that of the background.
Font	The fonts available to you. **Tips:** • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.
Layer	The layers available to you.
Relative to	Places the label at the X and Y location relative to the component or the jumper. If you clear this check box, the label is placed at the X and Y location relative to the design origin.
X,Y	Places the decal label in a specified location.
Rotation	Specifies the rotation angle of the label.

Table 45-13. Add New Label Dialog Box Contents (cont.)

Name	Description
Size	Specifies the size of the font. **Size (pts):** This is font size in points and appears for system fonts **Size (mils):** This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters. 
Line Width	Specifies the line width for stroke fonts only. 
Mirrored	Flips the label - text is considered readable from the bottom side of the board.
Horizontal, Vertical	Sets the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change. **Tips:** • For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**. • Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.
Right reading	Controls whether labels are readable (from left to right, or from bottom to top if the label is rotated). Click the **None**, **Orthogonal**, or **Angled** button to indicate the direction of reading you want.

Related Topics

Adding a New Part Label

Add Pin Dialog Box

Use the Add Pin dialog box to select specific pins to add to the *Pin: Plated:* list in the Pad Stacks Properties dialog box.

Accessing

- **Setup** menu > **Pad Stacks** > select a Decal or Via > click the **Add** button under the *Pin: Plated:* box.

Figure 45-14. Add Pin Dialog Box

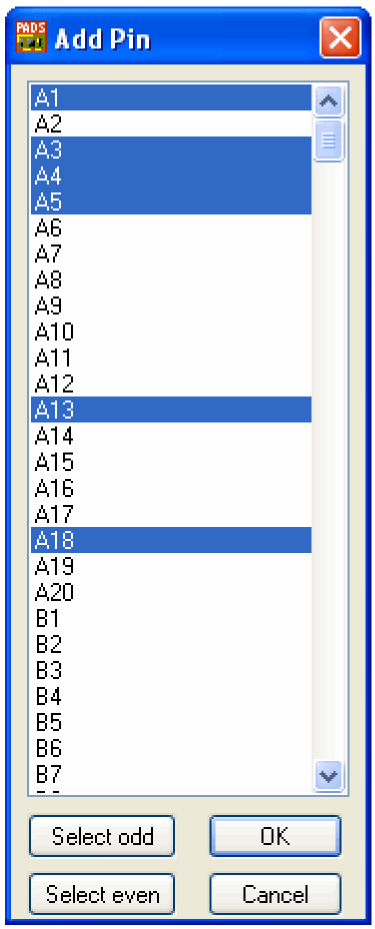


Table 45-14. Add Pin Dialog Box Contents

Name	Description
Pin list	Select the specific pin(s) you want to add to the *Pin: Plated:* list in the Pad Stacks Properties Dialog Box. The pins will be listed individually and can be deselected once added to the Pin: Plated: list if you choose not to give them the all the same pad stacks. • Ctrl+Click multiple items • Shift+click for a range of items • Click and drag for a range of items
Select odd/Select even	You can use these buttons as a shortcut to select all the odd or even pins in the Pin List.

Add Pin Pairs to Group Dialog Box

Use the Add Pin Pairs to Group dialog box to create a collection of pin pairs with a common set of design rules.

Accessing

- Select pin pairs > **Right-click** > **Make Group**

Figure 45-15. Add Pin Pairs to Group Dialog Box

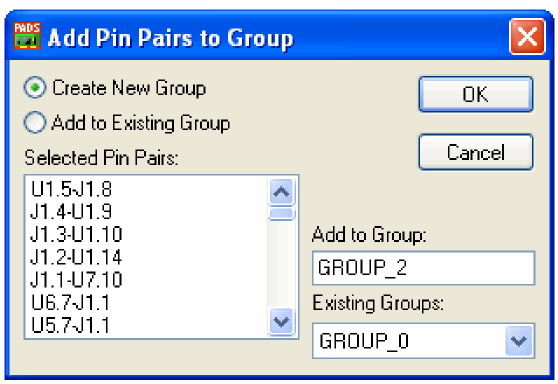


Table 45-15. Add Pin Pairs to Group Dialog Box

Name	Description
Create New Group	Specifies to make a new group for the selected pin pairs.
Add to Existing Group	Specifies to add the selected pin pairs to an existing group.

Table 45-15. Add Pin Pairs to Group Dialog Box (cont.)

Name	Description
Selected Pin Pairs	Lists all pin pairs selected in the design.
Add to Group:	Specifies the name of the new group. **Tip:** This is unavailable if Add to Existing Group is selected.
Existing Group:	Lists the groups available in which to add the selected pin pairs.

Related Topics

Creating Group Design Rules

Add Pins Dialog Box

Use the Add Pins dialog box to add pins to a part type.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > **New** button > **Pins** tab > **Add Pins** button

  or

- **File** menu > **Library** > select a Library > **Parts** button > select a part > **Edit** button > **Pins** tab > **Add Pins** button

Figure 45-16. Add Pins Dialog Box

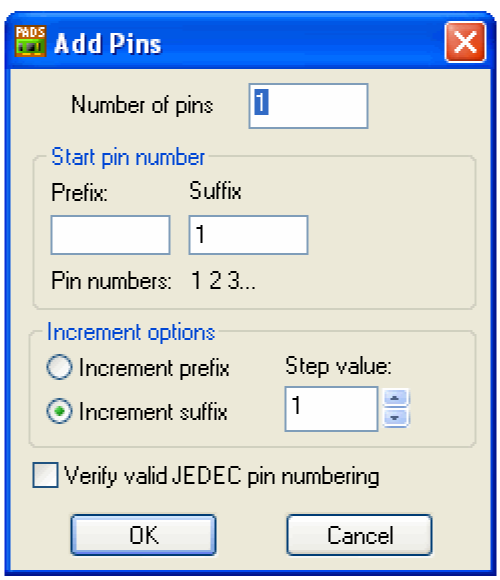


Table 45-16. Add Pins Dialog Box Contents

Name	Description
Number of pins	Specifies the number of pins to add using the Add Pins dialog box.
Prefix	The prefix you want for your pins. **Tips**: • Alphabetic and numeric values can be used. For example, A1 or 1A. • For a single numeric, use either Prefix or Suffix box, and void the other box.
Suffix	The suffix you want for your pins. **Tips**: • Alphabetic and numeric values can be used. For example, A1 or 1A. • For a single numeric, use either Prefix or Suffix box, and void the other box.
Pin numbers	A preview of pin numbers based on your input in the Prefix and Suffix boxes.
Increment prefix/Increment suffix	Indicates whether you want the prefix or the suffix to increment.
Step value	A positive or negative number by which to increase or decrease the pin numbers with consecutive or stepped values.
Verify valid JEDEC pin numbering	Ensures that legal alphanumeric values are used.

Related Topics

Adding a Series of Pins to the Pins Table

Add Substrate Bond Pad Dialog Box

Use the Add Substrate Bond Pad dialog box to create new bond pads in your design.

Restriction: This information applies only to the BGA toolkit.

Accessing

- **BGA Toolbar** button > **Wire Bond Editor button** > **click the BGA** > **Right-click** > **Add SBP**

Figure 45-17. Add Substrate Bond Pad Dialog Box

Table 45-17. Add Substrate Bond Pad Dialog Box contents

Name	Description
Pin Name	Assigns a pin name to the currently selected substrate bond pad.
Function	Defines the function of the currently selected bond pad.
Layer list	Lists all the electrical layers for creating SBPs, so you can create an SBP on a specific layer.
Length	Assigns a physical length, in current design units, for the currently selected bond pad.
Width	Assigns a physical width, in current design units, for the currently selected bond pad.
Add button	Dynamically attaches the substrate bond pad and wire bond to the cursor for placement.

Related Topics

To Add Substrate Bond Pads

Add/Rename SBP Ring Dialog Box

Use the Add SBP Ring dialog box to add a ring for the substrate bond pads.

Figure 45-18. Add SBP Ring Dialog Box

Table 45-18. Add SBP Ring Dialog Box Contents

Name	Description
Text box	Enter the name of the ring as you want it to appear in the Wire Bond Wizard dialog box.

Add Terminals Dialog Box

Use the Add Terminals dialog box to associate pads or pins with a decal.

Accessing

- **Tools** menu > **PCB Decal Editor** > **Drafting Toolbar** button > **Terminals** button

Figure 45-19. Add Terminals Dialog Box

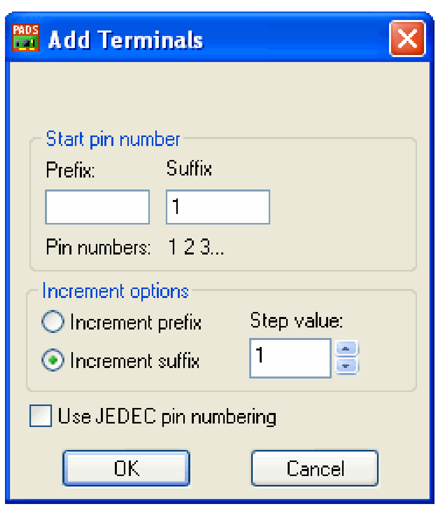


Table 45-19. Add Terminals Dialog Box Contents

Name	Description
Prefix	The prefix you want for your pins. **Tips**: • Alphabetic and numeric values can be used. For example, A1 or 1A. • For a single numeric, use either Prefix or Suffix box, and void the other box.
Suffix	The suffix you want for your pins. **Tips**: • Alphabetic and numeric values can be used. For example, A1 or 1A. • For a single numeric, use either Prefix or Suffix box, and void the other box.
Pin numbers	A preview of pin numbers based on your input in the Prefix and Suffix boxes.
Increment prefix/Increment suffix	Indicates whether you want the prefix or the suffix to increment.
Step value	A positive or negative number by which to increase or decrease the pin numbers with consecutive or stepped values.
Use JEDEC pin numbering	Ensures that legal alphanumeric values are used.

Related Topics

To Add a Terminal

Align Parts Dialog Box

Use the Align Parts dialog box to align selected objects along the left side, right side, vertical center, top, bottom, or horizontal center.

Accessing

- **Select the parts you want to align** > right-click > **Align**

Figure 45-20. Align Parts Dialog Box

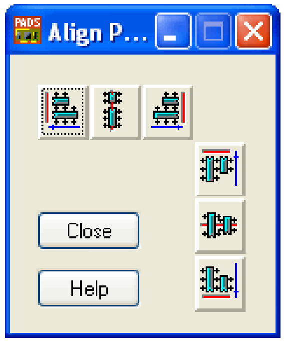
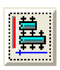
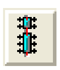
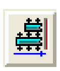
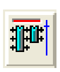
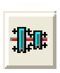
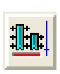


Table 45-20. Align Parts Dialog Box

Name	Description
	Aligns selected items vertically along the left edge.
	Aligns selected items vertically through the middle.
	Aligns selected items vertically along the right edge.
	Aligns selected items horizontally across the top.
	Aligns selected items horizontally through the middle.
	Aligns selected items horizontally across the bottom.

Related Topics

To Align Objects

Archive Navigator Options Dialog Box

Use the Options dialog box to set the display format and display order of the archives in the Vault view.

Accessing

- **Vault view** > **Options** button

Figure 45-21. Archive Navigator Options Dialog Box

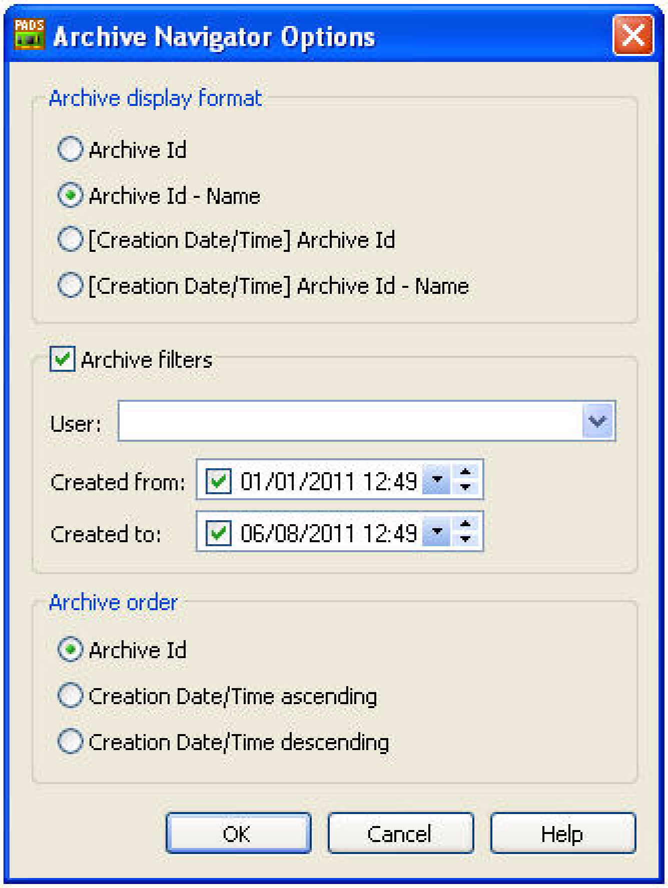


Table 45-21. Archive Navigator Options Dialog Box Contents

Name	Description
Archive display format	Specifies how archives should be listed in the vault view tree, as follows: • Select **Archive Id** to display only the assigned archive ID (Archive 1, Archive 2, ...). • Select **Archive Id - Name** to display the archive ID and the optional archive Name. • Select **[Creation Date/Time] Archive Id** to display the archive creation date & time and the archive ID. • Select **[Creation Date/Time] Archive Id - Name** to display the archive creation date & time, the archive ID, and the optional archive Name.

Table 45-21. Archive Navigator Options Dialog Box Contents

Name	Description
Archive filters	Specifies which archives should be displayed in the Vault view, as follows: • **User drop-down list**—Select a user to display all archives whose User attribute contains that user name. If no user is selected, archives with any user name are returned. • **Created from—**Select this checkbox and specify a date/time to display all archives created after the specified date/time. • **Created to** —Select this checkbox and specify a date/time to display all archives created up to the specified date/time.
Archive order	Specifies the order in which archives are displayed in the Vault view, as follows: • **Archive Id**—Orders by assigned archive ID (Archive 1, Archive 2, ...). • **Creation Date/Time ascending**—Orders by creation date/time, most recent at the bottom. • **Creation Date/Time descending**—Orders by creation date/time, most recent at the top.

Related Topics

Setting Display Options in the Vault View

Archive Navigator Window

Use the Archive Navigator window to:

- Create, organize and manage your PCB project archives.
- Work with the files in an archive.

The Archive Navigator window has two *views*:

- The Vault view, where you view and manage vaults, folders, projects and archives.
- The Working Folder view, where you view and work with the files in the current working folder.

Accessing

- **Tools** menu > **Archive Navigator**

Related Topics

Adding a Project Container to the Vault

Archive Navigator Vault View

Use the Archive Navigator Vault view to view and manage vaults, folders, projects and archives.

Figure 45-22. The Archive Navigator Vault View

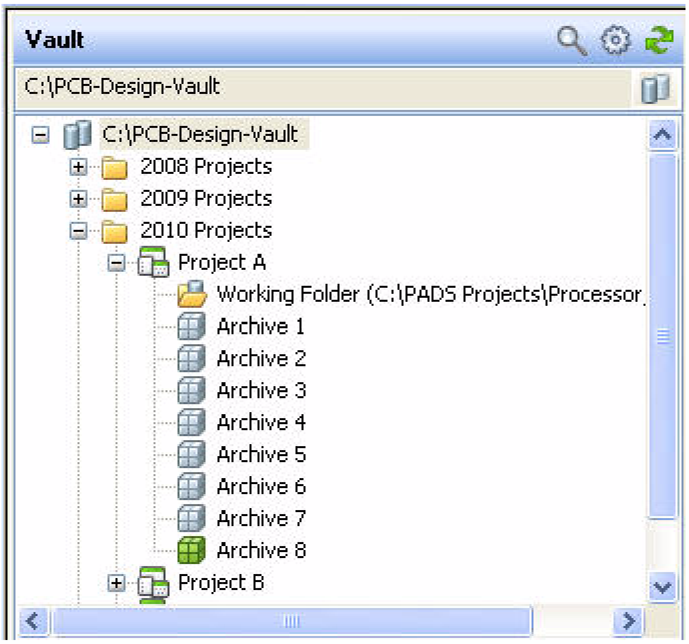


Table 45-22. Archive Navigator Vault View Contents

Name	Description
Find in Vault	Opens the Find in Vault dialog box, where you can search the vault for projects and archives.
Options	Opens the Options dialog box, where you can specify how archives are displayed in the vault view tree.
Refresh Vault	Redisplays the current contents of the vault.
Select Vault	Opens the Select Vault dialog box, where you can select a different vault, or create a new one.
Name Created User Description	Displays the attributes of the item selected in the vault view tree: **Name**—The system generated archive ID (and optionally the user-supplied archive name) **Created**—The date and time when the archive was created **User**—The user who created the archive **Description**—The user-supplied description of the archive
Vault View tree area	Displays the vault's hierarchy of projects, folders, and archives. **Tip:** The archive currently restored to the working folder is highlighted in green.

Vault View Operations

You can perform the following operations on items in the vault view tree:

Vaults	Create a new vault.	See Creating a Vault.
	Create a new empty project in the vault.	Select vault, right-click > Create Empty Project
	Create a new folder in the vault.	Select vault, right-click > Create folder
	Filter the display of archives in the vault.	See Setting Display Options in the Vault View.
	Set the display order and format of archives in the vault.	See Setting Display Options in the Vault View.
Folders	Create a new empty project in the folder.	Select folder, right-click > Create Empty Project
	Create a new folder in the folder.	Select folder, right-click > Create Folder
	Move or copy a folder to a new location in the vault.	Select folder, right-click > Move to or Copy to

	Delete the folder from the vault.	Select folder, right-click > Delete
Projects	Set or clear the working folder for the project.	Right-click the working folder > Set Working Folder or Clear Working Folder
	Delete the project from the vault.	Select project, right-click > Delete
	Move or copy a project to a new location in the vault.	Select project, right-click > Move to or Copy to
	Edit a project's templates.	Select project, right-click > Edit Templates
	View the properties of the project.	Right-click the project > Properties
Archives	View the contents of an archive.	Right-click the archive > View Content
	Restore an archive to the working folder.	Right-click the archive > Restore
	Delete an archive from the vault.	Right-click the archive > Delete
	View the properties of an archive.	Right-click the archive > Properties

Related Topics

Creating a Vault

Adding a Project Container to the Vault

Creating a Vault Folder

Changing Vaults

Deleting Items from the Vault

Viewing and Editing Properties of a Vault Item

Setting Display Options in the Vault View

Viewing the Contents of an Archive

Finding Projects or Archives in the Vault

Archive Navigator Working Folder View

Use the Working Folder window to view and work with the files and folders in the current working folder.

Figure 45-23. Archive Navigator Working Folder View

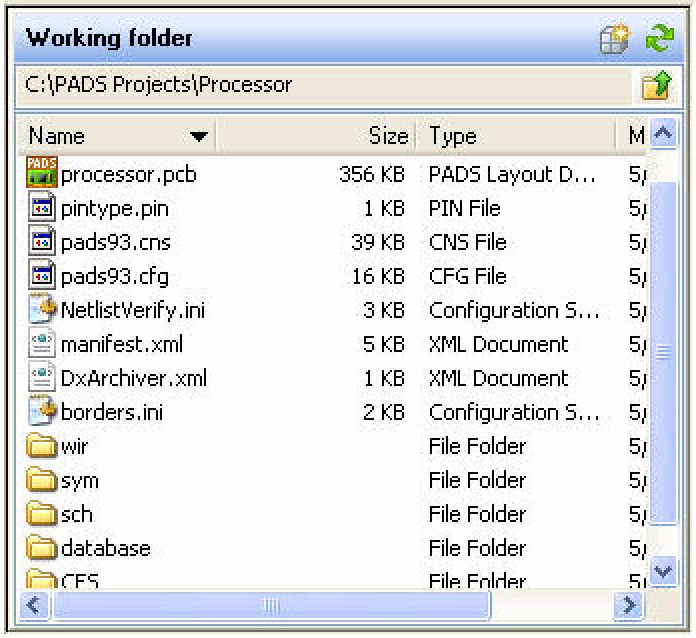


Table 45-23. Archive Navigator Working Folder View Contents

Name	Description
Add archive to Vault	Opens the Add Archive to Vault dialog box, where you define and save an archive of the current state of the working folder.
Refresh working Folder	Redisplays the working folder list, showing the current contents.
Folder up	Opens the parent folder of the folder currently open. **Restriction:** You cannot go above the working folder.
List area	Displays the contents of the working folder. Double-click on any item to open it, as follows: • Folders are opened in the Working Folder view. • .pcb files are opened in the current PADS Layout instance. • Other files are opened in the application associated with the file type.

Related Topics

Adding a Project Container to the Vault

Adding an Archive to the Vault

Creating an Archive Template

Restoring an Archive to the Working Folder

Archive Navigator Status Bar

The Status Bar displays the state of the Archive filters and Archive order controls in the Options Dialog box.

Figure 45-24. Archive Navigator Status Bar

See the Archive Navigator Options Dialog Box topic for a description of these settings.

Archive Properties Dialog Box

Use the Archive Properties dialog box to view and edit an Archive Navigator archive's properties.

Accessing

- In the Archive Navigator Vault view, right-click an archive, and click **Properties.**

Figure 45-25. Archive Properties Dialog Box

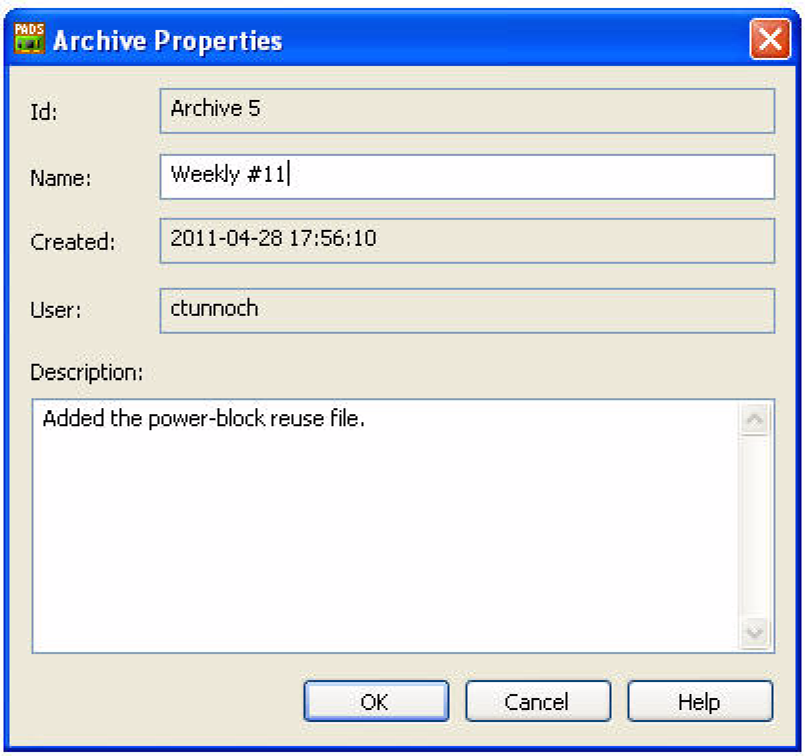


Table 45-24. Archive Properties Dialog Box Contents

Name	Description
Id	The system-generated archive ID
Name	Specifies a name attribute for the archive. **Tip:** Create a name you can search for with the Find in Vault tool.
Created	The date/time when the archive was created
User	The user who created the archive
Description	Specifies a description attribute for the archive. **Tip:** Create a description you can search for with the Find in Vault tool.

Archiver Dialog Box

Use the Archiver dialog box to create archives of your designs, projects, files and folders, and libraries.

Accessing

- **File** menu > **Archive**

Figure 45-26. Archiver Dialog Box

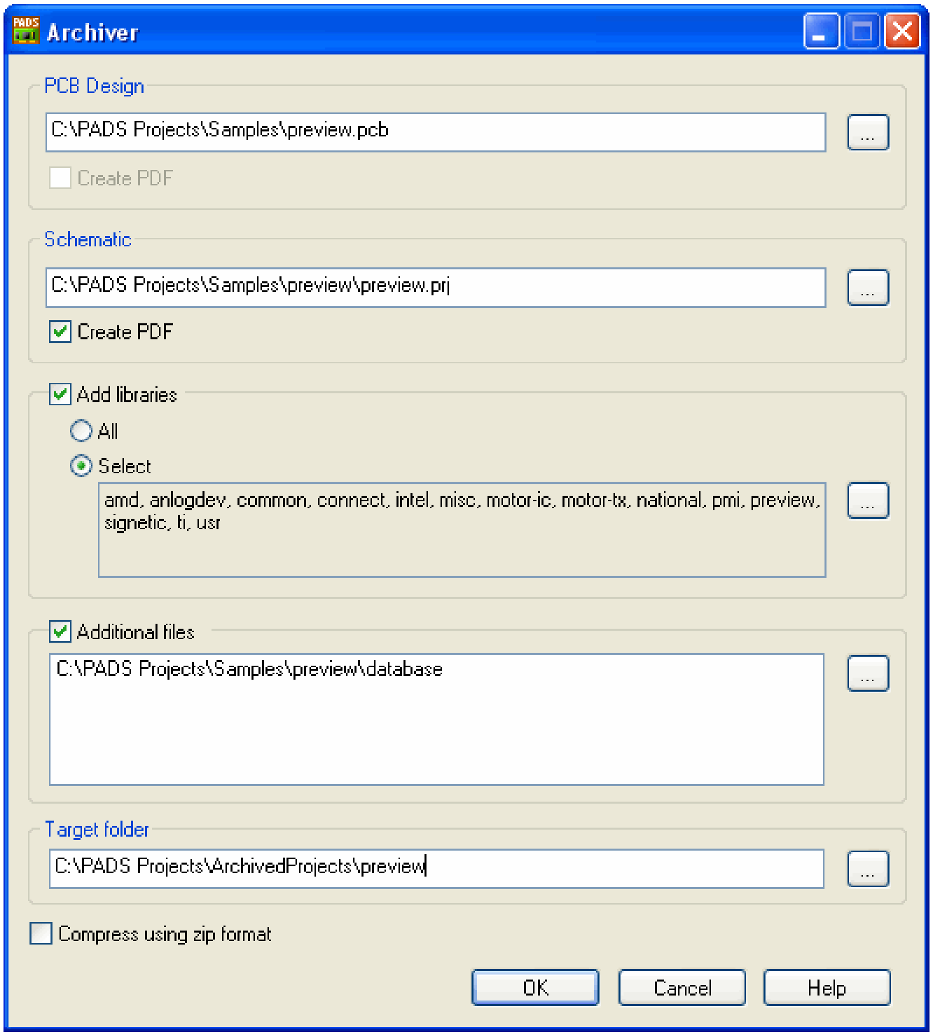


Table 45-25. Archiver Dialog Box

Name	Description
PCB Design	Specifies the location and name of the PCB design you want to archive. This is automatically populated with the information from the current design. To change the design, or if no design was opened, type the location or click the **Browse** button. Select **Create PDF** to create a PDF file of the PCB design. **Restriction:** This is unavailable if the file you chose is different from the current design.

Table 45-25. Archiver Dialog Box (cont.)

Name	Description
Schematic	Specifies the location and name of the schematic file you want to archive. To choose the file you want, type the location or click the **Browse** button. Select **Create PDF** to create a PDF file of the schematic file. **Restriction:** This is only available if you chose a DxDesigner file; it is unavailable if you chose a PADS Logic file.
Add libraries	Specifies that you want to include libraries in the archive. • **All**—Add all of your libraries to the archive. • **Select**—Add only the libraries you specify. Click the **Browse** button to open the Archiver: Libraries dialog box.
Additional files	Specifies that you want to include other files and folders in your archive. Click the **Browse** button to open the Archiver: Additional Files dialog box.
Target folder	Specifies where you want the archive to be located. Type the path or click the **Browse** button. **Restriction:** If the **Compress using zip format** checkbox is unchecked, the target folder must be empty.
Compress using zip format	Specifies to create a zip file. The filename will be in the following format: `<project_name>YYYYMMDDHHMMSS.zip` Where YYYY is the year, MM is the month, DD is the day, HH is the hour - in military time, MM is the minute, and SS is the second of the exact time you created the file.

Related Topics

Archiving Your Design

Archiver: Additional Files Dialog Box

Use the Archiver: Additional dialog box to add files and folders to the design you want to archive.

Accessing

- **File** menu > **Archive** > **Additional Files check box** > **Browse button**

Figure 45-27. Archiver: Additional Files Dialog Box

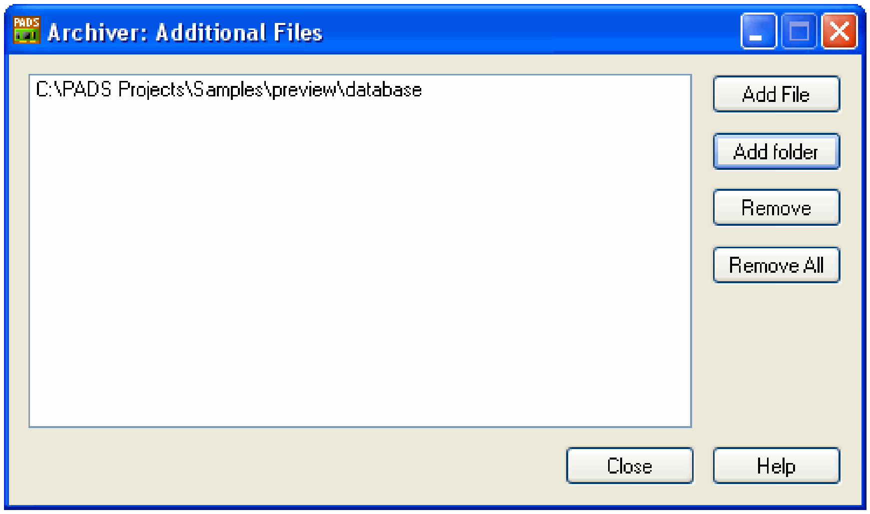


Table 45-26. Archiver: Additional Files Dialog Box

Name	Description
Additional files list	Lists the files and folders you want to include in your archive.
Add File button	Opens the Additional File dialog box where you can select individual files you want to add to the Additional files list.
Add folder button	Opens the Browse for Folder dialog box where you can select an entire folder to add to the Additional files list.
Remove button	Removes the selected file or folder from the Additional files list.
Remove All button	Removes all of the files and folders from the Additional files list.

Related Topics

Archiving Your Design

Archiver: Libraries Dialog Box

Use the Archiver: Libraries dialog box to add libraries to the design you want to archive.

Accessing

- **File** menu > **Archive** > **Add libraries check box** > **Select** > **Browse button**

Figure 45-28. Archiver: Libraries Dialog Box

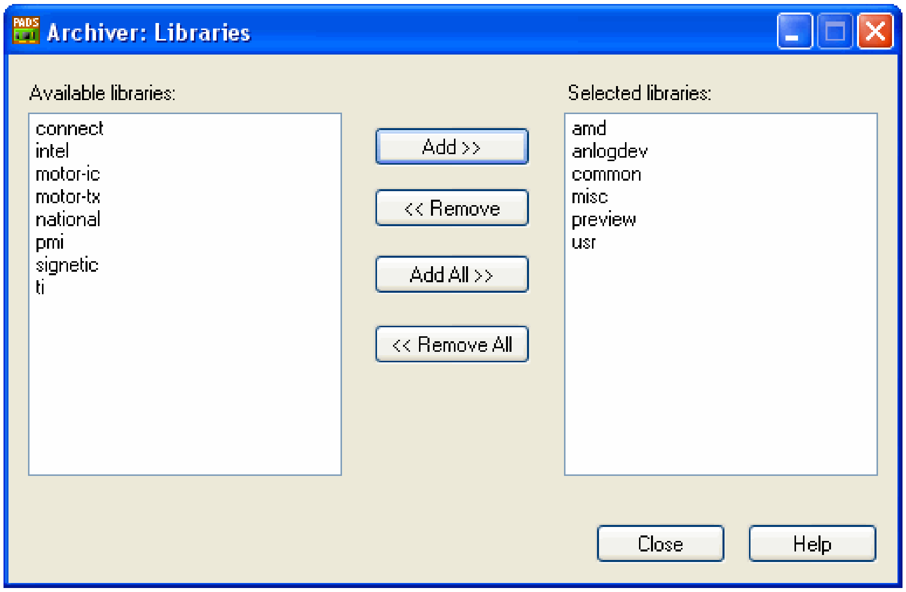


Table 45-27. Archiver: Libraries Dialog Box

Name	Description
Available libraries	Lists all of the libraries available for you to add to the archive. **Restriction:** If your library is not listed in the Library Manager, it will not appear in this list.
Add >> button	Moves the selected library from the Available libraries list to the Selected libraries list.
<< Remove button	Moves the selected library from the Selected libraries list to the Available libraries list.
Add all >> button	Moves all of the libraries from the Available libraries list to the Selected libraries list.
<< Remove all button	Moves all of the libraries from the Selected libraries list to the Available libraries list.

Related Topics

Archiving Your Design

Arrow Properties Dialog Box

The Arrow Properties dialog box displays coordinate information for the selected arrow and provides several areas for modifications.

The Arrow Properties dialog box remains open until you click OK or Cancel. Selecting another arrow while the dialog box is open updates the information for the selected object.

Accessing

- Select an arrow > right-click > **Properties**.

Figure 45-29. Arrow Properties Dialog Box

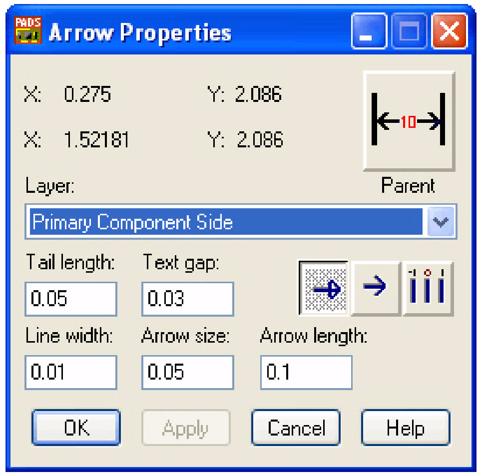


Table 45-28. Arrow Properties Dialog Box contents

Name	Description
X and Y values	Displays the X and Y coordinate locations of the selected object(s).
Parent button	Opens the Dimension Properties Dialog Box for the dimension object with which the selected object is associated.
Layer list	Lists the current working layer. Select a new layer from the list to change it.
Tail Length	Specifies the current minimum length of the arrow tail, which is the line extending beyond the arrow. Type a new value to change the tail length.
	Indicates the current arrow type. Click an alternate button to modify.
Text Gap	Specifies the current spacing between the tail and the measurement text. Type a new value to change the spacing.
Line Width	Lists the current line width of the tail and arrow lines. Type a new value to change the line width.

Table 45-28. Arrow Properties Dialog Box contents (cont.)

Name	Description
Arrow Size	Lists the current arrow width, which is the height of the arrow. Type a new value to change the width.
Arrow Length	Lists the current arrow length, which is the measurement between the arrow tip and the end of the arrow. Type a new value to change the arrow length.

Related Topics

Moving Dimensions and Dimension Objects

ASCII Output Dialog Box

You can use ASCII files to exchange design data between PADS Layout and external translators or previous versions of PADS Layout.

Accessing

- **File** menu > **Export** > **Select ASCII Files type** > **Save**

Figure 45-30. ASCII Output Dialog Box

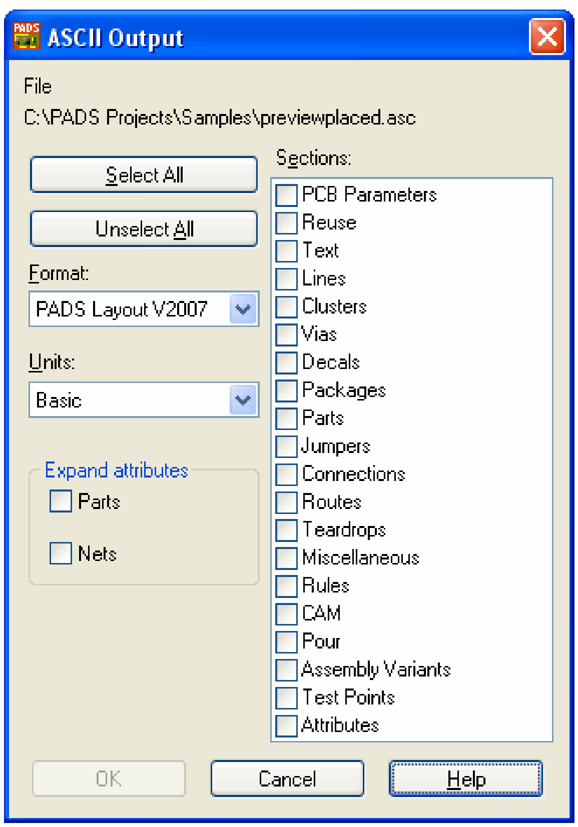


Table 45-29. ASCII Output Dialog Box Contents

Name	Description
File	The name of the file you are exporting.
Sections	Specifies the sections of the ASCII file you want to export. See also: Exported Item Descriptions
Select All	Selects all items in the Sections list.
Unselect All	Unselects all items in the Sections list.
Format	Specifies the format of the ASCII file. **Tips:** • Use PowerPCB V3.0 to export to both PowerPCB and PowerBGA V3.0 ASCII formats. • PADS Layout does not export to PADS-Perform 6 ASCII format.
Units	Specifies the units to export. **Tip:** If you plan to use the ASCII file for an external translator or another ASCII-reading program, select Current. If you plan to re-import the ASCII file and save it as a .pcb database, select Basic. Basic units represent how values are stored in the software. They do not use standard units of measure, but they record precise positioning values for database items.
Parts	Specifies that you want to export attributes assumed from higher levels in the attribute hierarchy.
Nets	Specifies that you want to export attributes assumed from higher levels in the attribute hierarchy.

Exported Item Descriptions

The following table provides descriptions for each item in the Sections area of the ASCII Output dialog box:

Table 45-30. ASCII Output Item Descriptions

Item	Exported information
PCB Parameters	Global design information, such as units and colors
Reuse	Elements in, and the definition of, a physical design reuse
Text	Text

Table 45-30. ASCII Output Item Descriptions (cont.)

Item	**Exported information**
Lines	Two-dimensional (2D) lines
Clusters	Clusters and unions
Vias	Vias (including dangling vias), jumpers, and padstacks
Decals	Footprints
Packages	Electrical information
Parts	Component instances
Jumpers	Jumpers **Tip:** If you plan to export to the PowerPCB V1.1 ASCII format, you cannot output complete jumper information. This is because PADS Layout considers jumper pins as vias and jumpers are exported as vias when you select the Vias check box.
Connections	Unrouted pin pairs
Routes	Traces, including route loops
Teardrops	Teardrops **Tip:** If you plan to export to the PowerPCB V1.1 ASCII format, you cannot output teardrops. **Requirement:** You must export routes to export teardrops.
Miscellaneous	Information not included in other items
Rules	Clearance, routing, and high-speed rules
CAM	Information related to plot file configurations generated using CAM
Pour	Copper pours
Assembly Options	Assembly variants
Test Points	Test points and the test side (top, bottom, or both)
Attributes	Attribute Dictionary and all individual attributes and value assignments in the design. Status of attributes (read-only, system, ECO-registered, or hidden). Attributes are exported to the extent possible for formats previous to V3.0. Previous versions do not support all of the default attributes. **Tip:** Values through the attribute hierarchy are not exported.

Related Topics

Exporting ASCII Files

Importing and Exporting Files

Assembly Variants Dialog Box

Use the Assembly Variants dialog box to create a new variant, review or edit a variant, preview variants, delete variants, and create reports for variants.

Accessing

- **Tools** menu > **Assembly Variants**

Figure 45-31. Assembly Variants Dialog Box

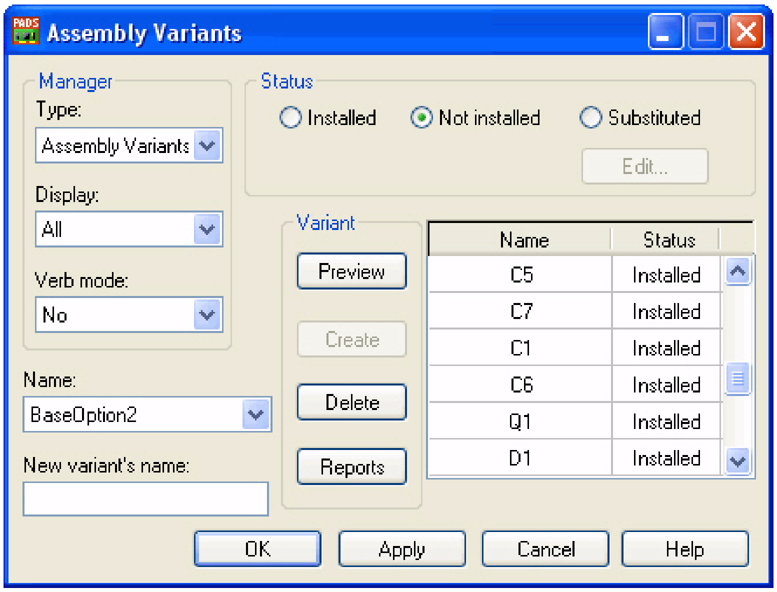


Table 45-31. Assembly Variants Dialog Box contents

Name	Description
Type list	Specifies what displays in the table: Assembly Variants or Components.
Display list	Filters the Variant table based on your selection: All, Installed, No Installed, Substituted.
Verb mode	Specifies what action to take and perform on components in the Layout Editor (outside the dialog box): Install, Uninstall, Substitute, No.
Name	Specifies the name of the Assembly Variant or the Component you want listed in the table. **Tip:** Changes depending on the selection made in the Type list.
New Variant’s name	Specifies the name of the new variant, up to 26 characters.

Table 45-31. Assembly Variants Dialog Box contents (cont.)

Name	Description
Status area	Specifies the status for the selected variant: Installed, Not installed, Substituted. **Tip:** If you click Substituted, the Variant/Substitute Dialog Box opens.
Edit button	Opens the Variant/Substitute Dialog Box. **Restriction:** Unavailable unless the component has already been substituted.
Preview button	Opens the Preview for dialog box.
Create button	Creates a new variant and adds it to the Name list.
Delete button	Removes the selected variant from the Name list.
Reports button	Opens the Reports Dialog Box.
Variant/Components table	**Lists the name and status of the variants or components selected in the Name list.** **Tip:** Changes depending on the selection made in the Type list.

Related Topics

Using the Assembly Variants Dialog Box

Assign CBPs to Rings Dialog Box

Restriction: This information applies only to the BGA toolkit.

Use this dialog box to specify which component bond pads to wire bond to which ring. You can view this placement in preview mode.

Accessing

- **BGA Toolbar** button > **Wire Bond Wizard** button > **Assign CBPs** button

Figure 45-32. Assign CBPs to Rings Dialog Box

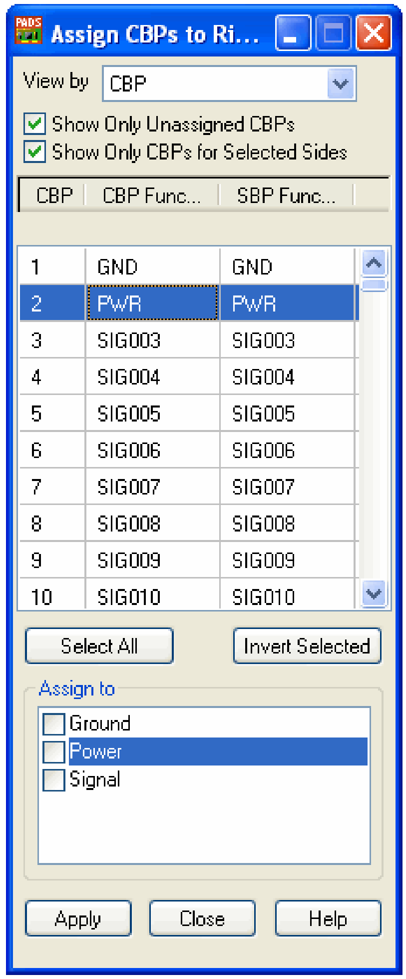


Table 45-32. Assign CBPs to Rings Dialog Box contents

Name	Description
View by	Select an option for viewing the component bond pad data in the CBP list: CBP, CBP Function, SBP Function.
Show Only Unassigned CBPs	Displays only the unassigned component bond pads in the CBP list.
Show Only CBPs for Selected Sides	Displays only component bond pads specified in the Generate Fanout for area of the Wire Bond Wizard Dialog Box. If you do not select this option, the list displays all component bond pads, no matter what the current settings are in the Generate Fanout for area.

Table 45-32. Assign CBPs to Rings Dialog Box contents (cont.)

Name	Description
CBP column	Displays the CBP number. **Restriction:** Displayed only when CBP is selected from the View by list.
CBP Function column	Component bond pad function. **Restriction:** Displayed only when CBP or CBP Function is selected from the View by list.
SBP Function column	Substrate bond pad function. **Restriction:** Displayed only when CBP or SBP Function is selected from the View by list.
CBP Count	The component bond pad count for each bond pad function. **Restriction:** Displayed only when CBP Function or SBP Function is selected from the View by list.
Select All button	Selects all component bond pads in the CBP list to assign to the currently selected rings.
Invert Selected button	Inverts all selections currently in the list. If one item is currently selected, this button will deselect that one item and select all other items instead.
Assign to area	Use to select the ring or rings to which you want the selected component bond pad or pads assigned. You can select multiple rings at a time for component bond pad assignment.
Apply button	Applies your settings for assigning component bond pads to rings. Also clears the selections in the Assign to list. It does not close the Assign CBPs to Rings Dialog Box, so you can continue to make assignments.

Related Topics

To Assign CBPs to Rings

Assign Color to All Layers Dialog Box

Use the Assign Color to All Layers dialog box to make objects visible, or make objects invisible, in a design.

Accessing

- **Setup** menu > **Display Colors** > **Assign All** button

Figure 45-33. Assign Color to All Layers Dialog Box

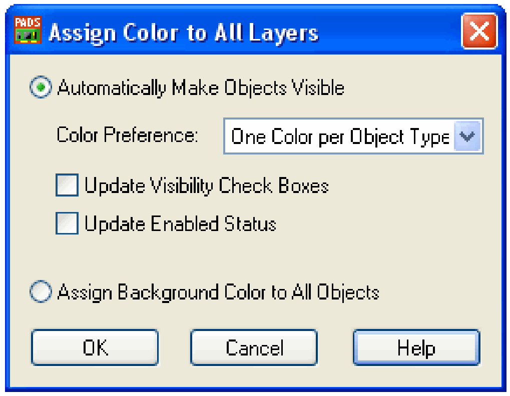


Table 45-33. Assign Color to All Layers Dialog Box Contents

Name	Description
Automatically Make Objects Visible	Specifies to make objects visible.
Color Preference	Specifies the way you want to make objects visible in a design.: • **One Color per Object Type**—Assigns color for a certain object type, when it is currently set to the background color, on all layers. The program assigns color according to the color set in the immediately adjacent tile, or, if no adjacent tile exists, according to color palette order. • **One Color for a Layer**—Assigns color for all objects, when they are currently set to the background color, on the same layer. The program assigns color according to the color set in the immediately adjacent tile, or, if no adjacent tile exists, according to color palette order. • **Selected Color**—Assigns the color you select from the Display Colors Setup dialog box color palette to all objects, which are currently set to the background color, on all layers.
Update Visibility Check Boxes	Specifies that the visibility check boxes surrounding the color matrix are selected if data exists on a layer, and clear if no data exists on a layer.
Update Enabled Status	Specifies color assignment based on layer settings in the Enable/Disable Layer dialog box. When you select this option, nonelectrical layers that do not contain data are disabled.

Table 45-33. Assign Color to All Layers Dialog Box Contents (cont.)

Name	Description
Assign Background Color to All Objects	Specifies to make all objects on all layers invisible by changing the color of all objects, on all layers, to the current background color.

Related Topics

Setting Colors of Objects in the Display

Assign Decal to Gate Dialog Box

Use the Assign Decal to Gate dialog box to assign default and alternative CAE decals to gates.

Tip: This dialog box works similarly to the PCB Decals tab on the Part Information dialog box, except that it deals with logic decals instead of PCB decals.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **Gates** tab > double-click CAE Decal cell > **Browse** button

Figure 45-34. Assign Decal to Gate Dialog Box

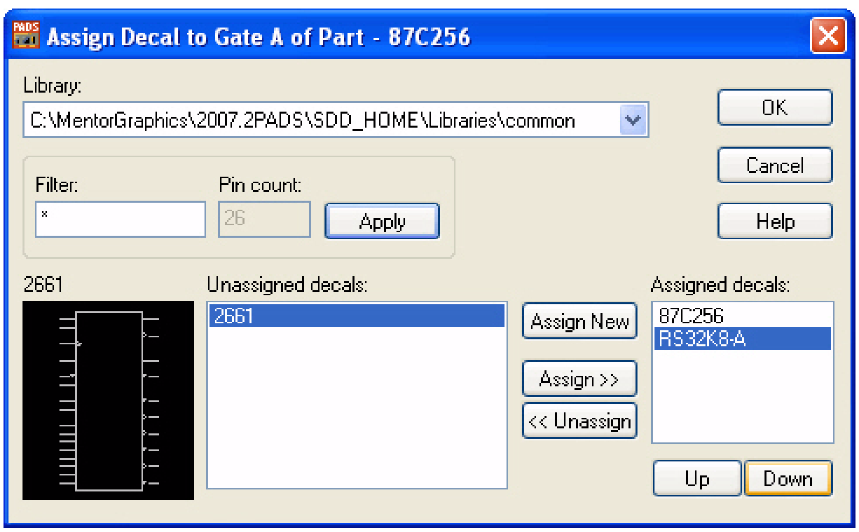


Table 45-34. Assign Decal to Gate Dialog Box Contents

Name	Description
Library list	Lists all libraries available to you.

Table 45-34. Assign Decal to Gate Dialog Box Contents (cont.)

Name	Description
Filter	Narrows down your Unassigned decals list. **Tip:** You can use wildcards in this box.
Pin Count	Lists the pin count for the selected gate.
Apply	Executes the filter arguments.
Preview area	Shows the selected Decal.
Unassigned Decals list	Lists all unassigned decals available to assign to the selected gate in the selected library.
Assign New	Opens the Assign New Gate Decal Dialog Box, where you can enter the name of the new decal for the gate.
Assign >>	Moves the selected decal from the Unassigned Decals list to the Assigned Decals list.
<< Unassign	Moves the selected decal from the Assigned Decals list to the Unassigned Decals list.
Assigned Decals list	Lists all assigned decals to the selected gate in the selected library.
Up/Down	Moves the selected Decal up or down.

Related Topics

Assigning CAE Decals to Gates

Assign New Gate Decal Dialog Box

Use the Assign New Gate dialog box to assign a new gate decal when it doesn't yet exist in the Library.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **Gates** tab > double-click CAE Decal cell > **Browse** button > **Assign New**

Figure 45-35. Assign New Gate Decal Dialog Box

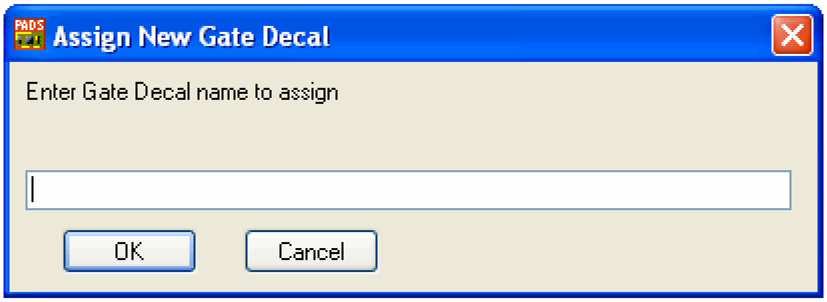


Table 45-35. Assign New Gate Decal Dialog Box Contents

Name	Description
Text box	Enter the name of the new gate decal you intend to add to the library.

Assign New PCB Decal Dialog Box

Use the Assign New PCB Decal dialog box to assign a new PCB Decal when it doesn't yet exist in the Library.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **PCB Decals** tab > **Assign New**

Figure 45-36. Assign New PCB Decal Dialog Box

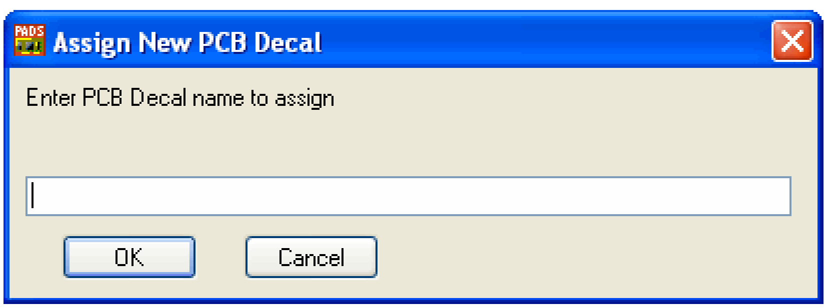


Table 45-36. Assign New PCB Decal Dialog Box Contents

Name	Description
Text box	Enter the name of the new PCB decal you intend to add to the library.

Assign Pin Numbers Dialog Box

If you update a part type from the library and there is a change in pin numbers, you must reassign connections to the new decal pin numbers. Use the Assign Pin Numbers dialog box to correctly map old decal connections to new decal connections.

Accessing

- This dialog box opens when you perform a task that can change the number of pins, such as changing a component **or** updating a part type from the library**.**

Figure 45-37. Assign Pin Numbers Dialog Box

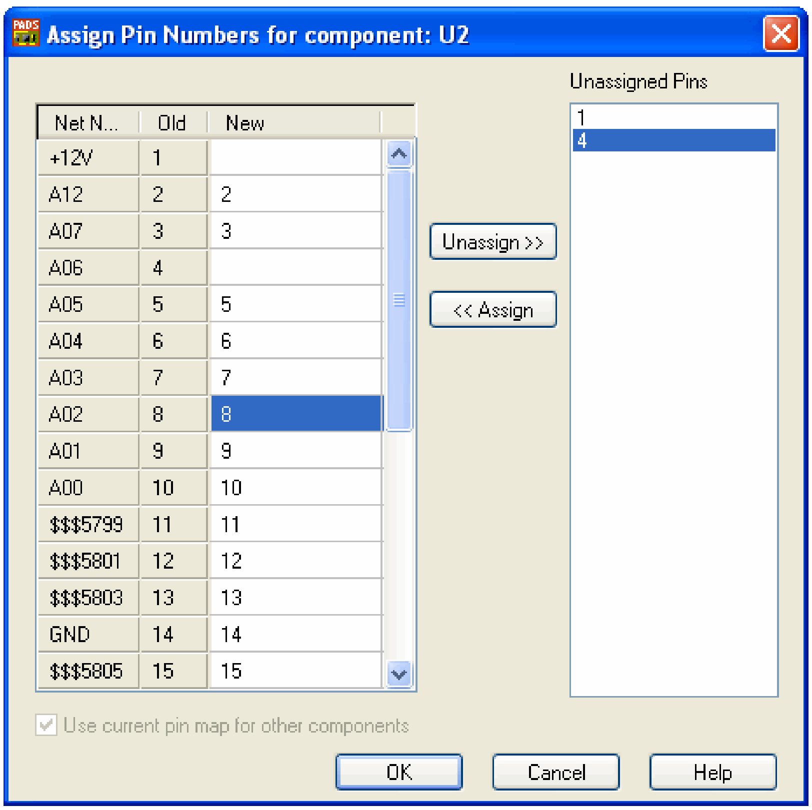


Table 45-37. Assign Pin Numbers Dialog Box contents

Name	Description
Net name column	Displays the net connected to the existing decal pin numbers - listed in the Old column.
Old	Displays the existing pin number connected to the net.

Table 45-37. Assign Pin Numbers Dialog Box contents (cont.)

Name	Description
New	Specifies the new pin number connected to the net. **Tip:** If a pin has a signal assigned to it in the part type, the signal will be shown after the pin name, in brackets. New decal pin numbers are matched where possible. If no match exists, the cell in the New column will be empty and/or a pin number will be listed in the Unassigned Pins list.
Unassign	Moves the selected new pin(s) to the Unassigned Pins list
Assign	Moves the selected unassigned pin(s) to the selected new pin cell(s). Assignment of pin numbers begins with the first selected cell in the New column.
Unassigned Pins list	Lists all unassigned pins associated with this component. **Tip:** If a pin has a signal assigned to it in the part type, the signal will be shown after the pin name, in brackets.
Use current pin map for other components	Specifies to use this pin mapping for all other components. If you clear this check box, you are prompted to change the pin mapping for other selected components in the design. You may not cancel without canceling the entire operation.

Related Topics

Updating a Part Type from the Library in ECO Mode

Changing a Component in ECO Mode

Assign Shortcut Dialog Box

Create a new shortcut key with using the Assign Shortcut dialog box.

Accessing

- **Tools menu** > **Customize** > **Keyboard and Mouse tab** > **New button**

Figure 45-38. Assign Shortcut Dialog Box

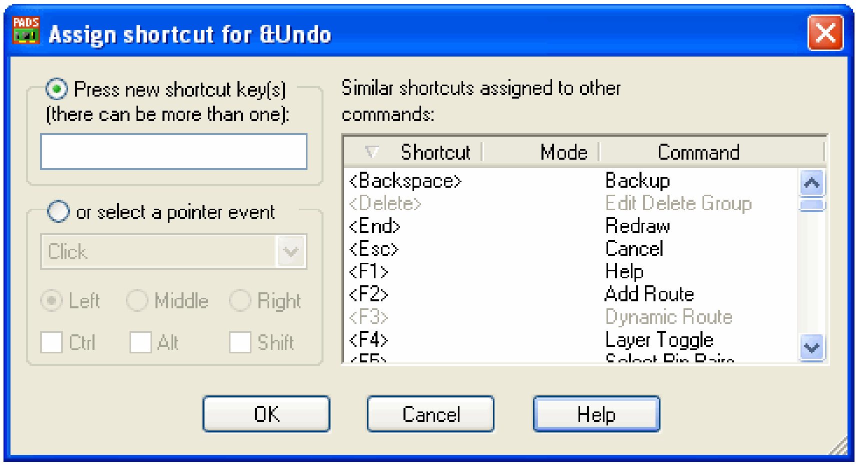


Table 45-38. Assign Shortcut Dialog Box Contents

Name	Description
Press new shortcut key	Type the shortcut you want to use.
Select a pointer event	Set a pointer event shortcut
Similar shortcuts list	Lists the shortcut keys already assigned to other commands.

Associated Net Rules Dialog Box

Use the Associated Net Rules dialog box to select a list of associated nets and:

- Assign HiSpeed rules to them, or
- Clear all rules defined for them.

Accessing

- **Setup** menu > **Design Rules > Associated Nets button**
  or
  Select an associated net > right-click **> Show Rules**

Figure 45-39. Associated Net Rules Dialog Box

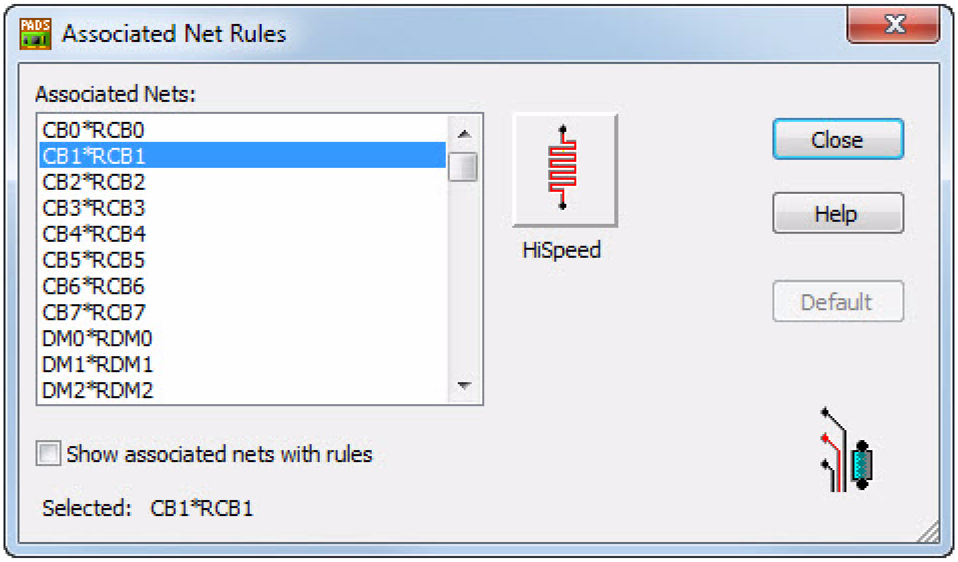


Table 45-39. Associated Net Rules Dialog Box

Name	**Description**
Associated Nets list	Lists the associated nets in the design. **Tip:** (H) identifies associated nets having existing rules.
HiSpeed button	Opens the HiSpeed Rules Dialog Box, where you can define min/max length rules and create matched length groups for the selected associated nets.
Show associated nets with rules	Lists only the associated nets that have rules.
Default button	Removes all rules from the selected associated nets.

Associated Nets Dialog Box

Accessing

- **Setup** menu > **Associated Nets**

Figure 45-40. Associated Nets Dialog Box

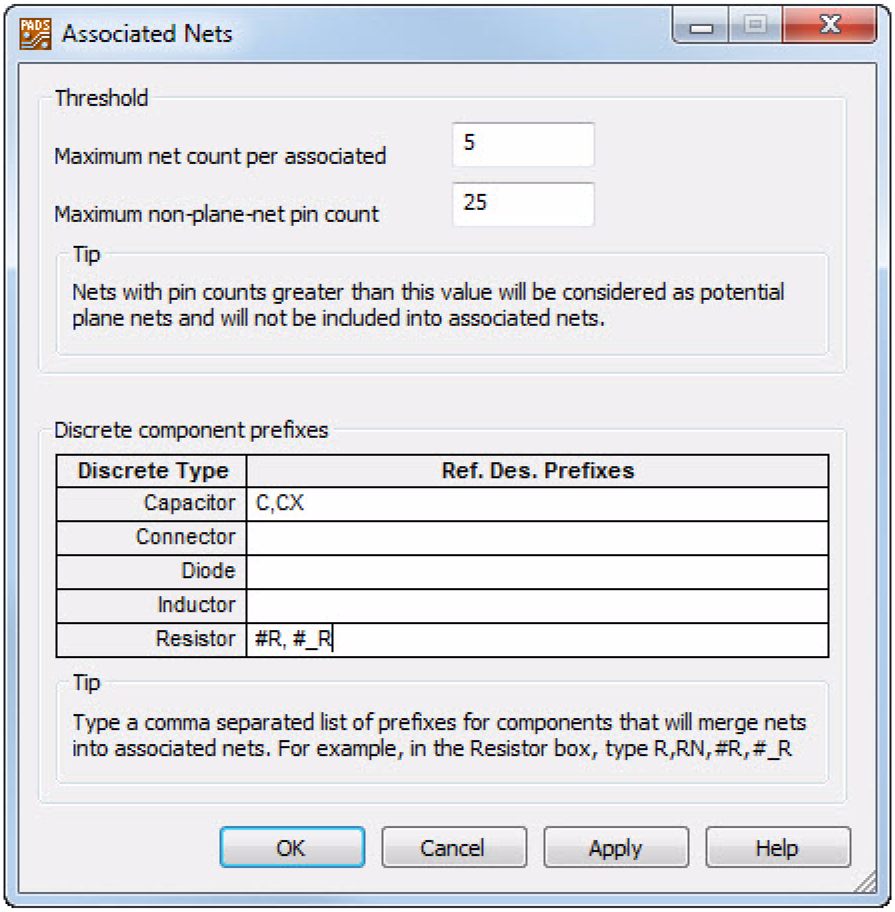


Table 45-40. Associated Nets Dialog Box Contents

Name	Description
Maximum net count per associated net	When creating associated nets, it's possible that associated nets containing an unreasonable number of nets could be created. To prevent this, specify the maximum number of nets to allow in an associated net. Associated nets having more than this limit are not created. **Tip:** This limit also applies to the creation of associated nets manually by component and by net.

Table 45-40. Associated Nets Dialog Box Contents

Name	Description
Maximum non-plane-net pin count	When creating associated nets, it's possible that nets that are potential plane nets could be included in an associated net. (Plane nets are not allowed in associated nets.) To prevent this, specify the maximum pin count for nets included in associated nets. **Tip:** This limit also applies to the creation of associated nets manually by component and by net.

Discrete component prefixes	Specify the refdes prefixes of the associating components to create new associated nets. Examples:
	R specifies all R<*num*> components, where <*num*> is a non-empty number.
	#R specifies all <*num1*>R<*num2*> components; <*num1*> can be empty.
	#_R specifies all <*num1*>_R<*num2*> components, where <*num1*>and <*num2*> are non-empty numbers.
	
	Tips:
	• The discrete type categories are only for convenience; prefixes for any type of component can be entered in any field.
	• If a selected component has more than two pins, the following conditions must apply, or the component cannot be an associating component, that is, the associated net can't go through the component:
	  • All pins must connect to a gate.
	  • Each gate must have exactly two pins.
	
	Delete refdes prefixes of existing associating components to remove them from associated nets.

Attribute Dictionary Dialog Box

Although you can add new attributes to design objects using the Object Attributes dialog box (select object > right-click > Attribute), you must use the Attribute Dictionary to set the properties for attribute values. It is recommended that you use the Attribute Dictionary to create new attributes for, or to edit and delete attributes in, your design. You can also use the Attribute Dictionary to assign attributes for the design, or remove attributes from objects.

Accessing

- **Edit** menu > **Attribute Dictionary**

Figure 45-41. Attribute Dictionary Dialog Box

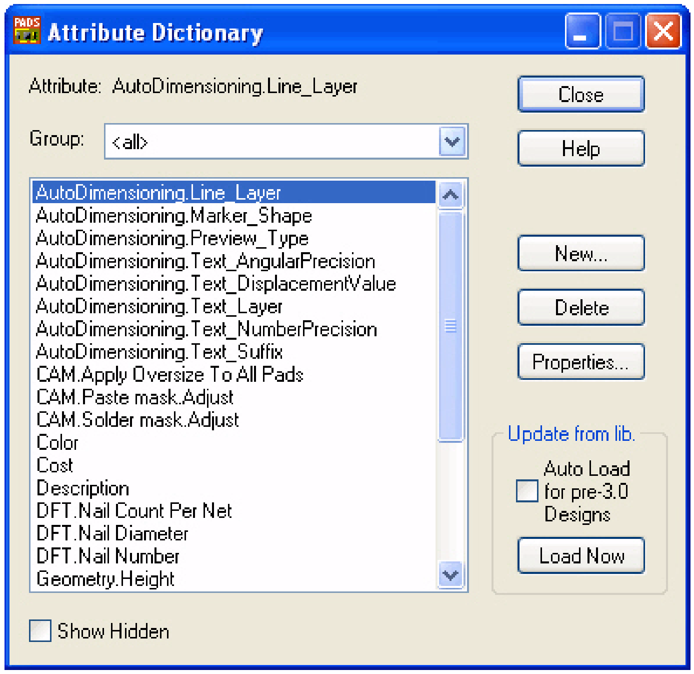


Table 45-41. Attribute Dictionary Dialog Box Contents

Name	Description
Attribute	The name of the attribute selected in the Attribute list.
Group	Filters the Attribute list by showing only the selected group.
Attribute list	Lists all available attributes
Show Hidden	Specifies to show attribute groups that have no visible attributes. **Tip:** You set whether an attribute is hidden on the Objects tab of the Attribute Properties dialog box.
New	Opens the Attribute Properties dialog box.
Delete	Removes the selected attribute. **Tip:** You can delete the default attributes; however, it is not recommended. Because the default attributes are only provided for your design and not assigned to objects, you do not need to delete these attributes.
Properties	**Opens the Attribute Properties dialog box.** **Requirement:** If an attribute is ECO-Registered, you must be in ECO mode to modify the attribute. **Tip:** You can modify the default attributes; however, it is not recommended.

Table 45-41. Attribute Dictionary Dialog Box Contents (cont.)

Name	Description
Auto Load for pre-3.0 Designs	Specifies to automatically update attributes as soon as the design is loaded.
Load Now	Automatically loads attributes for part types and decals from the current libraries to the Attribute Dictionary

Related Topics

Using the Attribute Dictionary

Attribute Manager Dialog Box

Use the Attribute Manager to view all of the attributes on all objects in the design. The Attribute Manager provides a spreadsheet view of all the attributes in the design. You can use the Attribute Manager to add, edit, and delete attribute values on multiple object types. You can also create value summaries of an attribute that is based on every value of the attribute assigned to objects of the same type.

Accessing

- **Edit** menu > **Attribute Manager**

Figure 45-42. Attribute Manager Dialog Box

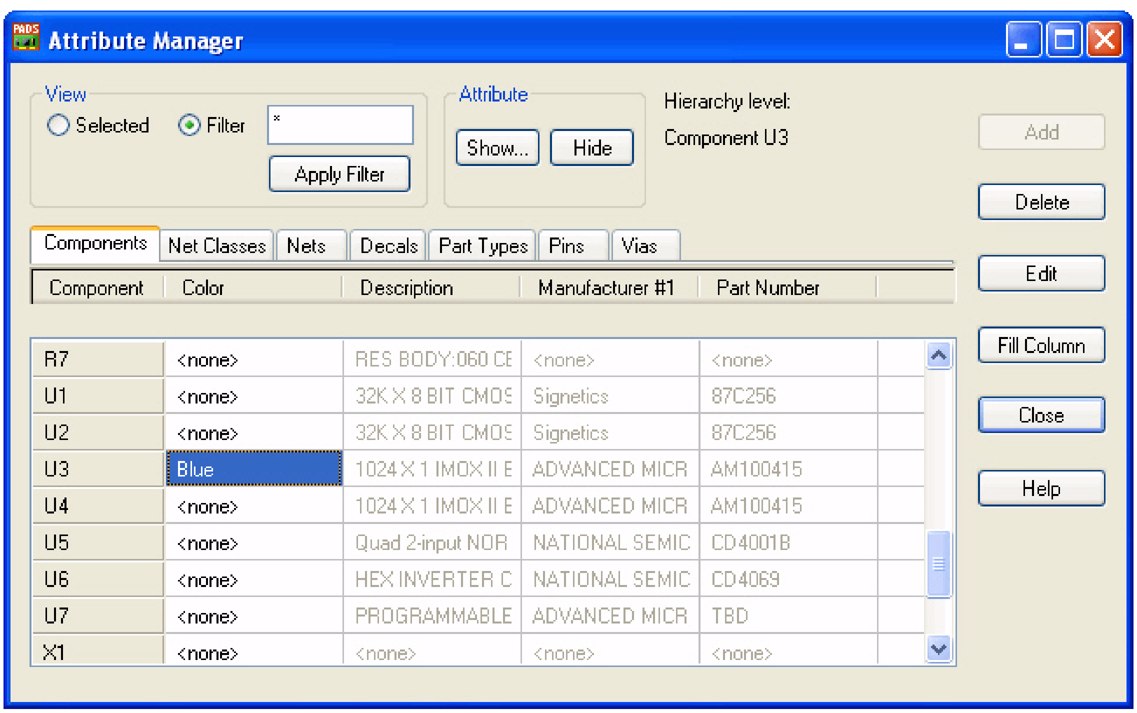


Table 45-42. Attribute Manager Dialog Box Contents

Name	Description
View area	Specifies what to show in the spreadsheet tabs. • **Selected**—Shows the attributes selected in the design. • **Filter**—Narrows down your unassigned decals list. **Tip:** You can use wildcards in this box. **Tip:** Attributes are shown in the spreadsheet only after you've selected the ones to show from the Show Attributes dialog box.
Apply Filter	Executes the filter arguments.
Show	Opens the Show Attributes dialog box.
Hide	Hides the selected column from the spreadsheet tabs.
Hierarchy level	The order in which attribute values have priority. The object at the top of the list has the highest priority.
Spreadsheet tabs	Lists attributes based on selections made in the View area, the attribute area, and which tab you're on: Components, Net Classes, Nets, Decals, Part Types, Pins, and Vias.
Add	Adds an attribute to the selected cell.
Delete	Removes the attribute from the selected cell.
Edit	Makes the selected cell available for editing.
Fill Column	Populates all cells in a column with what you've added to one cell in that column.

Related Topics

Using the Attribute Manager

Attribute Properties Dialog Box, Objects Tab

Use the Objects tab to assign the attribute to objects and set up the hierarchy for the attribute.

If the attribute is a system attribute, all of the options in this dialog box are unavailable, except the System attribute option on the Objects tab (which you would use to turn off the system attribute flag).

Accessing

- **Edit** menu > **Attribute Dictionary** > **Select attribute** > **Properties button** > **Objects tab**

  or

- **Edit** menu > **Attribute Dictionary** > **New button** > **Objects tab**

Figure 45-43. Objects Tab

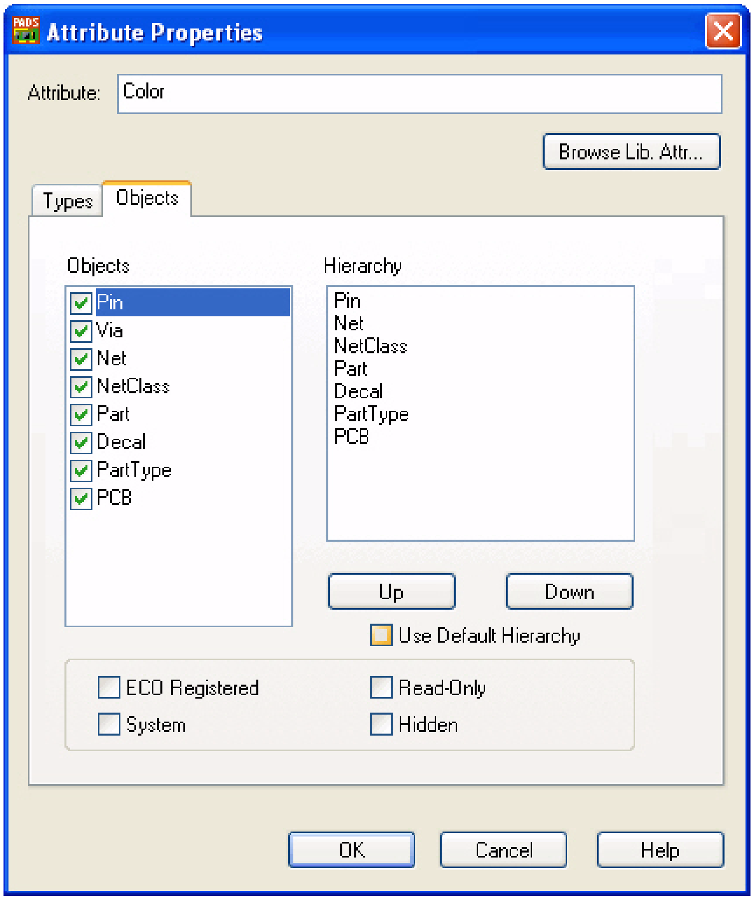


Table 45-43. Objects Tab Contents

Name	**Description**
Attribute	The name of the attribute.
Browse Lib Attr	Opens the Browse Library Attributes dialog box.
Objects	The list of objects you can make available to the attribute. NetClass—refers to a net class created in the Class Rules dialog box. Part—refers to a component in the design. PCB—refers to the design as a whole. **See also:** Assigning Attributes in the *Concepts Guide*.
Hierarchy	The order in which attribute values have priority. The object at the top of the list has the highest priority.

Table 45-43. Objects Tab Contents (cont.)

Name	Description
Up/Down	Moves the object hierarchy up or down. The object at the top of the list has the highest priority.
Use Default Hierarchy	Click to clear if you want to change the default hierarchy. **Restriction:** Hierarchy modification is restricted. PADS Layout automatically sets a certain logical hierarchy. For example, you cannot place Net Class above Net in the hierarchy because a net cannot usually be derived from a Net Class. PADS Layout automatically places Net above Net Class in the hierarchy.
ECO Registered	Allows you to specify if the attribute is ECO registered. If so, changes to the attribute are registered in the ECO file. If this check box is clicked, you can only modify attributes when the ECO toolbar is open (ECO mode).
System	Shows whether the attribute is a system attribute. System attributes are used by PADS Layout, an external program, or an Automation script (such as Sax Basic). The System check box prevents you from modifying an attribute that is internally set by, and critical to, PADS Layout operation.
Read-Only	Shows whether the attribute value is read-only, which means it cannot be changed outside of the library. However, you can modify the attribute properties. If you want to modify the attribute value, do so in the library.
Hidden	Hides the attribute, so that it is not visible or editable. It will not appear in any dialog boxes.

Related Topics

Attribute Properties Dialog Box, Types Tab

Setting Attribute Properties

Attribute Properties Dialog Box, Types Tab

Use the Type tab to set the attribute type.

Accessing

- **Edit** menu > **Attribute Dictionary** > **Select attribute** > **Properties button**

  or

- **Edit** menu > **Attribute Dictionary** > **New button**

The Types tab controls change depending on what you have selected. The major differences are:

- Figure 45-44: Types Tab - Number
- Figure 45-45: Types Tab - Measure
- Figure 45-46: Types Tab - List

Figure 45-44. Types Tab - Number

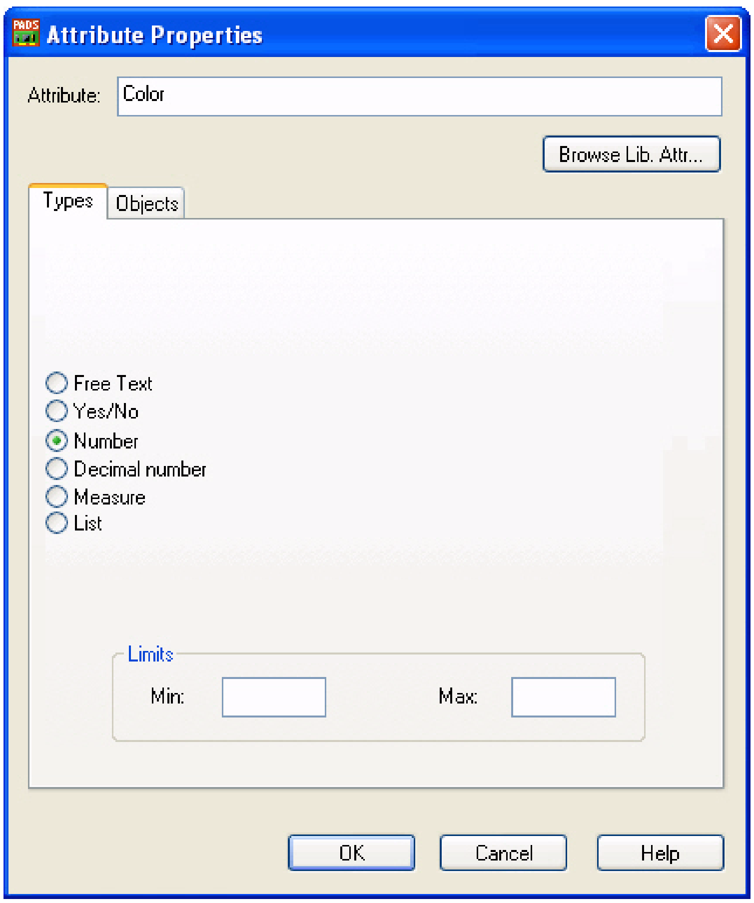


Figure 45-45. Types Tab - Measure

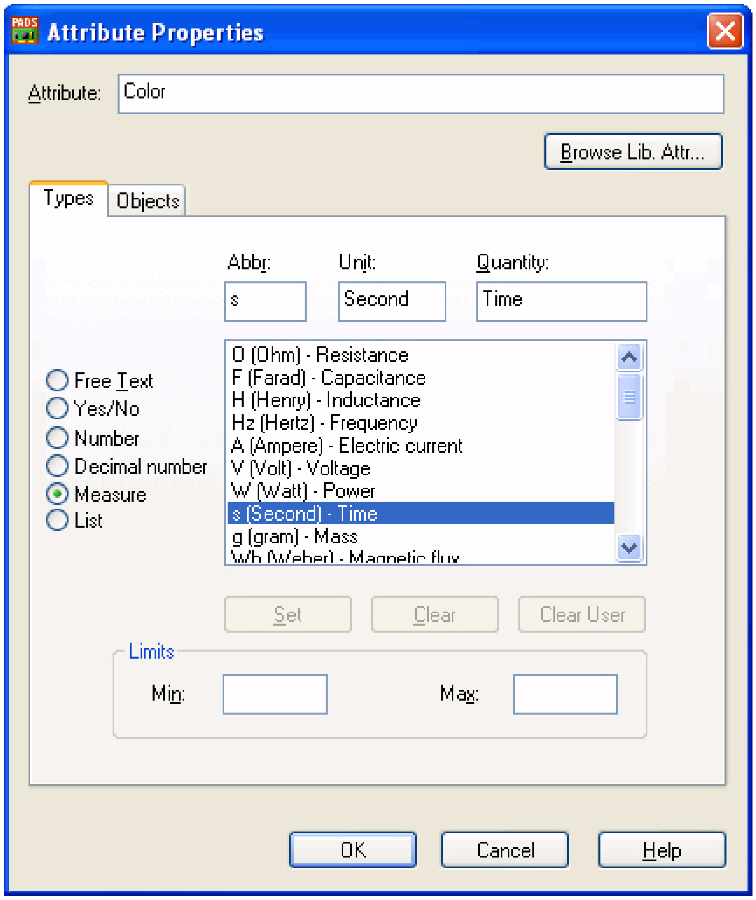


Figure 45-46. Types Tab - List

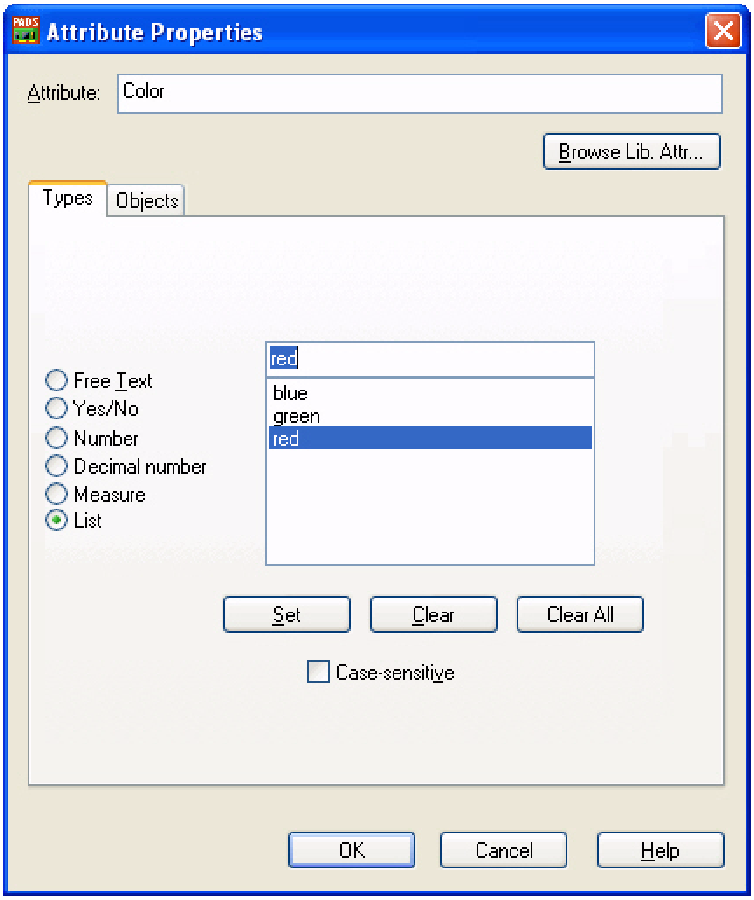


Table 45-44. Types Tab Contents

Name	Description
Attribute	The name of the attribute.
Browse Lib Attr	Opens the Browse Library Attributes dialog box.
Free Text	Allows you to type a text string as the attribute value.
Yes/No	Creates a list box where you can select Yes or No as the attribute value.
Number	Allows you to type a number as the attribute value.
Decimal number	Allows you to type a decimal number for the attribute value.
Measure	Allows you to determine a measurement for the attribute value. It is a physical value associated with units.
List	Allows you to create a list from which you choose the value.

Table 45-44. Types Tab Contents (cont.)

Name	Description
Case-sensitive	preserves the letter case of List entries. **Tip:** This setting affects sorting and matching in the Find dialog box and the Attribute Manager dialog box.
Limits Min/Max	Specifies a range for the Measure attribute type. Type in the Min and Max boxes to set the range. PADS Layout checks against the Limits area values.
Abbr	Abbreviation to use for the unit.
Unit	The name of the unit.
Quantity	The quantity, or what it measures.
List list	Specifies a user-defined list for the attribute.
Set	Adds the item to the list.
Clear	Removes the selected item from the list.
Clear All	Removes all items from the list.
Clear User	Removes all user-defined units from the list. All default units remain in the list.

Related Topics

Attribute Properties Dialog Box, Objects Tab

Setting Attribute Properties

Auto Placement Prompt

Use the Auto Placement prompt to control the interruption of the cluster placement process.

Accessing

- **Tools** menu > **Cluster Placement** > **Place Clusters** > **Run** > **Interrupt**

Figure 45-47. Auto Placement Prompt

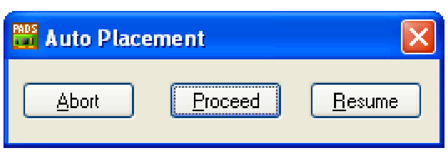


Table 45-45. Auto Placement Prompt Contents

Name	Description
Abort	Quits the Cluster Placement process. You are promted to allow the adjusted placement where clicking No will undo any placement.
Proceed	Continues with the original cluster placement process.
Resume	You are prompted to allow the process to proceed based on the already adjusted placement.

Related Topics

Cluster Placement Status Dialog Box

Cluster Placement Dialog Box

AutoRenumber Dialog Box

Once you complete the placement of a design, you can renumber the parts to make the reference designators follow a specific pattern. This helps you find components on the fabricated board. AutoRenumbering reassigns the reference designators to one of the eight provided patterns.

Accessing

- On the ECO toolbar, click Auto Renumber.

Figure 45-48. AutoRenumber Dialog Box

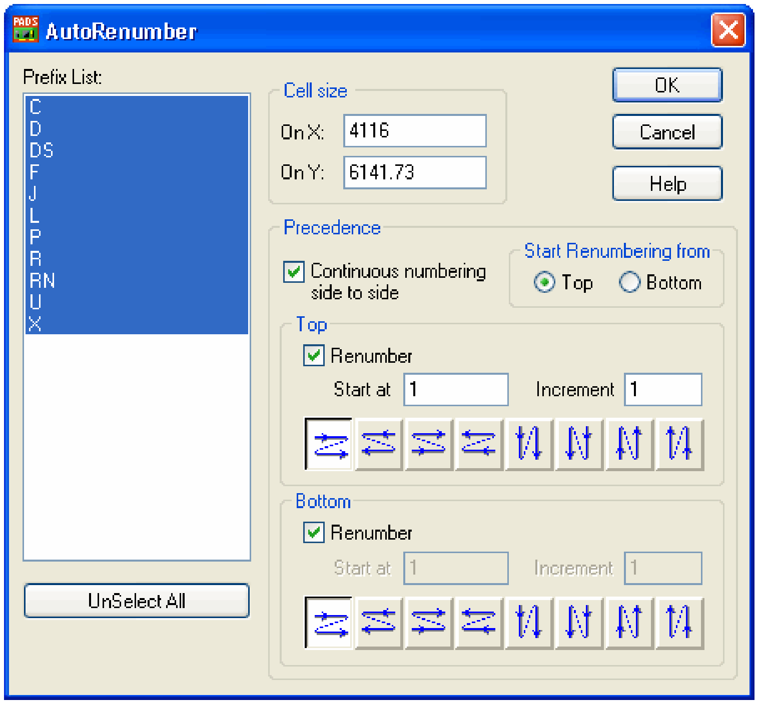


Table 45-46. Contents of AutoRenumber Dialog Box

Field or Button	Description
Prefix List	Lists the standard prefix for each part in the design. You can select individual prefixes to renumber associated parts. You can use the Shift+click, Ctrl+click, or click and drag shortcuts to select multiple items.
Select All	The Select All button changes to Unselect All when prefixed are selected. You can renumber all parts by clicking the Select All button or clear all prefixes by clicking Unselect All.
Cell size	Enter the cell size of the cell for the part-numbering sweep. Renumbering sorts and renumbers parts one cell at a time starting at the corner indicated by the Directional Pattern setting. For circular board outlines, an invisible bounding box determines the location of the first cell. The board outline width is taken into account when the reference point for positioning user-specified cells is calculated. Cells are counted from the outer edge of the board outline. **See also**: AutoRenumber Sweeps

Table 45-46. Contents of AutoRenumber Dialog Box

Field or Button	Description
Continuous numbering side to side	Controls numbering between sides. Select this option to number all parts sequentially beginning with the *Start Renumbering from layer and continuing to the opposite side. Clear the check box to if you want to specify a separate start at number and increment value for each side of the board.*
Start Renumbering from	Indicate the side on which to start renumbering the parts. **Restriction:** This feature is unavailable when parts are only on one side of the board or if the *Continuous numbering side to side* check box is cleared
Renumber (Top or Bottom)	Select the check box to renumber the corresponding side of the board. **Restriction:** This feature is unavailable when parts are not on the layer.
Start at (Top or Bottom)	Specify the starting number for renumbering for the corresponding side of the board. Renumbered parts maintain their alphabetic prefix and are assigned a numeric suffix beginning with this value. **Restriction:** This feature is unavailable when *Continuous numbering side to side* is enabled and it is not the selected layer of the *Start Renumbering from* setting.
Increment (Top or Bottom)	Specify the value by which to increment the reference designators. **Restriction:** This feature is unavailable when *Continuous numbering side to side* is enabled and it is not the selected layer of the *Start Renumbering from* setting.
Directional patterns (Top or Bottom)	Select the button that shows the starting location and direction in which you want to renumber the board. **See also**: AutoRenumber Sweeps

Related Topics

Changing the Reference Designators of Multiple Components in ECO Mode (Autorenumbering)

AutoRenumber Sweeps

Backward Annotation Dialog Box

In DxDesigner Link, backward annotation sends data from a PADS Layout design file to a DxDesigner schematic and updates the schematic to match the PADS Layout design.

Use the Backward Annotation dialog box to specify the layout design data to use in backward annotation.

Tip: To avoid unexpected changes during backward annotation, consider comparing data before you back-annotate.

Accessing

- **Tools** menu > **DxDesigner > Backward From PCB button**

Figure 45-49. Backward Annotation Dialog Box

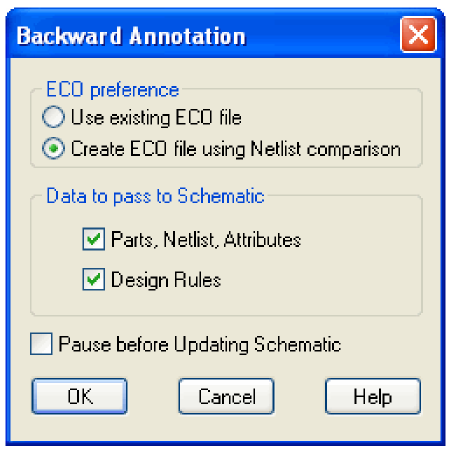


Table 45-47. Backward Annotation Dialog Box contents

Name	Description
ECO preference area	• **Use existing ECO file**—Specifies to use an exiting ECO file. **Tip:** The backward annotation operation first searches for a file with the same name as the layout design but with an .eco extension. If the operation does not find such a file, you can select an ECO file. • **Create ECO file using Netlist comparison**—Specifies to have DxDesigner compare the PADS Layout design to the schematic and create a new ECO file.

Table 45-47. Backward Annotation Dialog Box contents (cont.)

Name	Description
Data to pass to Schematic area	Specifies the data to send to the DxDesigner schematic. You can send: • Parts, netlist, and attributes names and values • Design rules **Tip:** To include design rules in the forward annotation operation, select the Design Rules option, even if you already selected **Compare Design Rules** on the Preferences tab.
Pause before Updating Schematic	Specifies that you want to review the ECO file containing changes that the update will make to the DxDesigner schematic.

Related Topics

Back Annotating from PADS Layout to DxDesigner

Basic Script Editor Dialog Box

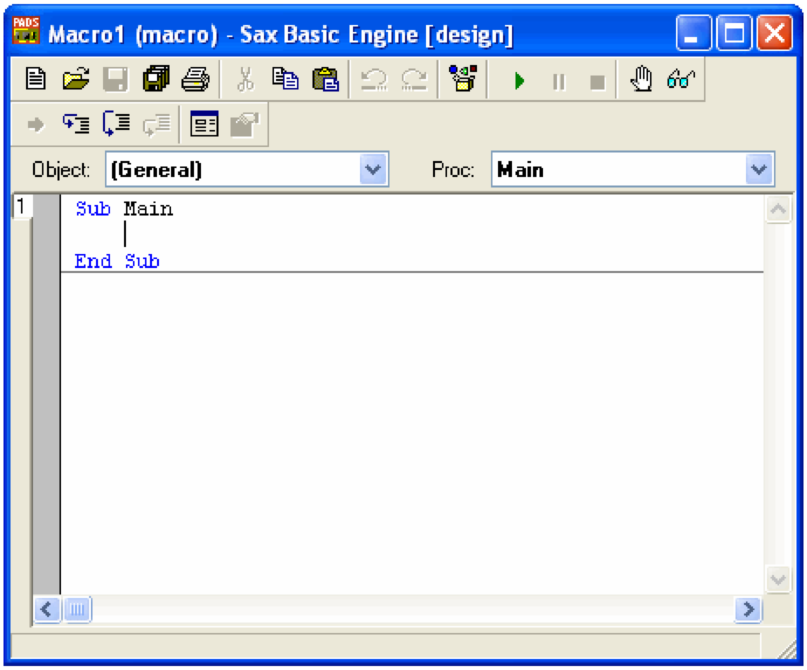
Basic is a simple scripting language. Like many Windows applications, such as Microsoft Word and Excel, PADS applications include Basic capabilities to allow users to customize their applications using a standard scripting language.

You can use the Basic Script Editor to create, edit, run, and troubleshoot Basic scripts from PADS applications.

Accessing

- **Tools menu** > **Basic Scripts** > **Basic Script Editor**.

Figure 45-50. Basic Script Editor



Related Topics

Managing Scripts
Creating Scripts
Running Scripts
Basic Scripting
Debugging Scripts
The Macro Language
Router Basic Scripting

Basic Scripts Dialog Box

Use the Basic Scripts dialog box to access your scripts. In this dialog box, you can load and run your most commonly used scripts. You can also unload those scripts you don't use often. This dialog box runs only existing sample scripts.

Accessing

- **Tools menu > Basic Scripts > Basic Scripts**

Figure 45-51. Basic Scripts Dialog Box

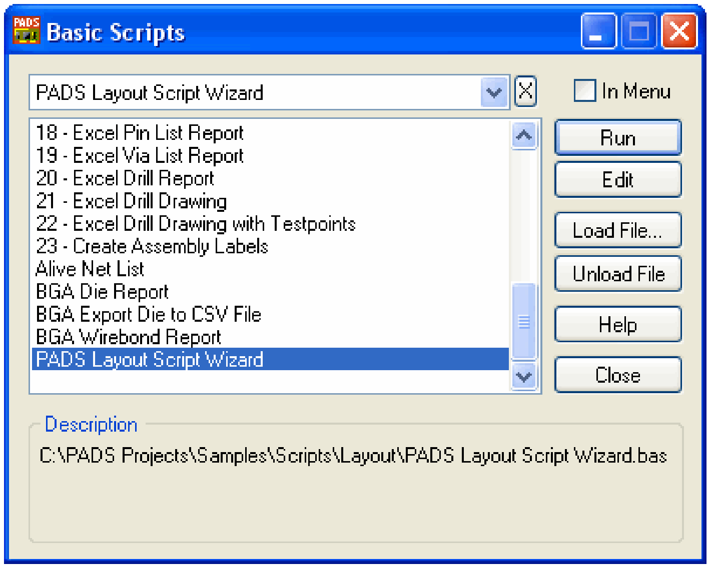


Table 45-48. Basic Scripts Dialog Box contents

Name	Description
Script lists	Lists the loaded scripts. The Script dropdown list and the Script list are synchronized
	When you click the X button, the lower half of the dialog box “disappears”; leaving only the top left corner of the dialog box for interaction. When in reduced mode, click a script from the list and press Enter, or double-click on a script in the list.
In Menu	Adds scripts to the Basic Scripting submenu from the Tools menu, where you can run often-used scripts directly from the menu without opening the Basic Scripts dialog box.
Run	Runs the currently selected script.
Edit	Opens the Sax Basic Engine dialog box where you can edit the selected script.
Load File	Loads scripts into the list. You can load up to 32,767 scripts. Scripts are not compiled when they are loaded; they are compiled when you run them. This list of scripts you load into this dialog box is saved in the VBScripts.ini file, so these scripts load every time you open the Basic Scripts dialog box.
Unload File	Unloads the selected script from the dialog box.

Related Topics

Basic Script Editor Dialog Box

BGA Route Wizard Dialog Box

When you select the BGA Route Wizard button on the BGA toolbar, the BGA Route Wizard dialog box appears. Use the BGA Route Wizard dialog box to generate connections only, or generate connections and routes.

Restriction: This information applies to the BGA toolkit only.

When you open the BGA Route Wizard dialog box for the first time (per design) the Die Ref Des and BGA Ref Des drop down list boxes automatically display reference designators based on the logic families in the current design.

> **Tip:** For die reference designators, the BGA Route Wizard looks for parts having either a die part or flip chip special purpose setting. If more than one of these are found, one is randomly selected. If none is found, a component is randomly selected.
>
> **Tip:** For BGA reference designators, the BGA Route Wizard looks for a BGA logic family. If no die with this logic family is found, a component is selected randomly. If more than one die has this logic family, one of the dies in that logic family is randomly selected.

When you click either Run or OK, the selections you make in the BGA Route Wizard dialog box are stored in the design.

The Die Wizard has 4 tabs:

- Connections
- Routing
- Select Pads
- BGA Fanouts

The availability of tabs depends on whether you select Generate Connections or Generate Connections and Route from the Action area.

- If you select Generate Connections, the Routing tab grays.
- If you select Generate Connections and Route, the Connections tab grays.
- If you are creating a single-sided design, the BGA Fanouts tab grays.

Accessing

- **BGA Toolbar** button > **Route Wizard button**

Figure 45-52. BGA Route Wizard Dialog Box

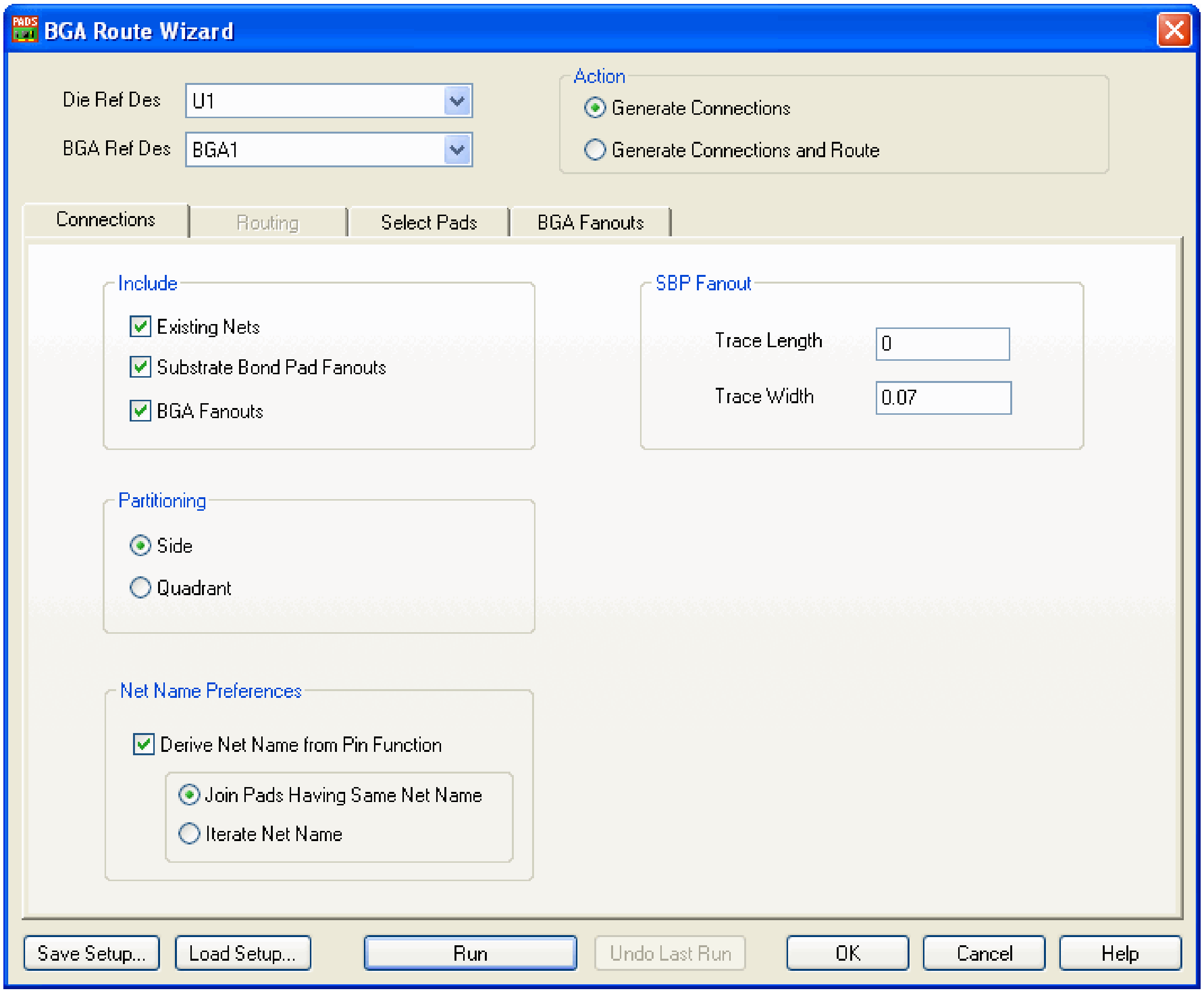


Table 45-49. BGA Route Wizard Dialog Box contents

Name	Description
Die Ref Des list	Displays the die reference designator when you open the BGA Route Wizard dialog box for the first time (per design), based on the logic families in the current design. The BGA Route Wizard looks for parts having either a die part or flip chip special purpose setting. If more than one of these are found, one is randomly selected. If none is found, a component is randomly selected.

Table 45-49. BGA Route Wizard Dialog Box contents (cont.)

Name	Description
BGA Ref Des list	Displays the BGA reference designator when you open the BGA Route Wizard dialog box for the first time (per design), based on the logic families in the current design. The BGA Route Wizard looks for a BGA logic family. If no die with this logic family is found, a component is selected randomly. If more than one die has this logic family, one of the dies in that logic family is randomly selected.

Action area	• **Generate Connections—**Generates logical connections between die and BGA component pins. Connections are not routes; however, BGA fanouts and SBP fanouts are generated if specified in the Connections tab.
	• **Generate Connections and Route—**Generates logical connections between component pins and creates trace patterns to route them. During processing, the following occurs:
	  • BGA fanouts are generated. This is performed on double-sided designs only.
	  • Serpentine routes and plating tails are generated.
	  • SBPs are logically assigned to the ends of serpentine traces and logical connections between die pins, and BGA pins are generated.
	  • SBP fanouts and any-angle coupling traces are generated.
	  • The BGA Route Wizard generates route patterns on die and BGA layers only. The BGA layer contains only BGA fanouts. All other parts of route patterns (serpentine routes and plating tails, SBP fanouts and any-angle coupling traces) are created on the die layer.

tabs	• Connections • Routing • Select Pads • BGA Fanouts
Save Setup button	Saves the selections you make in the BGA Route Wizard dialog box and stores them in a text file with a .brw extension. This file is stored in the \PADS Projects folder.
Load Setup button	Opens .brw files that contain BGA Route Wizard settings.
Run button	Runs the routing process based on the selections made in the BGA Route Wizard dialog box.
Undo Last Run button	Returns the design to its state prior to route processing. **Undo Last Run** is unavailable until you run the routing process.

Figure 45-53. BGA Route Wizard, Connections tab

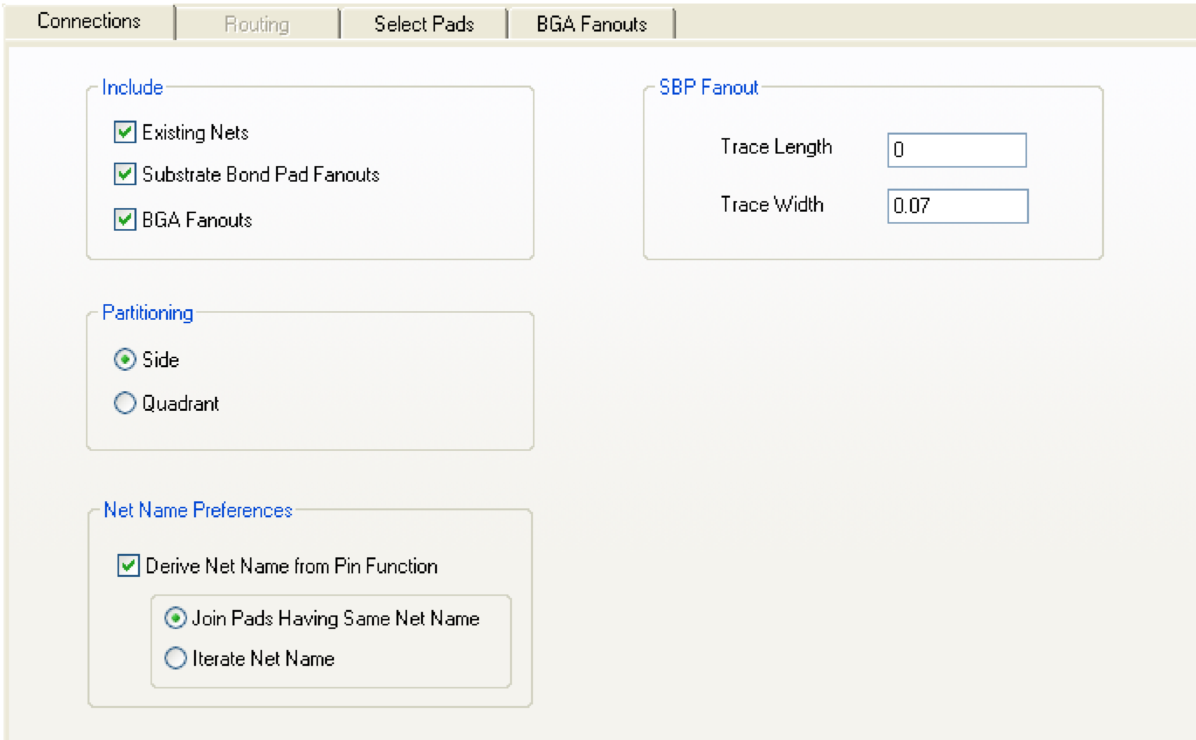


Table 45-50. Connections tab contents

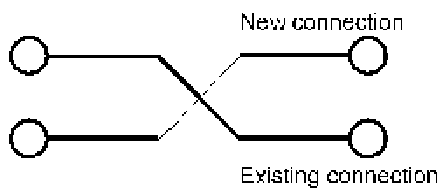
Name	Description
Existing Nets	Controls predefined connections processing. • When this option is selected, predefined connections are processed so BGA and SBP fanouts are generated for them. • When this option is cleared, Die and BGA pins that already belong to signals are excluded from processing. BGA fanout and SBP fanout processing, if specified in the Connections tab, are also not performed for existing connections.  The BGA Wizard interprets pins as obstacles and routes around them. If a new trace interferes with an existing trace, no shorts are placed and the new connection is unrouted. If a BGA or SBP pin has an existing fanout, it is used during processing. **Tip:** You can exclude some predefined connections using the **Select Pads** tab.
Substrate Bond Pad Fanouts	Creates SBP fanouts during connection generation. Specify the escape length and trace width in the SBP Fanout area.
BGA Fanouts	Creates BGA fanouts during connection generation. This option is available only for double-sided packages.
Partitioning area	Controls the pad partitioning displayed in the Select Pads tab; creating sets of die pads and BGA pads to connect.: • **Side—**Die is partitioned by sides so that pad sets are: **Right**, **Left**, **Top**, and **Bottom**. • **Quadrant—**Die is partitioned into quadrants: **Top Left**, **Top Right**, **Bottom Left**, and **Bottom Right**. **See also:** Partitioning a Die
Derive Net Name from Pin Function	Controls new net name generation. Select this option to use a pin's function as a basis for naming a new net.

Table 45-50. Connections tab contents (cont.)

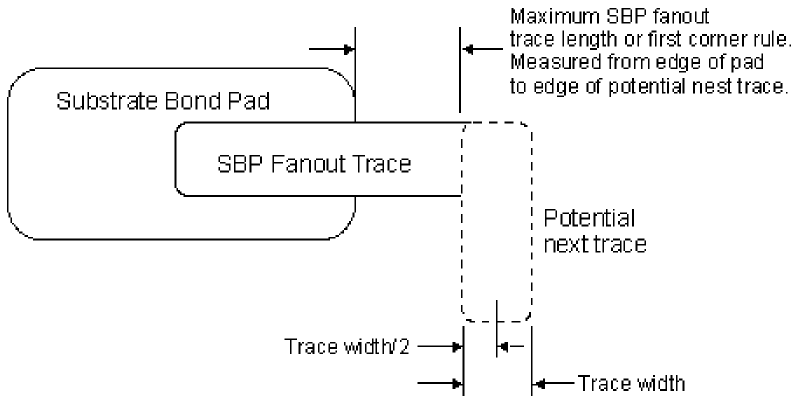
Name	Description
Net Name Preferences area	• **Join Pads Having Same Net Name—**Combines die pads with the same function name into a single net. • **Iterate Net Name—**Generates separate nets for each die pad with the same name; for example, several GND die pads would have nets named GND1, GND2, and so on. Net name preferences are shared between the Connections tab and the Routing tab. If you change net name preferences in one tab, the same changes occur in the other tab.
Trace Length	Defines the SBP fanout trace length. The trace length is measured from the edge of the pad to the end of the trace. If a value of zero is entered the trace is not created. The BGA Route Wizard also takes the **SMD to Corner** rule into account when generating this trace. This rule can make the BGA Route Wizard generate a longer trace than requested in the **Trace Length** box. 
Trace Width	Defines the SBP fanout trace width. The trace width must adhere to the minimum and maximum rule set in the Clearance Rules dialog box. **See also:** Design Rule Hierarchy

Figure 45-54. BGA Route Wizard, Routing tab

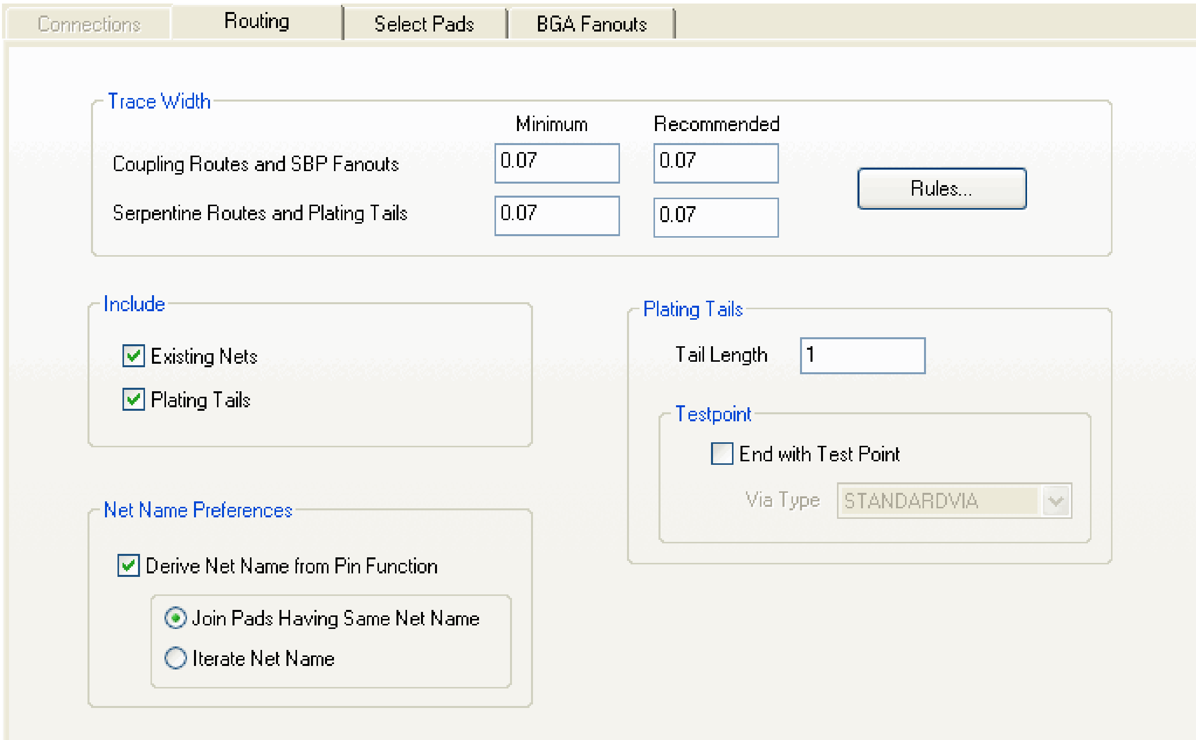


Table 45-51. Routing tab contents

Name	Description
Coupling Routes and SBP Fanouts	Sets the trace width rules during route pattern creation for SBP fanouts and any-angle coupling routes. Type a value into the **Minimum** and **Recommended** boxes. Values must be between the Minimum and Maximum trace width values defined in the Clearance Rules dialog box. The BGA Route Wizard uses the **Recommended** value whenever possible, and uses the **Minimum** value only for connections that it cannot route using the Recommended value.
Serpentine Routes and Plating Tails	Sets the trace width rules during route pattern creation for serpentine patterns and plating tails. Type a value into the **Minimum** and **Recommended** boxes. Values must be between the Minimum and Maximum trace width values defined in the Clearance Rules dialog box. **See also:** Design Rule Hierarchy The BGA Route Wizard uses the **Recommended** value whenever possible and uses the **Minimum** value only for connections that it cannot route using the **Recommended** value.

Table 45-51. Routing tab contents (cont.)

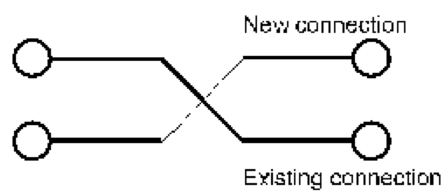
Name	Description
Rules button	Opens the Default Rules dialog box.
Existing Nets	Controls predefined connections processing. • When this option is selected, predefined connections are processed so BGA and SBP fanouts are generated for them. • When this option is cleared, Die and BGA pins that already belong to signals are excluded from processing. BGA fanout and SBP fanout processing, if specified in the Connections tab, are also not performed for existing connections.  The BGA Wizard interprets pins as obstacles and routes around them. If a new trace interferes with an existing trace, no shorts are placed and the new connection is unrouted. If a BGA or SBP pin has an existing fanout, it is used during processing. **Tip:** You can exclude some predefined connections using the **Select Pads** tab.
Plating Tails	Specifies to create plating tails for serpentine traces.
Derive Net Name from Pin Function	Controls new net name generation. Select this option to use a pin's function as a basis for naming a new net.
Net Name Preferences area	• **Join Pads Having Same Net Name—**Combines die pads with the same function name into a single net. • **Iterate Net Name—**Generates separate nets for each die pad with the same name; for example, several GND die pads would have nets named GND1, GND2, and so on. Net name preferences are shared between the Connections tab and the Routing tab. If you change net name preferences in one tab, the same changes occur in the other tab.

Table 45-51. Routing tab contents (cont.)

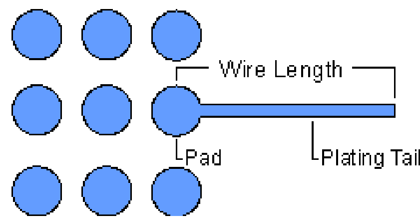
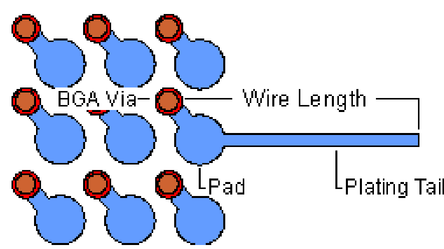
Name	Description
Tail Length	Defines the length of the plating tail wire from the outside row of a BGA (single-sided package) or from a BGA fanout via (double-sided package) to the end point of the plating tail. On single-sided packages the wire length is measured from the center of the BGA pad to the end of the plating tail, even if a test point via is added.  On double-sided packages the wire length is measured from the center of the BGA via to the end of the plating tail, even if a test point via is added.  **Tip:** The BGA Route Wizard extends plating tails through the board outline, if necessary, without generating an error in the BGA Route Wizard Report. However, **Clearance Checking** will produce errors in this circumstance.
End with Test Point	Creates a test point at the end of each new plating tail. **Tip:** On the File menu, click Reports. Run the DFT Extended test point report to check that all traces have a plating tail.
Via Type list	Defines the via type to use for the test point. The list contains the vias that are defined in the PCB Decal Editor for each part.

Figure 45-55. BGA Route Wizard, Select Pads tab

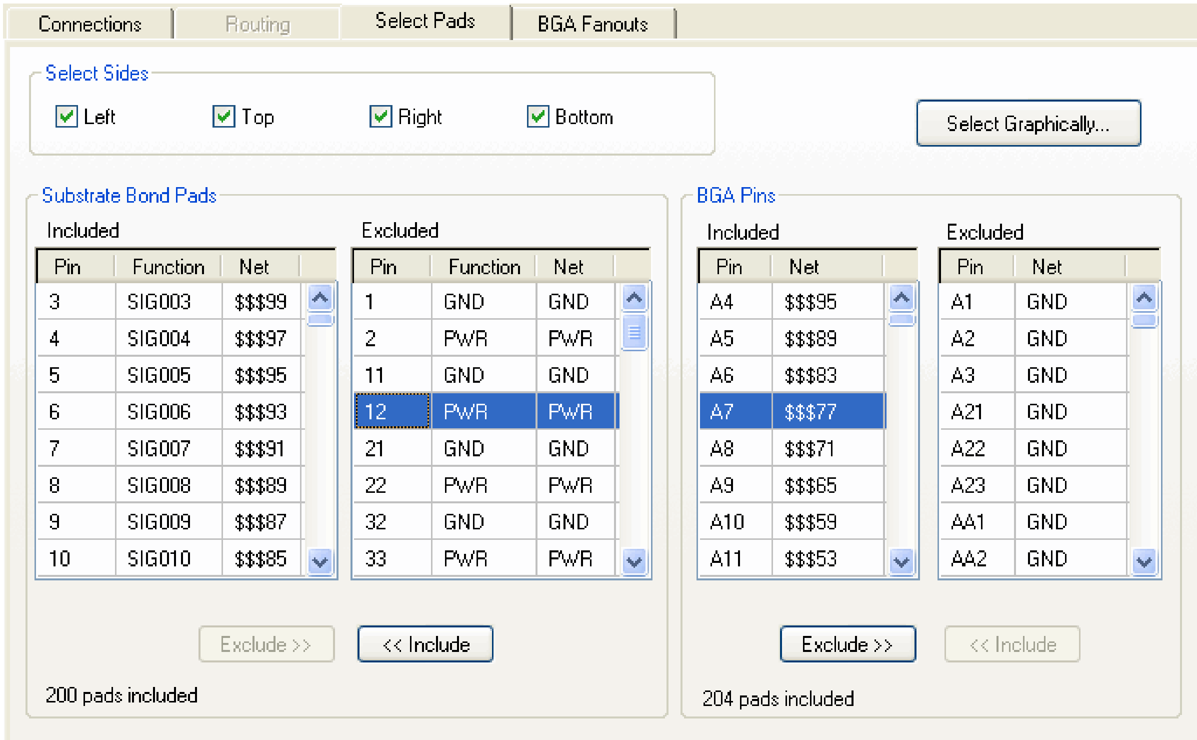


Table 45-52. Select Pads tab contents

Name	Description
Select Sides/Select Quadrants	The available partitioning type depends on whether the **Generate Connections** or **Generate Connections and Route** option is selected in the Action area. Use the partition check boxes to select partition sets. The available Side partition sets are **Right**, **Left**, **Top**, and **Bottom**. The available Quadrant partition sets are **Top Left**, **Top Right**, **Bottom Left**, and **Bottom Right**. Only the pins from selected sets appear in the **Substrate Bond Pad** and **BGA Pins** lists. **See also:** BGA Operations
Select Graphically button	Opens the Select Graphically dialog box. **See also:** To Use Graphical Selection mode
Substrate Bond Pads area	Lists the substrate bond pads to include in or exclude from processing. **Tip:** Die and BGA pads that already belong to signals are excluded from processing unless **Existing Nets** on the Connections tab is selected.

Table 45-52. Select Pads tab contents (cont.)

Name	Description
BGS Pins list	Lists the BGA pins to include in or exclude from processing. **Tip:** Die and BGA pins that already belong to signals are excluded from processing unless **Existing Nets** on the Connections tab is selected.
Exclude button	Removes selected pins from the **Included** list and adds them to the **Excluded list.**
Include button	Removes selected pins from the **Excluded** list and adds them to the **Included** list**.**
Number of Included Pads	Displays the number of pads currently listed in the **Included** list.

Figure 45-56. BGA Route Wizard, BGA Fanouts tab

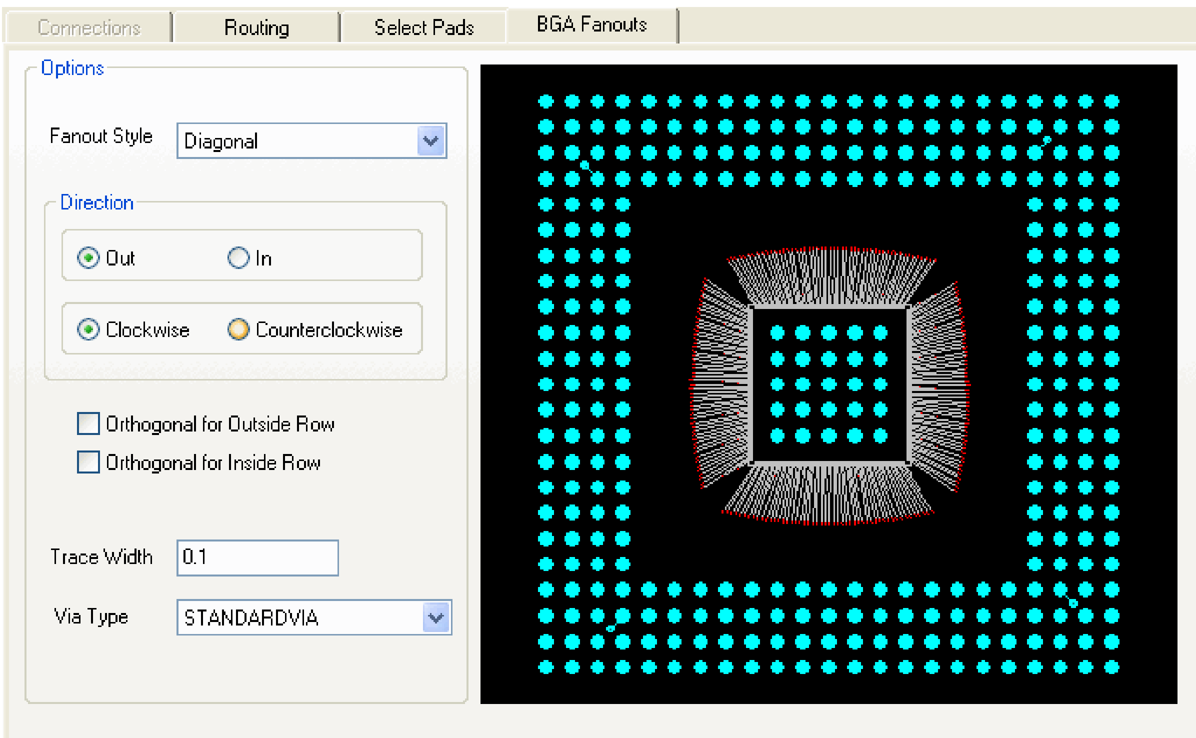


Table 45-53. BGA Fanout tab contents

Name	Description
Fanout Style list	Sets the fanout pattern. The available fanout styles depend on the grid array geometry. Herringbone and Diagonal are available for regular (non-staggered) arrays. Vortex and Double Vortex are available for staggered arrays.
Direction area	• **Out—**Turns BGA fanouts to the outside of the design. • **In—**Turns BGA fanouts to the inside of the design. • **Clockwise—**Turns BGA fanouts in a clockwise direction. • **Counterclockwise—**Turns BGA fanouts in a counterclockwise direction. If either **Orthogonal for Outside Row** or **Orthogonal for Inside Row** is selected, the direction options only affect BGA fanouts within the central rows when more than two rows of BGA pads exist.
Orthogonal for Outside Row	Turns BGA fanouts on the outside row outward at an orthogonal angle with respect to the die. If this option is selected, the Direction options have no effect on the outside row.
Orthogonal for Inside Row	Turns BGA fanouts on the inside row inward at an orthogonal angle in respect to the die. If this option is selected, the Direction options have no affect on the inside row.
Trace Width	Defines the trace width to use for BGA fanouts. The trace width must adhere to the minimum and maximum rule set in the Clearance Rules dialog box. **See also:** Design Rule Hierarchy
Via Type list	Defines the via type to use for BGA fanouts. The list contains the vias that are defined using **Pad Stacks** on the **Setup** menu for this design.
Preview Window	Shows the fanout pattern defined by the current settings on the BGA Fanouts tab. In this window you can see that pads excluded from processing on the Select Pads tab do not receive a fanout pattern. **Tip:** If **Existing Nets** on the Connections tab or Routing tab is selected, all pads, including pads that are not currently part of a predefined connection, are assigned a fanout pattern.

Related Topics

Using the BGA Route Wizard

To Create a Wire Bond Fanout

BGA Operations

BoardSim Dialog Box

Use the BoardSim dialog box to export the design in the form of a HYP file and open it in BoardSim. For information about the format of the HYP file and creating a "BoardSim-friendly" design, see the HyperLynx online Help. PADS Layout passes the Value, Tolerance, Voltage, HyperLynx, and PowerGround attributes to the HYP file. BoardSim uses these attributes to obtain values for resistors and capacitors, and to transfer information about fixed voltage nets.

Restrictions

- Only HyperLynx v8.0 or newer can open the file that is exported by this dialog box. The .hyp file that is created is a v2.34 format file.
- You would use this dialog box only if you have HyperLynx BoardSim installed since it's the quickest method. If you don't have it installed and you still need to create the file to send it to another computer, use the File > Export method of only producing the .hyp file. See HYP Export Dialog Box.

Accessing

- **Tools** menu > point to Analysis > click Signal/Power Integrity

Figure 45-57. BoardSim Dialog Box

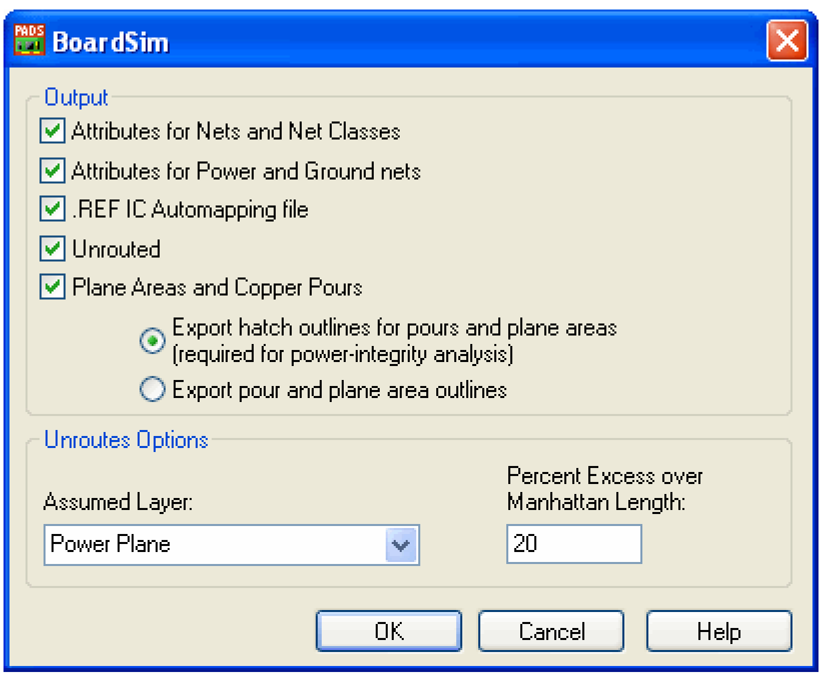


Table 45-54. BoardSim Dialog Box Contents

Name	Description
Output area	
Attributes for Nets and Net Classes	Specifies to export attributes for nets and net classes.
Attributes for Power and Ground nets	Specifies to export attributes for power and ground nets. **Restriction:** This check box is only available if a net has both the PowerGround attribute set to Yes, and the Voltage attribute with a value.
.REF IC Automapping file	Specifies to create a HyperLynx .ref file, which maps IC reference designators in the design to the BoardSim models that represent the ICs. BoardSim uses the mappings to automatically load IC models when you select a net for simulation. **Restriction:** This check box is only available if a component has the HyperLynx.Model attribute with a value.
Unrouted	Specifies to export unrouted nets.
Plane Areas and Copper Pours	Specifies to export plane areas and copper pours. • **Export hatch outlines for pours and plane areas**—Specifies to export hatch outlines for pours and plane areas. **Requirement:** This setting creates extra constructs in the .hyp file and is required for power-integrity analysis. • **Export pour and plane area outlines**—Specifies to export pour and plane area outlines.
Unroutes Options area **Tip:** Available only if Unrouted is selected in the Output area.	
Assumed Layer	Specifies the layer on which to implement the unrouted nets.
Percent Excess over Manhattan Length	Specifies the value to estimate the routing lengths. **Tip:** This value adds a percentage of the Manhattan length to the route length, to account for indirect routing paths. Net lengths are based on the Manhattan distance between pin pairs, which is Delta X plus Delta Y.

Related Topics

HYP Export Dialog Box

Creating HyperLynx BoardSim - HYP Files

Browse for Special Symbols Dialog Box

Use the Browse for Special Symbols dialog box to assign one or more CAE decals, or Special Symbols, to a pin type. Special Symbols indicate the function of the connector pin in PADS Logic.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **Connector** tab > Double-click Special Symbol cell > **Browse** button

Figure 45-58. Browse for Special Symbols Dialog Box

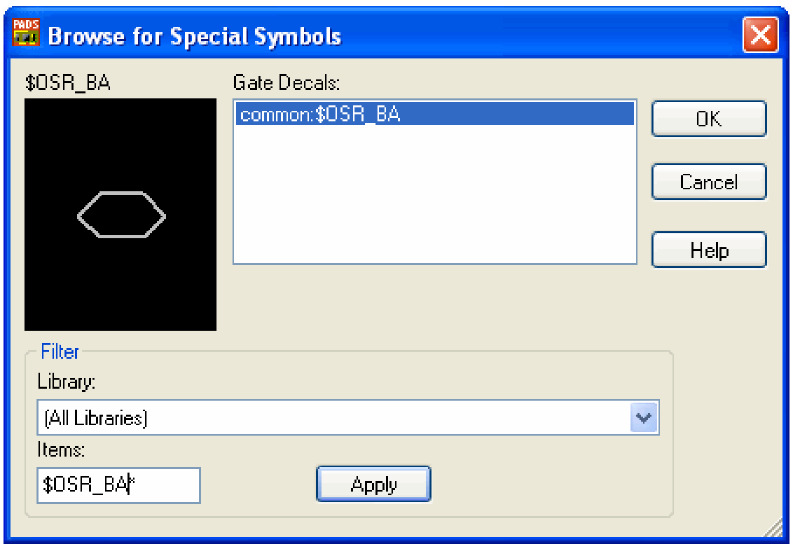


Table 45-55. Browse for Special Symbols Dialog Box Contents

Name	Description
Preview area	Shows the item selected in the Gate Decals list.
Gate Decals	List of available Gate Decals.
Library list	Lists all libraries available to you.
Items	Narrows down your Gate Decals list. **Tip:** You can use wildcards in this box.
Apply	Executes the filter arguments.

Related Topics

Part Information Dialog Box, Connector Tab

Creating a Connector Part Type

Assigning Special Symbols to a Connector

Browse Library Attributes Dialog Box

Use the Browse Library Attributes dialog box to select an attribute from a list of all part and decal attributes available to the design.

Accessing

- **File** menu > **Library** > Library Manager dialog box > **Attr Manager** button > **Browse Lib. Attr** button
- **File** menu > **Library** > Library Manager dialog box > **Attr Manager** button > **Add Attr** button > **Browse Lib. Attr** button
- Attribute Properties dialog box **> Browse Lib. Attr. button**

Figure 45-59. Browse Library Attributes Dialog Box

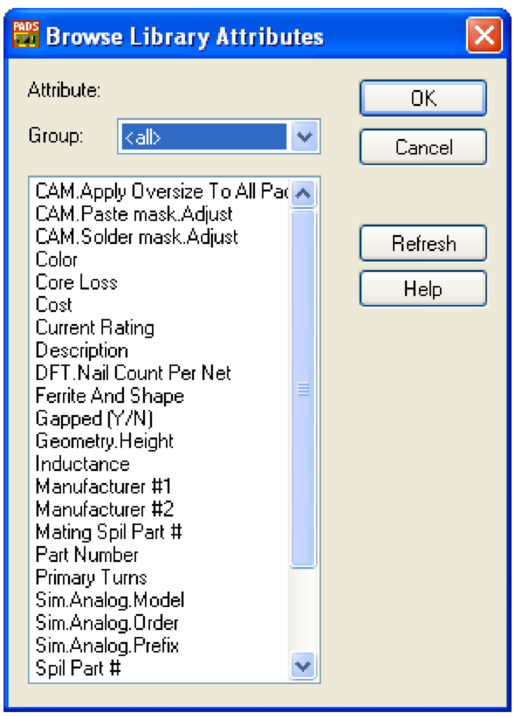


Table 45-56. Browse Library Attributes Dialog Box Contents

Name	Description
Attribute	Displays the selected attribute.
Group	Filters the attribute list. (Includes structured attributes.)
Refresh	Manually updates the attribute list if you change the list of libraries in the Library Manager.

Related Topics

Renaming Attributes of Library Items

Build Clusters Setup Dialog Box

Use the Build Clusters Setup dialog box to create new clusters from parts and unions. Creation is based on connectivity.

Accessing

- **Tools menu** > **Cluster Placement** > **Build Clusters** button > **Setup** button

Figure 45-60. Build Clusters Setup Dialog Box

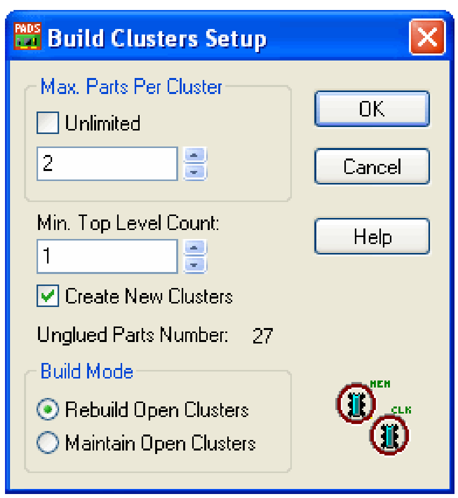


Table 45-57. Build Clusters Setup Dialog Box Contents

Name	Description
Unlimited	Specifies unrestricted parts per cluster.

Table 45-57. Build Clusters Setup Dialog Box Contents (cont.)

Name	Description
Max. Parts Per Cluster box	Sets the maximum number of parts allowed in one cluster.
Minimum Top Level Count	Sets the minimum number of top-level clusters allowed. A top-level cluster is a cluster not contained within another cluster. If you set this to a low number and set Maximum Parts Per Cluster to Unlimited, all parts are grouped into one large cluster.
Create New Clusters	Allows you to create new clusters. Clear the check box to modify only previously created clusters.
Unglued Parts Number box	Displays current number of unglued parts in the design. The box remains unavailable for editing.
Build Mode area	Clusters are identified as either open or closed: **Rebuild Open Clusters**—You can delete or replace an open cluster during automatic cluster creation. Automatically created clusters default to open. Maintain Open Clusters—You cannot delete or replace a closed cluster during automatic cluster creation. Manually created clusters default to closed.

Related Topics

Using the Cluster Placement Dialog Box

CAM Plus Dialog Box

The CAM Plus command generates computer-aided manufacturing (CAM) output files that are compatible with a variety of automatic assembly and pick-and-place machines. Before you use CAM Plus prepare an information file called part.def.

Accessing

- **File Menu > CAM Plus**

Figure 45-61. CAM Plus Dialog Box

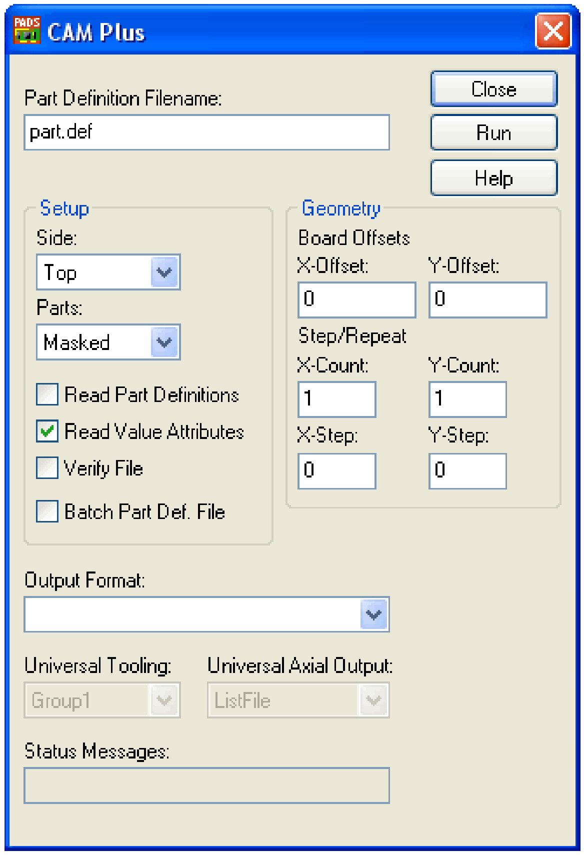


Table 45-58. CAM Plus Dialog Box contents

Name	Description
Part Definition Filename	The name of the Part Definition file. The file, **part.def**, is read from the \Libraries folder by default.
Side	The side of the board on which you want to report: Top or Bottom.
Parts	The type of parts to include in the report: SMT, ThruPin, All, and Masked. **Tip:** Masked parts are those that are assigned to the machine selected format as an insert class.

Table 45-58. CAM Plus Dialog Box contents (cont.)

Name	Description
Read Part Definition	Specifies to add the additional information contained in the Part Definition File, part.def, to the parts contained in a design. This information defines the insertion class for all parts. Read Part Definition scans the Part Definition File for information about the parts in the database. When an exact match is found between a part type name in the database and a part type name in the definition file, the information combines to provide the manufacturing output.
Read Value Definition	Specifies to read the Value attributes for each part in the PADS Layout design and append the Value attribute to the part type name when matching each part type in the Part Definitions file, part.def. **Example:** An R1/4W part type with Value attribute 100K could have an entry in the Part Definitions file as follows: R1/4W{100K},ins=un6241,bodydiam=200,leaddiam=30,anvil=2
Verify File	Specifies to produce an ASCII verification file. This ASCII file is stored in the \PADS Projects\Cam\<board_file_name> folder. This file contains 2D line data that describes the path for the inserted parts. The name of the file is the name of the interface program created, with the .asc extension. You can read this file into PADS Layout with the ASCII In command. It states the insertion path as 2D lines on layer 19. If a Part_Num value for a part is not found, Part_Num is set to Missing.
Batch Part Def. File	Specifies to run all of the outputs with a single command. For each program you run, an output file is produced with the name suffix bt or bb, for example dym318bt.smt or un6241bb.put. If this file already exists, the message “Overwrite existing file (Y/N)?” appears. Click either Yes or No. If you click Yes, the file is overwritten and placed in the CAM subfolder. Each of the parts in the selected category is added to the file. A report called insert.lst lists each part and the machine that inserts it. The Batch command does not create a Verify file.
X-/Y-Offset	Defines the offset of the machine’s location dowel with regard to the 0,0 system origin board. These offset values convert the design coordinates to the machine origin. Allowable values are from 0 to 10 inches. Offset values are in inches; for example, 1250 is 1.25 inches. You may need to define a new Board offset for each machine.

Table 45-58. CAM Plus Dialog Box contents (cont.)

Name	Description
X-/Y-Count and X-/Y-Step	define whether to treat the board as a single design when creating the output program file, or to insert a number of boards simultaneously. When you insert a number of boards, you can define the number of steps in the X and Y direction and the step and repeat interval to use, as shown below. CAM Plus uses current design units for the offset and step and repeat values. Each machine has its own units type to which the data is always converted regardless of the current design units. CAM Plus generates assembly program files for inserting parts on all boards. • **X-Count—**Number of copies in X direction. The maximum is 20. • **Y-Count—**Number of copies in Y direction. The maximum is 20. • **X-Step—**The step distance in the X direction between the origin of each board. The maximum is 10 inches. • **Y-Step—**The step distance in the Y direction between the origin of each board. The maximum is 10 inches. The default is no step and repeat, equivalent to a step of 1 in X and Y.
Output Format	Specifies the machine format. Files are produced for all parts of a selected class: masked, through hole, SMT, top, bottom, and so on. All parts in the class are included in this output.
Universal Tooling and Universal Axial Output	Specifies the desired settings, if applicable. **Restriction:** Universal-specific instructions are available when you select Universal machine formats or check Batch Part Def. File.
Status Messages	Displays the current state of output. Populated after you click Run.
Run	Generates the output file.

Related Topics

Using the CAM Plus Assembly Machine Interface

CAM Preview Setup Dialog Box

Use the CAM Preview Setup dialog box to invert, show plot orientation, or overlay multiple CAM Documents - you can change the preview attributes of all documents.

Tip: You must have at least one CAM document defined to use preview setup.

Accessing

- **File** menu > **CAM** > Select a document > **Preview** button > **Setup** button

Figure 45-62. CAM Preview Setup Dialog Box

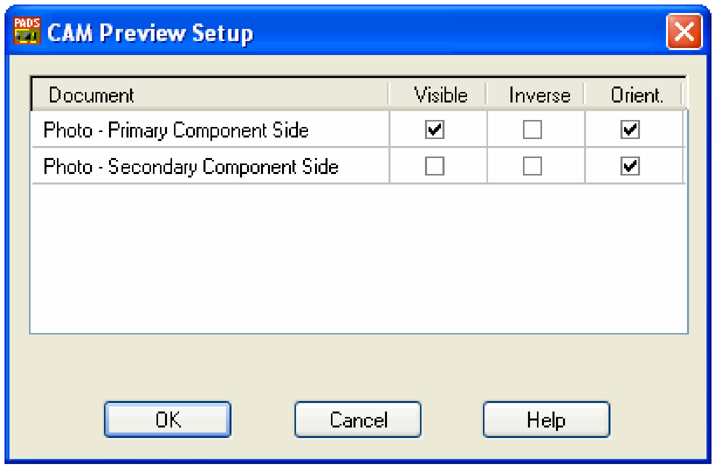


Table 45-59. CAM Preview Setup Dialog Box contents

Name	Description
Document	Displays the document name.
Visible	Specifies the documents you want to see; you can overlay multiple CAM documents.
Inverse	Specifies to invert the view of a CAM document. You might want to invert the view of CAM Plane layers since they are negative layers and will appear different than all other layers.
Orient	Specifies to scale and orient the preview data as defined by the CAM document plot options.

Related Topics

Setting CAM Preview Options

CAM350 Link Dialog Box

Use the CAM350 dialog box to set options for, and to generate, a .cam output file.

Accessing

- **File menu > Export > select CAM350 Files > Save**

Figure 45-63. CAM350 Link Dialog Box

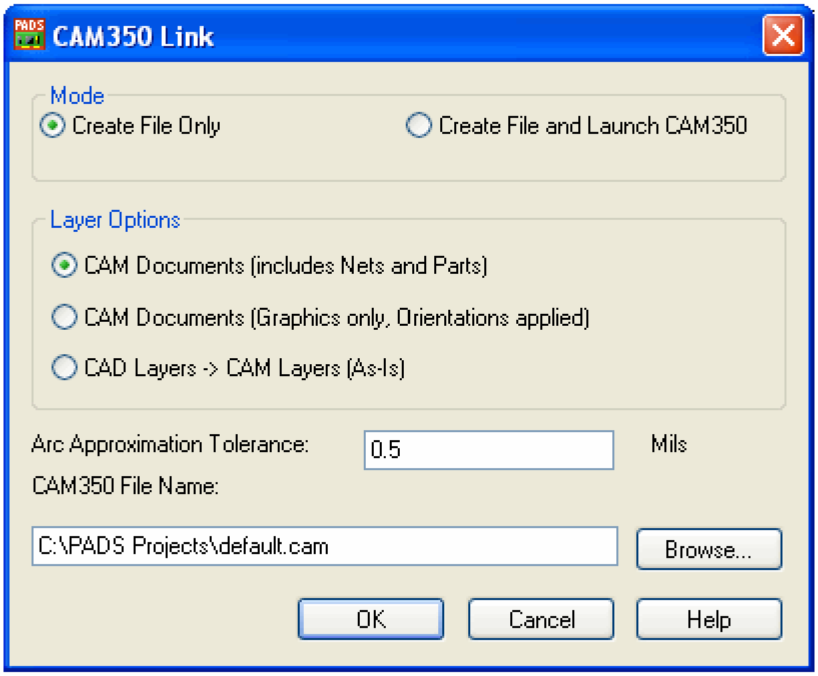


Table 45-60. CAM350 Link Dialog Box contents

Name	Description
Mode area	Specifies the CAM350 mode you want: • Create File Only to simply produce the .cam file • Create File and Launch CAM350 to start CAM 350 and load the resulting .cam file.
Layer Options area	Specifies the amount of PADS design detail to translate to the CAM350 database. • **CAM Documents (includes Nets and Parts)**—Translates CAM documents with part and net intelligence. Plot orientation, as specified in the CAM document, is not applied. This is useful for verifying net lists and DRCs in CAM350. **Requirement:** Before using this option, flood all copper pours and connect all split/mixed plane layers. Also, on the Tools > Options > Split/Mixed Plane tab, in the Save to PCB File area, click All Plane Data. This allows all pour data to be passed to CAM350.

Table 45-60. CAM350 Link Dialog Box contents (cont.)

Name	Description
Layer Options area (con't)	• **CAM Documents (Graphics only, Orientations applied)**—Translates CAM documents with plot orientation, including offset, rotation, mirroring, and scaling. This is useful for direct Gerber file translation to CAM350 where no component or net information is required. **Requirement:** Before using this option, if your PADS Layout photoplot output format is set to RS-274D, make sure your CAM350 default units (mils or inches) and CAM350 default precision are set to match the units for PADS Layout and the precision for PADS Layout File/CAM photoplot output, respectively. CAM350 Link uses the files pads3.arl (for English units) and pads3m.arl (for Metric units) with this option and when Gerber files are in RS274-D format. The *.arl files determine how the aperture report file (*.rep) is generated by CAM in PADS Layout will be interpreted by CAM350. • **CAD Layers -> CAM Layers (As-Is)**—Translates no CAM documents. Translates CAD layers as defined in the PADS Layout database. This is useful for legacy CAM processes that take advantage of CAD layer translation using the PADS-ASCII format import operation in CAM350. **Requirement:** Before using this option, flood all copper pours and connect all split/mixed plane layers. Also, on the Tools > Options > Split/Mixed Plane tab, in the Save to PCB File area, click All Plane Data. This allows all pour data to be passed to CAM350.
Arc Approximation Tolerance	Specifies the minimum allowable distance between the actual arc path and the approximated straight-line segments. **Tip:** This is shown in the design units you set on the Global tab.
CAM350 File Name	Specifies the path and name for the CAM350 file. The default file name and path are the same name as the current design file name and path. Tip: Click Browse to navigate to a location.

Related Topics

Exporting to CAM350

Annotating DFF Errors

CAM Preview Dialog Box

Use the CAM Preview dialog box to preview your CAM Documents.

Accessing

- **File** menu > **CAM** > **Preview** button

  or

- **File** menu > **CAM** > **Add** > **Preview Selections**

  or

- **File** menu > **CAM** > **Add** > **Layers** > **Preview** button

Figure 45-64. CAM Preview Dialog Box

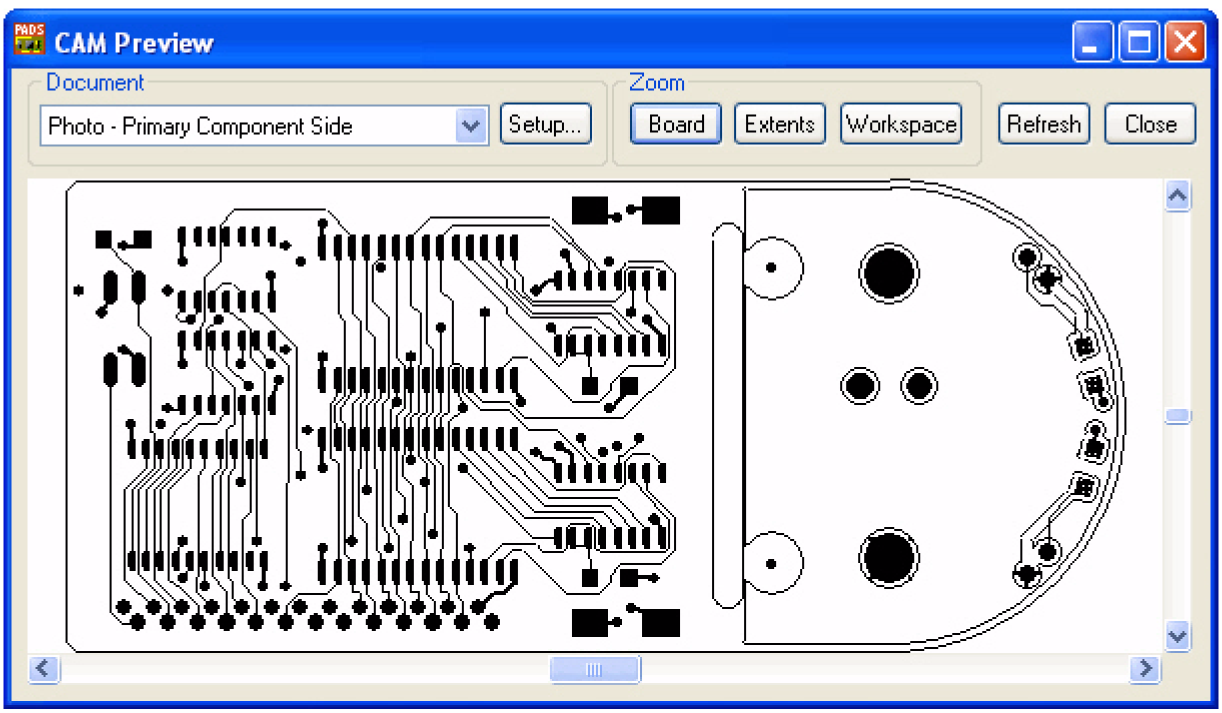


Table 45-61. CAM Preview Dialog Box Contents

Name	Description
Document list	Lists the available documents to preview.
Setup	Opens the CAM Preview Setup dialog box.

Table 45-61. CAM Preview Dialog Box Contents

Name	Description
Board	Zooms the view to the board outline.
Extents	Zooms the view to include the extents of the design space.
Workspace	Zooms the view to the workspace.
Refresh	Refreshes the view from changes in the design.

Related Topics

Adding or Editing CAM Documents

CBP Properties Dialog Box

The CBP Properties dialog box displays the pad number, function, position, and dimensions of the selected component bond pad. In the Wire Bond Editor, with one or more CBPs selected, right-click and click Properties to edit the CBP Properties.

Restriction: This information applies to only the BGA toolkit.

Accessing

- **BGA Toolbar** button > **Wire Bond Editor** button > Select a CBP > right-click > **Properties**

Figure 45-65. CBP Properties Dialog Box

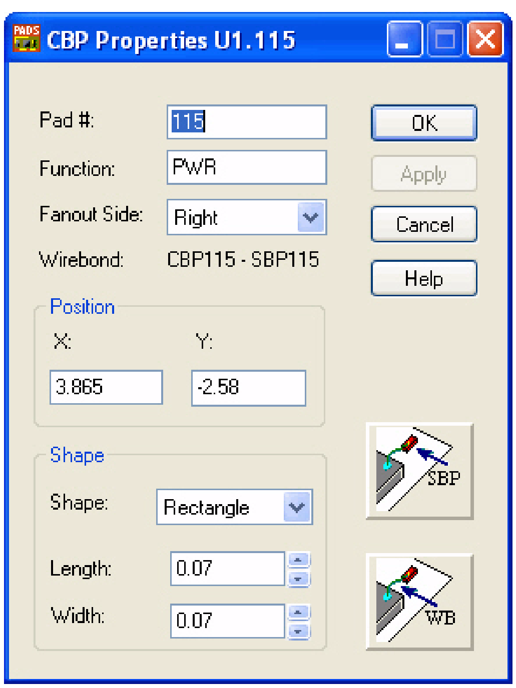


Table 45-62. CBP Properties Dialog Box contents

Name	Description
Pad #	Assigns a number to the currently selected component bond pad. By default, it is the same number as the substrate bond pad to which it is connected.
Function	Defines the function of the currently selected bond pad.
Fanout Side list	Selects the side of the SBP Guide to which the CBP should be wire bonded. The possible values are left, top, right, and bottom.
Wirebond	Displays the name of the substrate bond pad and the component bond pad that are connected by the wire bond.
X and Y	Displays the X and Y coordinates of the bond pad. Type new values to move the bond pad.
Shape list	Assigns a shape to the currently selected bond pad: **Rectangle** or **Oval**.
Length	Assigns a physical length, in current design units, for the currently selected bond pad.
Width	Assigns a physical width, in current design units, for the currently selected bond pad.
SBP button	Opens the SBP Properties dialog box for the component bond pad connected to the currently selected substrate bond pad. This button is unavailable if there is no connected component bond pad.
WB button	Click WB to open the Wire Bond Properties dialog box for the wire bond connected to the currently selected pad. This button is unavailable if there is no connected wire bond.

Related Topics

To Edit Component Bond Pads

CCE Export Dialog Box

Using the CCE Export dialog box, you can export design elements to CAMCAD, visECAD, and visEDOC.

Accessing

- **File** menu > **Export** > select **CCE Files** > **Save**

Figure 45-66. CCE Export Dialog Box

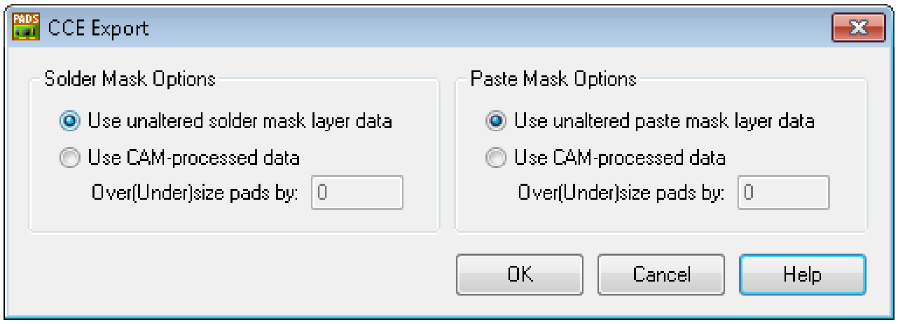


Table 45-63. CCE Export Dialog Box Contents

Name	Description
Solder Mask Options	
Use unaltered solder mask layer data	Select this option to use the solder mask data as it exists in the design. No additional pads are added and no oversizing or undersizing is applied.
Use CAM-processed data	Select this option to accept automatically-generated pads copied from the associated top or bottom layer, and oversizing or undersizing. • You are not required to create solder mask shapes in each pad stack. During export, pads are copied from the outside layer to the solder mask layer for use as the solder mask shape. • Oversizing or undersizing follows a hierarchical priority. For more information, see Control of Solder Mask and Paste Mask.
Over(Under)size pads by	Type a value to apply a global oversize or undersize to all solder mask pads. **Tip:** This global setting is the lowest of a hierarchy of possible values. For more information, see Control of Solder Mask and Paste Mask.
Paste Mask Options	
Use unaltered paste mask layer data	Select this option to use the paste mask data as it exists in the design. No additional pads are added and no oversizing or undersizing is applied.

Table 45-63. CCE Export Dialog Box Contents

Name	Description
Use CAM-processed data	Select this option to accept automatically-generated pads copied from the associated top or bottom layer, and oversizing or undersizing. • You are not required to create paste mask shapes in each pad stack. During export, pads are copied from the outside layer to the paste mask layer for use as the paste mask shape. • Oversizing or undersizing follows a hierarchical priority. For more information, see Control of Solder Mask and Paste Mask.
Over(Under)size pads by	Type a value to apply a global oversize or undersize to all paste mask pads. **Tip:** This global setting is the lowest of a hierarchy of possible values. For more information, see Control of Solder Mask and Paste Mask.
CCE	The CCE format is a compressed version of the CC file. If you want a smaller file size, use the CCE format.

Related Topics

Exporting CCE Files

Change Component Dialog Box

Use the Change Component dialog box to authorize changing a part type in the design and to control the retention of the attributes of the part in the design.

Accessing

- select a component > **ECO Toolbar** > **Change Component** > right-click > **Library Browse** > select a part type > **Replace**

Figure 45-67. Change Component Dialog Box

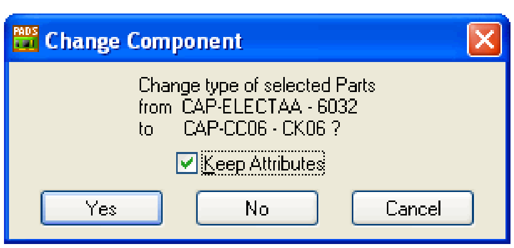


Table 45-64. Change Component Dialog Box Contents

Name	Description
Keep Attributes	Select the check box to keep the attributes of the part in the design and apply them to the replacement part.
Yes	Proceeds with the change of the part(s) in the design.
No	Cancels the Replace command and returns you to the Get Part Type from Library dialog box.
Cancel	Cancels the Change Component

Related Topics

Changing a Component in ECO Mode

Updating a Part Type from the Library in ECO Mode

Check for Updates Dialog Box

Use the Check for Updates dialog box to manually check for a new version of PADS, and to disable or enable automatic checks.

Accessing

- **Help** menu > **Check for Updates**

Figure 45-68. Check for Updates Dialog Box

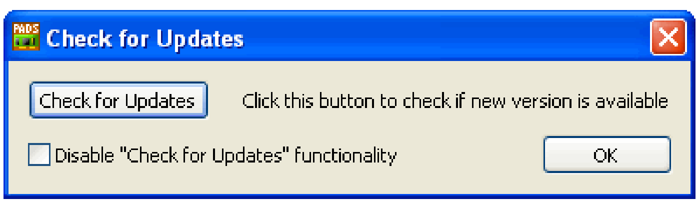


Table 45-65. Check for Updates Dialog Box contents

Name	Description
Check for Updates button	Manually checks for a new version of the PADS software.

Table 45-65. Check for Updates Dialog Box contents (cont.)

Name	Description
Disable “Check for Updates” functionality	Determines if PADS automatically checks for a new version of the software. Click to stop PADS from automatically checking for a new version of the PADS products; click to clear to have PADS automatically check for a new version.

Related Topics

Checking for PADS Updates

Check Teardrop Dialog Box

Use the Check Teardrops dialog box to check for and view teardrop errors. The error location, layer, and a short explanation of the error are reported.

Requirement: You must enable teardrops before you can check them. **Tools** menu > **Options** > **Routing** tab > select **Generate Teardrops** check box > click **OK**.

Accessing

- **Tools** menu > **Options** > **Teardrops** tab **>** **Check button**

Figure 45-69. Check Teardrops Dialog Box

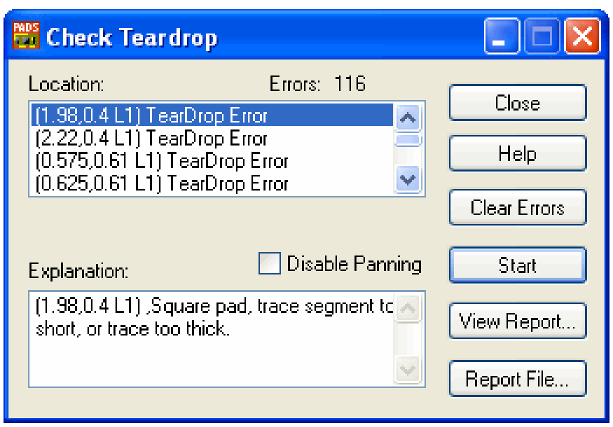


Table 45-66. Check Teardrops Dialog Box contents

Name	Description
Location list	The location of the teardrop error. **Tip:** Click an error to pan to it in the design.

Table 45-66. Check Teardrops Dialog Box contents (cont.)

Name	Description
Errors	Lists the total number of teardrop errors in the design
Explanation list	The reason for the error selected in the Location list.
Disable Panning	Specifies to not pan to the error selected in the Location list.
Clear Errors	Clears the error markers in the design.
Start	Runs the teardrop check.
View Report	Lists the most recently run report results in your default text editor.
Report File	Opens the Save As dialog box where you can specify the name of the error report and where to save it.

Related Topics

Modifying Net Properties

Class Rules Dialog Box

Use the Class Rules dialog box to create and manage classes of nets, and to define design rules that apply to them.

Accessing

- **Setup** menu > **Design Rules** > **Class** button

Figure 45-70. Class Rules Dialog Box

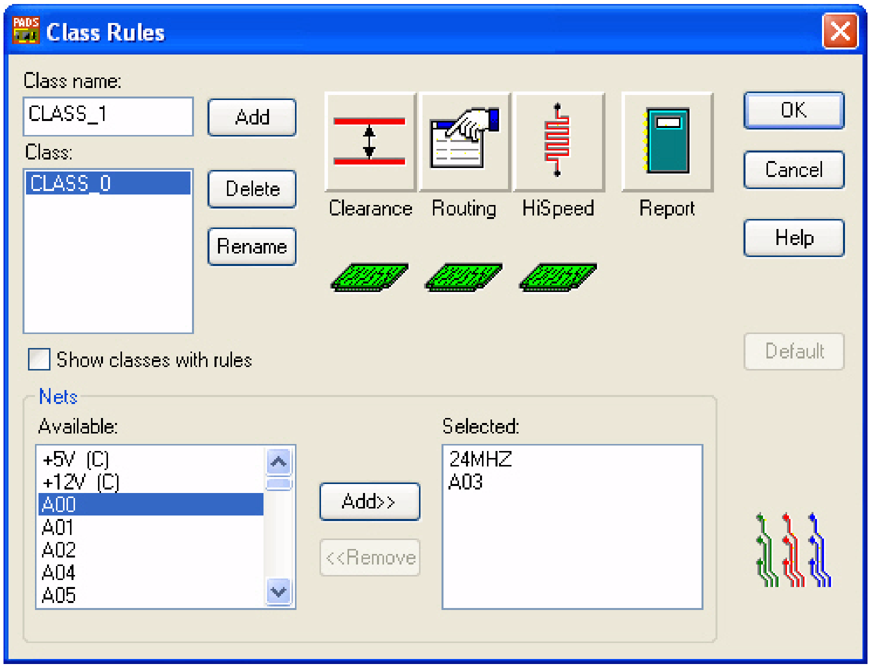


Table 45-67. Class Rules Dialog Box

Name	Description
Class name	Specifies the name of the class.
Class list	Lists all class names.
Add	Adds the class name to the Class list.
Delete	Removes the selected class from the Class list.
Rename	Renames the class selected in the Class list with the text in the Class Name box.
Show Classes with rules	Specifies to show only classes that have rules.
Clearance	Opens the Clearance Rules Dialog Box.
Routing	Opens the Routing Rules Dialog Box.
HiSpeed	Opens the HiSpeed Rules Dialog Box.
Report	Opens the Rules Report Dialog Box.
Nets Available list	Lists the nets available for this class. **Tip:** Nets cannot exist in more than one class. The Available list displays only nets that have not been assigned to a class.
Nets Selected list	Lists the nets selected for this class.

Table 45-67. Class Rules Dialog Box (cont.)

Name	Description
Add >>	Moves the net from the Available list to the Selected list.
<< Remove	Moves the net from the Selected list to the Available list.
Picture below rule button	The picture below each type of rule button identifies which rules hierarchy level is used for that rule type. The meaning of the picture corresponds to the button in the Hierarchy area of the Rules dialog box. For example, if you select a class in the Class list and a green polygon appears below the Clearance button, then the default values apply to the class. **See also**: Non-Default Rules Indicators
Default	Removes non-default rules from the selected classes, so that only default rules apply.

Related Topics

Creating Class Design Rules

Deleting a Design Rule Class

Adding Nets to an Existing Design Rule Class

Removing Nets from a Design Rule Class

Modifying Class Design Rules

Renaming a Design Rule Class

Resetting Class Rules to Default Rules

Displaying the Nets of a Class Design Rule

Design Rule Hierarchy

Clearance Checking Setup Dialog Box

Use the Clearance Checking Setup dialog box to specify which clearances to check during a Clearance verification.

Accessing

- **Tools** menu > **Verify Design** > **Clearance** check > **Setup button**

Figure 45-71. Clearance Check Setup Dialog Box

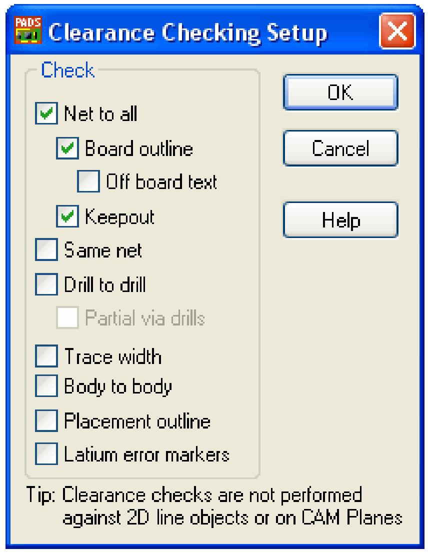


Table 45-68. Clearance Check Setup Dialog Box contents

Name	Description
Net to All	Checks clearance rules from all levels of the Rules Hierarchy (all nets against all foreign obstacles).
Board Outline	Checks clearance rules of net objects against the board outline and board cut outs (pins, vias, traces, and copper). When clearance checking against the board outline, the edge of the object is checked against the centerline of the board outline. **Requirement:** You must also select the Body to body check box to check component outlines against the Board outline.
Off Board Text	Checks for off board text and flags all instances of off board text as clearance errors.
Keepout	Checks for keepout restriction violations.

Table 45-68. Clearance Check Setup Dialog Box contents (cont.)

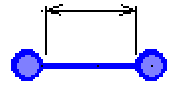
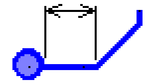
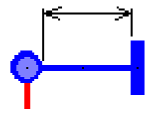
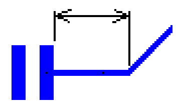
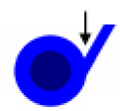
Name	Description
Same Net	Checks clearances between objects along the same net. Object to object checking includes the checks listed below: • Pad edge to pad edge—  • Pad edge to inside corner of trace—  • SMD edge to pad edge—  • SMD edge to inside corner of trace—  • Acute angle between pad and trace— 
Drill to Drill	Checks clearances between all drill holes. Pad stack drill size plus the drill oversize value calculate the diameter for plated holes. **Restriction:** Drill to Drill errors are reported for only one layer in a drill pair.
Partial Via drills	Checks for via configurations that can cause drilling problems during fabrication, specifically, configurations where two partial vias: • Are at the same location, • Have different drill sizes, and • Share a layer (for example, VIA1-2, VIA2-4) **Restriction:** This option is available only when **Drill to Drill** is selected.
Trace Width	Checks traces in excess of minimum and maximum widths.

Table 45-68. Clearance Check Setup Dialog Box contents (cont.)

Name	Description
Body to Body	Checks actual part outline against actual part outline. **Tip:** When used in combination with the Board outline check, you can check component outlines against the board outline. The part outline is the furthest extent of any decal object on the following layers. • component layer (top or bottom) • associated silkscreen layer for component layer (silkscreen top or bottom) • Layer 0 (zero) Typically, the furthest object is either the silkscreen or the pads or a combination of both. If you have a larger placement outline on Layer 20, see the Placement Outline check. **Restriction:** This option is not available for Latium design checking.
Placement Outline	In default layer mode, Placement Outline checks outline against outline on layer 20, not on electrical layers. **Tip:** Changing from default layer mode to increased layer mode increases all nonelectrical layer numbers by 100, so in increased layer mode, Placement Outline checks outline against outline on layer 120. You can create outlines on layer 20 (or layer 120) that do not exactly match the actual silkscreen component outline. By setting a larger outline on this layer you can leave an area near a component open for other purposes. Using the Placement Outline check, you ensure that this area is left open. **See also:** To Nudge Overlapping Parts
Latium Error Markers	Shows Latium error markers. A Latium error marker indicates that you have Latium rules in your design. You can only check these rules by using the Latium Design Verification option in the Verify the Design dialog box.

Related Topics

Setting Up Clearance Checking

Clearance Rules Dialog Box

Use the Clearance Rules dialog box at any level of the rules hierarchy to define minimum edge-to-edge spacing between objects and to define trace widths. Online DRC and design verification check and report clearance violations. Object types are checked against other object types as well as against themselves, such as traces. Values are in current design units.

Accessing

- **Setup** menu > **Design Rules** > choose a hierarchy level > **Clearance** button

Figure 45-72. Clearance Rules Dialog Box

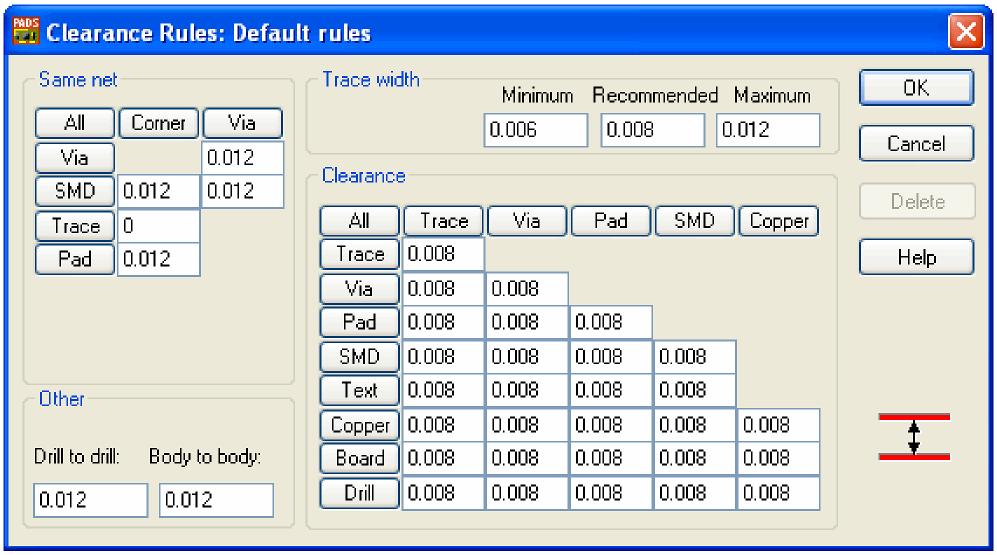


Table 45-69. Clearance Rules Dialog Box

Area	Description
Same net	Specifies the space that must be maintained between items that are of the same net. • **SMD**—surface mount pad **Tip:** The SMD-to-via value is also used for SMD-to-SMD and SMD-to-Pad rules • **Corner**—first trace bend point • **Pad**—through hole pad • **Trace-to-corner**—trace and the bend point of another trace. For example, a trace splits at a T-junction and one of the two traces has a bend point. **Restriction:** Trace-to-Corner clearance is only used by PADS Router. It is not used in Layout (either by Online DRC or checked by the Verify Design tool). **Tip:** To set the same value for an entire row, column or the table, click on a column heading (such as Corner, Via), row heading (such as Via, SMD), or All. Then type a value and click OK to apply the value.

Table 45-69. Clearance Rules Dialog Box (cont.)

Area	Description
Trace width	Specifies the allowed range of trace widths. • **Minimum**—Minimum width for interactive routing • **Recommended**—Width to use when routing begins • **Maximum**—Maximum width for interactive routing **Tip:** Routing respects the minimum and maximum values when it varies trace width to achieve high-speed routing functions, for example, impedance matching.
Clearance	Specifies the space that must be maintained between items of different nets. • **Trace to Trace**—The Gap value set in the Differential Pairs dialog box has precedence over this value. • **SMD**—surface mount pad • **Corner**—first trace bend point • **Pad**—through hole pad • The Copper to Via, Pad, SMD or Drill values are also used to create the default thermals and antipads. **See also:** Design Rule Versus Pad Stack - Thermals and Antipads. **Tips:** • To set the same value for an entire row, column or the whole table/matrix, click on a column heading, row heading, or All. Then type a value and click OK to apply the value. • Chamfered path copper uses the Solid Copper property which switches the copper to use trace clearances.
Other	Specifies the clearances between drill and component body objects.
Delete button	Removes non-default clearance rules at the current level of the rules hierarchy. **Restriction:** You cannot delete the Default clearance rules.
OK button	**Restriction:** This button is unavailable if you are not licensed for the advanced rules but you have opened this dialog box to an advanced level of the rules hierarchy through the Properties of an object. For example, with a pin pair selected, you right-click and click Properties. In the Trace Properties dialog box, you click the Rules button and in the Pin Pair Rules dialog box, you click Clearance. In the resulting Clearance Rules dialog box, the OK button is unavailable.

Tips:

- You can set default clearance rules on a per-layer basis by creating All-to-Layer conditional rules. **See also:** Conditional Rule Setup Dialog Box
- The BGA Route Wizard uses properties defined in this dialog box. **See also:** BGA Route Wizard Dialog Box

Related Topics

Design Rule Hierarchy

Cluster Information Properties Dialog Box

The Cluster Information Properties dialog box modifies cluster attributes for the selected component or union that belongs to a cluster.

Accessing

- Select a union within a cluster > Right-click > **Properties** > **Cluster Info button**

Figure 45-73. Cluster Info Properties Dialog Box

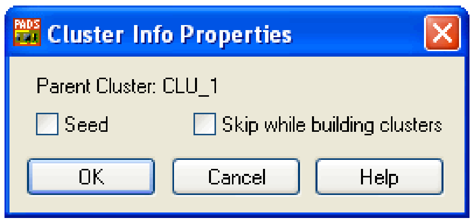


Table 45-70. Cluster Info Properties Dialog Box

Name	Description
Seed	Identifies a cluster as a base, or top level cluster, during build and grow operations. PADS Layout searches outward from seed clusters to identify other clusters that you can add to the seed cluster.
Skip while Building Cluster	Ignores the union or cluster during Grow Incremental and Grow Automatic operations.

Related Topics

Union Properties Dialog Box

Cluster Manager Dialog Box

Use Cluster Manager to display and manage cluster members and unions. You can move cluster members and unions from one cluster to another and break, or delete, clusters. Cluster Manager works similarly to the Microsoft Windows Explorer; with it you can view items at the top level or at any level of the hierarchy.

Accessing

- **Tools** menu > **Cluster Manager**

Figure 45-74. Cluster Manager Dialog Box

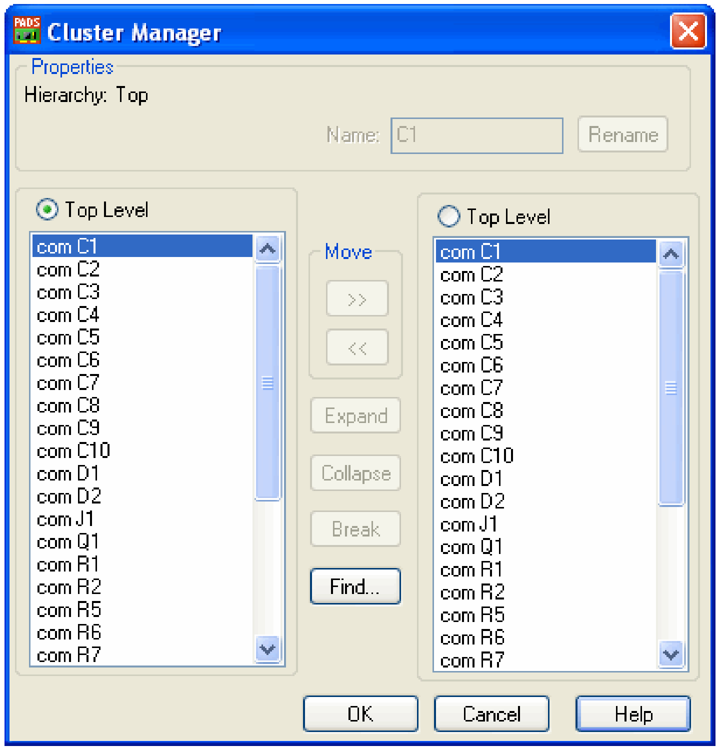


Table 45-71. Cluster Manager Dialog Box Contents

Name	Description
Hierarchy	Keeps track of the current cluster hierarchy. Top is the default level. Expanding a cluster causes the cluster, union, or member name to appear, separated by a forward slash (/).
Name	Displays the name of the highlighted cluster or union. Rename a cluster or union by typing a new name and clicking Rename.
Rename	Renames the selected cluster.
List Boxes	Use either of the two list boxes for viewing all clusters, unions, and components in the design. The Top Level radio button marks the active list. Items within the lists are identified by the prefixes shown: • **Clusters**—CLU • **Unions**—UNI • **Components**—com

Table 45-71. Cluster Manager Dialog Box Contents (cont.)

Name	Description
Move area	Moves members of a cluster from one list to another.
Expand	Changes the contents of the list box to show only the objects in the cluster. The Top Level radio button changes to display the cluster name.
Collapse	Returns to the top-level view, collapsing the hierarchy.
Break	Deletes the selected cluster. You must click OK to remove the cluster from memory.
Find	Opens the Find in hierarchy dialog box where you can find a specific cluster, union, or component name within the listed hierarchy. If a searched name is part of a cluster or union, it expands to show all members.

Related Topics

Using the Cluster Placement Dialog Box

Cluster Placement

Cluster Placement Dialog Box

Use the Cluster Placement dialog box to build new clusters, place clusters within the board outline, and place parts within the board outline.

Accessing

- **Tools menu > Cluster Placement**

Figure 45-75. Cluster Placement Dialog Box

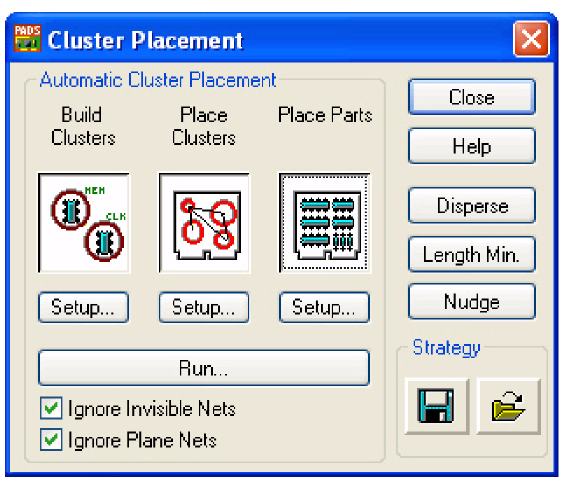


Table 45-72. Cluster Placement Dialog Box Contents

Name	Description
Build Clusters	Automatically creates new clusters outside the board outline. Automatically created clusters default to open. Click the Build Clusters Setup button to customize the clusters. **Setup**—Opens the Build Clusters Setup dialog box. Unavailable till you click the Build Clusters button.
Place Clusters	Automatically positions clusters within the board outline, but does not actually move their associated parts. **Setup**—Opens the Place Clusters Setup dialog box. Unavailable till you click the Place Clusters button.
Place Parts	Performs automatic placement operations. Click the Place Parts Setup button to customize placing clusters. **Setup**—Opens the Place Parts Setup dialog box. Unavailable till you click the Place Parts button.
Run	Click to activate one or more function operations; for example, you can build clusters, place clusters, and place parts in one pass. Starts the operation and displays the Cluster Placement Status dialog box. **Restriction:** This button is unavailable until you select one of the three buttons listed above. You can activate more than one function for multiple operations; you can build clusters, place clusters, and place parts in one pass.
Ignore Invisible Nets	Excludes any invisible nets from the build or placement strategy. To include invisible nets click to clear Ignore Invisible Nets.
Ignore Plane Nets	Excludes plane layer nets in the build or placement strategy. To include plane nets click to clear this option.
Disperse	Places unglued components outside the board outline to clear the board interior. Dispersion is based on part height and is sorted by length. Glued parts are not affected. Clusters are dispersed if you are in Cluster View Mode when you use Disperse.
Length Minimization	Rearranges the pin pairs on the board. The pin pairs are calculated based on the minimum length criteria you set up using Routing Rules.

Table 45-72. Cluster Placement Dialog Box Contents (cont.)

Name	Description
Nudge	Runs a Nudge pass in automatic mode, automatically adjusting parts that fail Clearance Checking. **Tips:** • Nudge considers test points as glued objects. • Nudge ignores components that are part of a physical design reuse.
Save	Saves the current strategy for all three operations.
Load	Opens a saved strategy file.

Related Topics

Using the Cluster Placement Dialog Box

Cluster Placement

Cluster Placement Status Dialog Box

The Cluster Placement Status dialog box displays the percentage of completion for each pass that you enable using Setup. The right side of this dialog box displays estimated and elapsed time for placement passes.

Figure 45-76. Cluster Placement Status Dialog Box

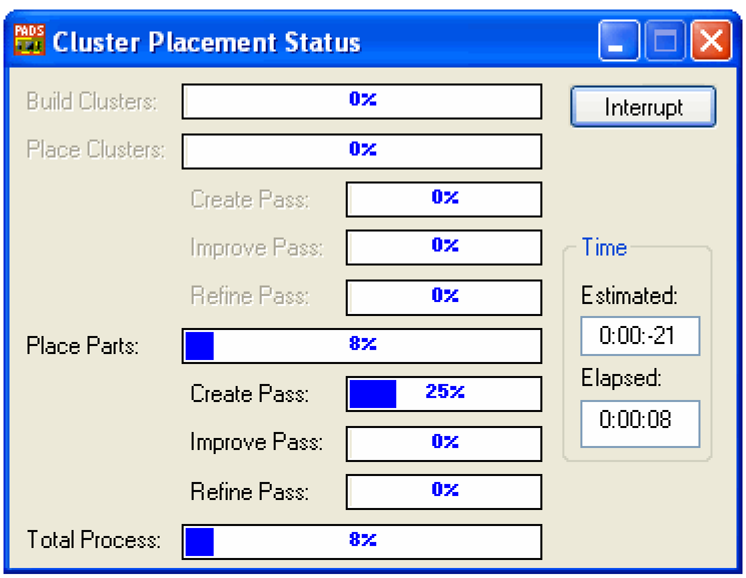


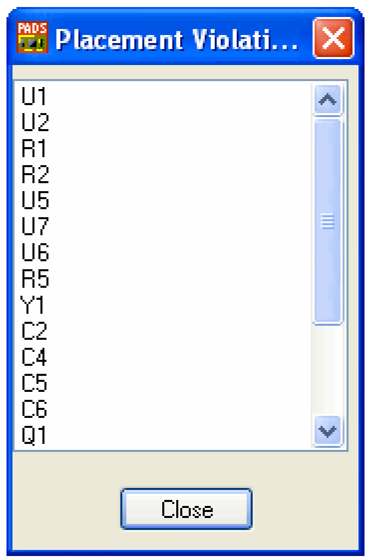
Table 45-73. Cluster Placement Status Dialog Box Contents

Name	Description
Interrupt	Pauses the placement process and opens the Auto Placement Prompt.

After completion of placement operations, errors appear in the Placement Violations dialog box.

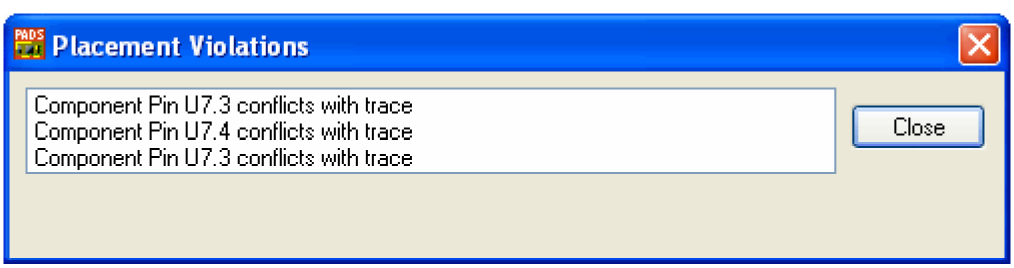
Placement Violations Dialog Box

Figure 45-77. Placement Violations Dialog Box



Double-click an item in the list for information.

Figure 45-78. Placement Violations Info Dialog Box



Related Topics

Using the Cluster Placement Dialog Box

Cluster Placement

Cluster Properties Dialog Box

Use the Cluster Properties dialog box for information on and to modify a selected cluster.

Accessing

- **Select a cluster** > **Right-click** > **Properties**

Figure 45-79. Cluster Properties Dialog Box

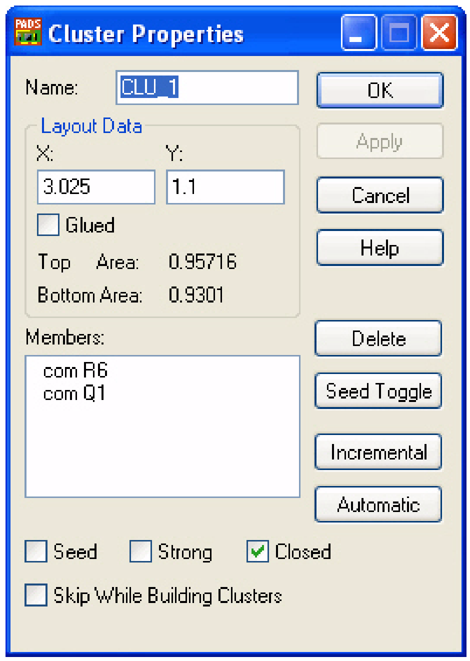


Table 45-74. Cluster Properties Dialog Box Contents

Name	Description
Name	Name of the currently selected cluster. To rename the cluster, type a new name.
X/Y Coordinates	Current coordinates of the cluster. To move the cluster to a new location, type new values.
Glued	Prevents the cluster from moving through manual or automatic placement processes.
Top Area/Bottom Area	The area the cluster encompasses based on the area of each cluster member.
Members	Individual parts that are members of the selected cluster.
Delete	Deletes the cluster and converts its members as individual parts.

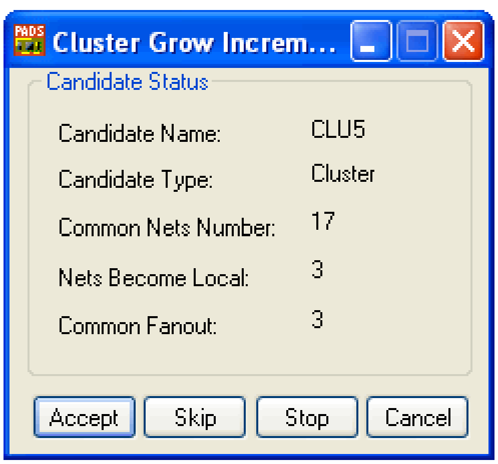
Table 45-74. Cluster Properties Dialog Box Contents (cont.)

Name	Description
Seed Toggle	Selects parts from which you should start building clusters. Analysis for other parts to add to the cluster is based on these parts and on connectivity.
Incremental	Opens the Cluster Grow Incremental dialog box to incrementally add new members to the cluster. **See also:** Modifying Existing Clusters
Automatic	Opens the Cluster Size Limit Definition dialog box to automatically add new members to the cluster. **See also:** Modifying Existing Clusters
Seed	Identifies a cluster as a base, or top level cluster, during build and grow operations. PADS Layout searches outward from seed clusters to identify other clusters that you can add to the seed cluster.
Strong	Places cluster members as close together as possible during placement operations. The minimum distance for placement is the same distance for part clearances in Design Rules.
Closed	Prevents the cluster from being deleted during the Build Clusters pass of Cluster Placement.
Skip while Building Cluster	Ignores the cluster during Grow Incremental and Grow Automatic operations.

Cluster Grow Incremental Dialog Box

Use the Cluster Grow Incremental dialog box to cycle through other clusters in the design and select which ones to absorb into the current cluster.

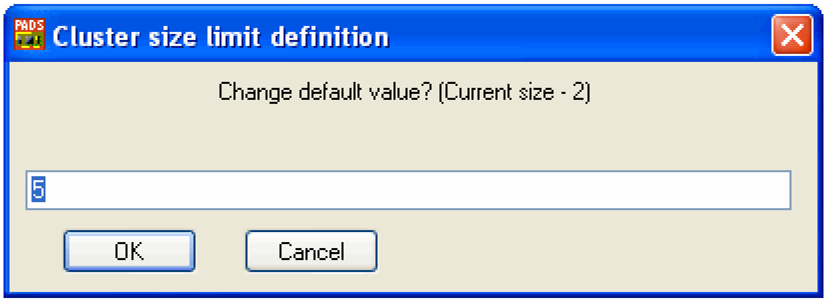
Figure 45-80. Cluster Grow Incremental Dialog Box



Cluster Size Limit Definition Dialog Box

Use the Cluster Size Limit dialog box to automatically add new members to the cluster based on a max number.

Figure 45-81. Cluster Size Limit Definition Dialog Box



Related Topics

Using the Cluster Placement Dialog Box

Cluster Placement

Collaboration Data Import Dialog Box

Use the Collaboration Data Import dialog box to target any imported 2D lines to a specific documentation layer. Collaboration data often includes markups that are physically associated with the issues. The markups are imported as 2D lines to the documentation layer selected in the dialog box.

Accessing

- In the **Collaboration Markups** dialog box, click the **Import** button, browse to and select a file, then click **Open**.

Figure 45-82. Collaboration Data Import Dialog Box

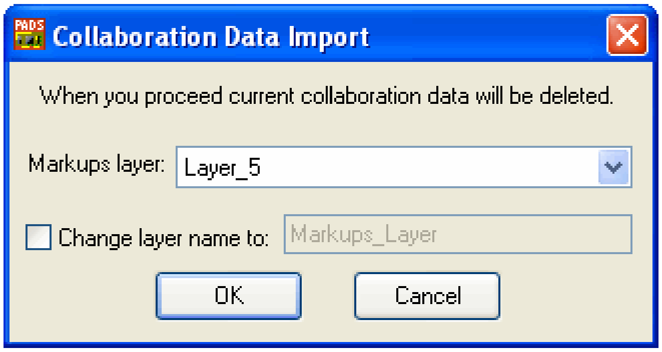


Table 45-75. Collaboration Data Import Dialog Box Contents

Name	Description
Markups Layer	Select a documentation layer from the list, as the location for any 2D line markups that might be included in the collaboration data.
Change layer name to	Select the check box to automatically change the name of the layer as listed in the Layers Setup. You can accept the default layer name or type a new one.

Compare/ECO Tools Dialog Box, Comparison Tab

Use the Comparison tab on the Compare/ECO Tools dialog box to specify the elements to include in the design comparison.

Tip: During design comparison, PADS Layout ignores the reuse definition and uses the actual elements in the physical design reuse.

Accessing

- **Tools Menu > Compare/ECO Tools > Comparison tab**

Figure 45-83. Comparison Tab

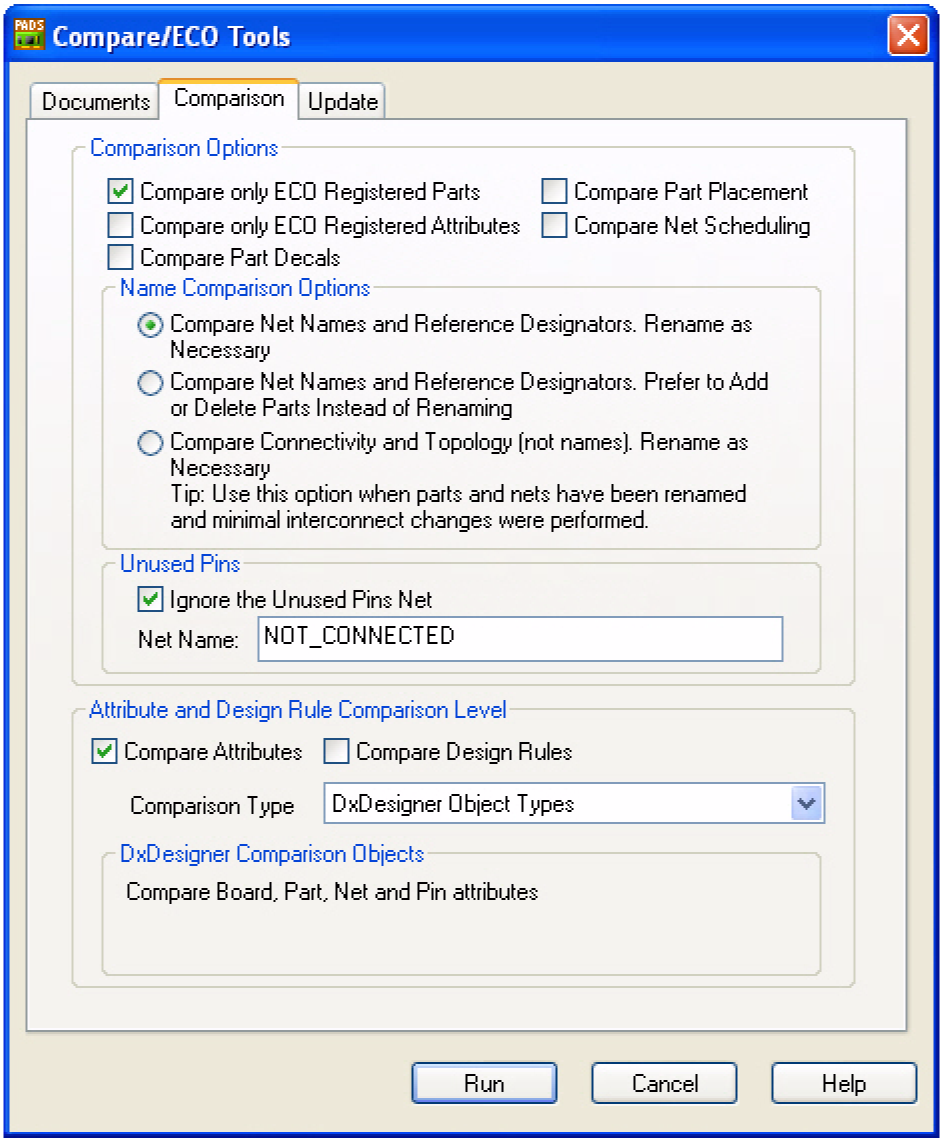


Table 45-76. Comparison Tab contents

Name	Description
Comparison Options area—if you are testing the effect of these options, you should save a copy of your design before you import an .eco file since there is no undo of the .eco import.	

Table 45-76. Comparison Tab contents (cont.)

Name	Description
Compare only ECO Registered Parts	Comparison excludes non-ECO-registered parts. Non-ECO-registered parts may include mechanical or non-electrical parts present in the PCB design and not present in the schematic. To include all parts during comparison, clear Compare Only ECO Registered Parts.
Compare only ECO Registered Attributes	Comparison excludes non-ECO-registered attributes. **Tip:** Via attributes are not ECO registered and cannot be added, deleted, or changed during the ECO process.
Compare Part Decals	Comparison includes part decals.
Compare Part Placement	Comparison includes positional differences in component placement.
Compare Net Scheduling	Comparison includes net scheduling differences. Net rescheduling is performed in PADS Router. For more information, see Rescheduling Nets in the PADS Router Help.
Name Comparison Options area	• **Compare Net Names and Reference Designators. Rename as Necessary**—Compare differences using reference designators and net names. Best used to minimize changes to routed traces. Selecting this option may result in the positional swapping of parts. For example, if routed trace changes are minimized when R1 and R12 are swapped, simultaneously rename R12 to R1 and R1 to R12, and then reconnect R1 and R12 to the original nets. • **Compare Net Names and Reference Designators. Prefer to Add or Delete Parts Instead of Renaming**—Compare differences using reference designators and net names on the basis that few reference designators have been renamed and nets have not been renamed. Best used to minimize the positional swapping of parts, and the design disruption that may result. • **Compare Connectivity and Topology (not names). Rename as Necessary**—Compare differences without using reference designators or net names. Compare differences using pin names, part type names, and so on. Best used to compare designs when parts and nets have been renamed, and minimal interconnect changes have been performed.

Table 45-76. Comparison Tab contents (cont.)

Name	Description
Ignore the Unused Pins Net	Specifies to exclude the unused pins net in the original design. The unused pins net contains pins that have no logical net association. An unused pins net may be created when routing with SPECCTRA or with other tools in the PCB design process. **Tip:** If you clear this option and you update the PCB layout from a schematic or previous PCB layout, the unused pins net may be deleted.\
Net Name	The name of the unused pins net. The maximum netname length is 47 characters. You can use any alphanumeric characters except curly braces { }, asterisks *, spaces, questions marks, or commas.
Attribute and Design Rule Comparison Level area	See the Comparing Design Element Attributes table.
Comparison Objects area	Lists what you've selected in the Attribute and Design Rule Comparison Level area
Run	Compares the designs. **Restriction:** Available when you select an option in the Output Options area on the Documents tab.

Table 45-77. Comparing Design Element Attributes

Select	With	To Compare
Compare Attributes	**PADS Logic Object Types** **Tip:** Select this option to compare PADS Logic netlists that have only parts defined.	Only part attributes (parts and nets) Each part receives attributes from its corresponding Decal and Part Type, but modification is performed only at the part level.
	DxDesigner objects **Tip:** Select this option to compare DxDesigner netlists where board, part, net, and pin attributes are defined.	Board, parts, nets, and pin attributes Each part receives attributes from its corresponding Decal and Part Type. Nets assume attributes from any net class to which they belong. Part and net differences are updated only at the part or net level respectively. There is no assumed hierarchy for pin attributes.
	All Object Types **Tip:** Select this option when you are comparing different versions of a PADS Layout design.	All attributes (board, parts, nets, pins, net classes, part types, part decals). No attribute hierarchy is assumed; all object types are compared and updated at their current level.
Compare Design Rules	**PADS Logic Object Types**	Net and net class rules
	DxDesigner Object Types	• Net and net class rules • General rules • Differential pairs rules
	All Object Types	All design rules
Both **Compare Attributes** and **Compare Design Rules**	**PADS Logic Object Types**	• Part and net attributes • Net and Net Class rules • General and Conditional rules • Differential Pairs rules
	DxDesigner Object Types	• Board, part, net, and pin attributes • Net and Net Class rules • General rules • Differential pairs rules
	All types of objects	• Attributes of all object types • All design rules

Related Topics

Comparing Designs

Working with PADS Logic

Working with DxDesigner

Compare/ECO Tools Dialog Box, Documents Tab

Use the Documents tab on the Compare/ECO Tools dialog box to specify the designs to compare and the files to create.

Accessing

- **Tools Menu** > **Compare/ECO Tools**

Figure 45-84. Documents Tab

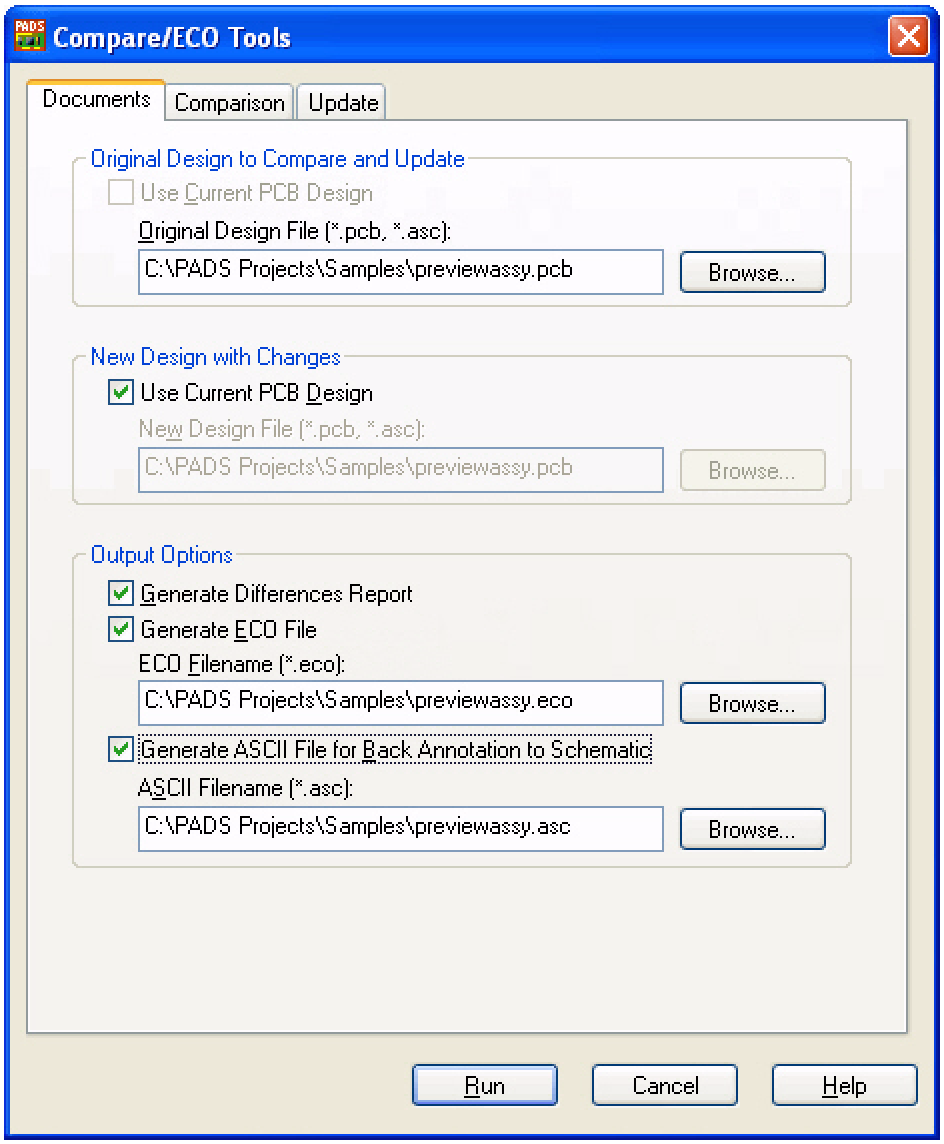


Table 45-78. Documents Tab contents

Name	Description
Original Design to Compare and Update	• **Use Current PCB Design**—Specifies to use the current design as the original to compare and update. • **Original Design File**—Specifies to browse for another design to use as the original to compare and update. **Tip:** Click to clear Use Current PCB Design to browse for another one.
New Design with Changes	• **Use Current PCB Design**—Specifies to use the current design as the new design with changes. • **New Design File**—Specifies to browse for another design to use as the new design with changes. **Tip:** Click to clear Use Current PCB Design to browse for another one.
Generate Differences Report	Specifies to create a report file containing a description of the differences between the two design versions. This file is named Layout.rep and is stored in the \PADS Projects folder.
Generate ECO File	Specifies to create an ECO file. Type or browse to the ECO file. The ECO file contains ECO commands that describe the changes needed to update the original design to match the new design.
Generate ASCII File for Back Annotation to Schematic	Specifies to create a file that contains information to send back to the schematic, click the **Generate ASCII File for Back Annotation to Schematic** box. You must also type the path and name of the file to create in the ASCII Filename box.
Run	Compares the designs. **Restriction:** Available when you select an option in the Output Options area.

Related Topics

Comparing Designs

Working with PADS Logic

Working with DxDesigner

Compare/ECO Tools Dialog Box, Update Tab

Use the Update tab on the Compare/ECO Tools dialog box to specify the types of data to update in the original design and to specify whether to update library data.

Accessing

- **Tools Menu > Compare/ECO Tools > Update tab**

Figure 45-85. Update Tab

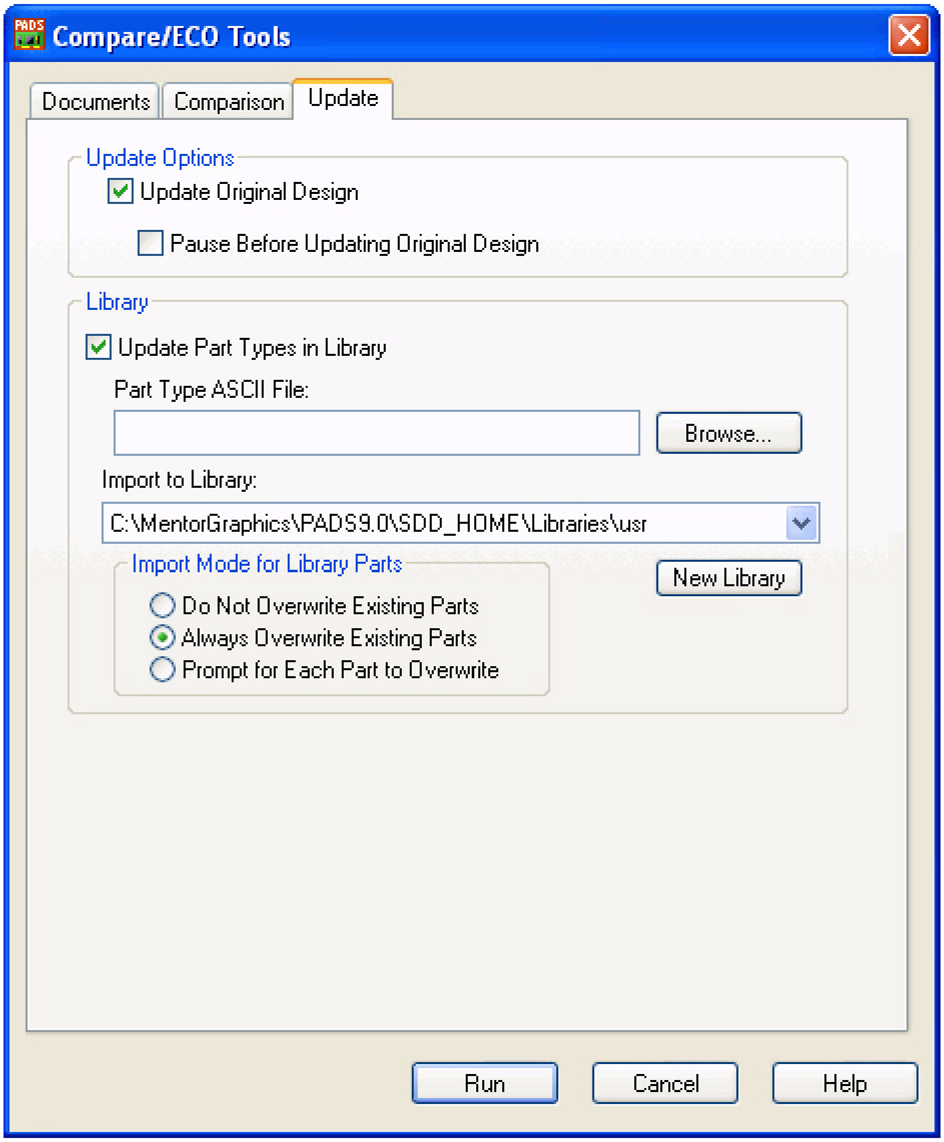


Table 45-79. Update Tab contents

Name	Description
Update Original Design	Specifies to update the PCB layout in memory to match the new design by automatically importing into PADS Layout the ECO file resulting from design comparison. **Tip:** The Update Original Design option is unavailable unless both of the following options are selected: • **Use Current Design** in the Original Design to Compare and Update area on the Documents tab. • **Generate ECO File** on the Documents tab.
Pause Before Updating Original Design	Specifies that you want to view the differences report or ECO file before automatically importing the ECO file into the original PADS Layout design in memory. **Restriction:** Available only when Update Original Design is checked on the Update tab *and* **Use Current PCB Design is checked** in the Original Design to Compare and Update area on the **Documents** tab.
Update Part Types in Library	Specifies to update the PADS Layout part type library by automatically importing the PADS-format ASCII part type file (.p) generated with ViewPCB in DxDesigner.
Party Type ASCII File	Specifies the ASCII file to use. Type or browse for the file name.
Import to Library list	Specifies the Library in which you want to import the updates.
New Library button	Specifies that you want to create a new library in which to import the updates.
Import Mode for Library Parts area	Specifies how you want the changes to be imported: • Do Not Overwrite Existing Parts • Always Overwrite Existing Parts • Prompt for Each Part to Overwrite
Run	Compares the designs. **Restriction:** Available when you select an option in the Output Options area on the Documents tab.

Related Topics

Comparing Designs

Working with PADS Logic

Working with DxDesigner

Component Layer Associations Dialog Box

Use the Component Layer Associations dialog box to associate, or otherwise map, which documentation layers go with the selected layer when a top or bottom layer is set as a component layer. CAM (Negative) planes should only have one net assigned. Split/Mixed may have multiple nets assigned.

The layer associations made in this dialog box are used by CAM routines for output. For example, when you output a silkscreen for the top, any items on the documentation layer you associated for silkscreen are automatically added to the CAM document.

Accessing

- **Setup** menu > **Layer Definition** > **Associations** button

Figure 45-86. Component Layer Associations Dialog Box

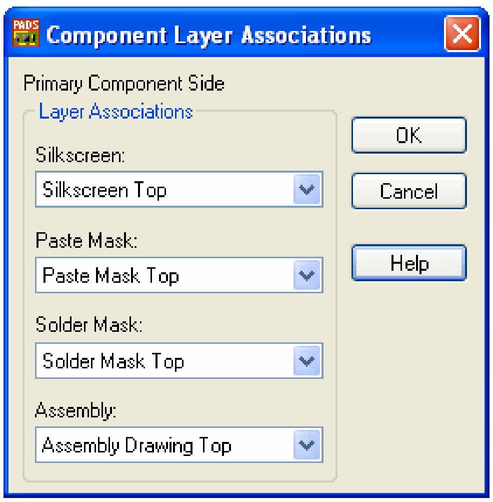


Table 45-80. Component Layer Associations Dialog Box Contents

Name	Description
Name	The name of the layer selected in the Layers Setup dialog box. This is the layer you are associating with the components.
Silkscreen	Specifies which Silkscreen layer you want to associate: • None • Silkscreen Bottom • Silkscreen Top

Table 45-80. Component Layer Associations Dialog Box Contents (cont.)

Name	Description
Paste Mask	Specifies which Paste Mask layer you want to associate: • None • Paste Mask Bottom • Paste Mask Top
Solder Mask	Specifies which Solder Mask layer you want to associate: • None • Solder Mask Bottom • Solder Mask Top
Assembly	Specifies which Assembly layer you want to associate: • None • Assembly Drawing Bottom • Assembly Drawing Top

Related Topics

Setting Up an Outer Layer

Component Properties Dialog Box

Use the Component Properties dialog box to modify component placement, decal, cluster, and label information.

Accessing

- Select a component > Right-click > **Properties**

Figure 45-87. Component Properties Dialog Box

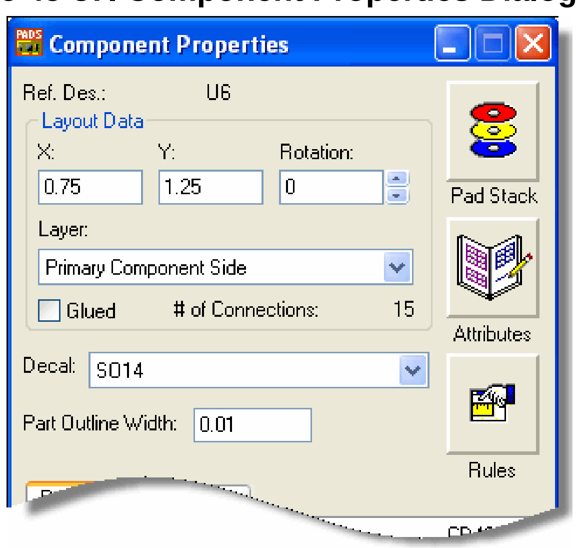


Table 45-81. Component Properties Dialog Box

Name	Description
Ref. Des.	The name of the reference designator of the selected part.
X,Y coordinates	Displays the current coordinates of the queried part. Type new coordinates to move the part.
Rotation list	Displays the current rotation angle of the part. Type a new rotation angle to change the rotation of the part.
Layer list	Displays the layer on which the part is located. Click a new layer from the list to move the component to another layer.
Glued	Sets whether the component is glued. Click to select the Glued check box to glue the component to the board and prevent component movement. Click to clear Glued to unglue a component. If multiple parts are selected, and some of the parts are glued and others are not, the Glued check box appears gray (in an undefined state). You can click to select the Glued check box to glue all selected parts or click to clear the Glued check box to unglue all selected parts.
# of Connections	Lists the number of connections attached to the part.
Decal list	Lists the current part decal. Click a different decal from the list to change the decal. **Caution:** You can assign decals using this dialog box outside of ECO mode. This allows attribute changes even if the attribute is ECO-registered. For example, you can change the decal for U1 from a DIP 14 with a Geometry.Height attribute set at 200, to a SOIC 14 with a Geometry.Height attribute set at 100. Since you are not in ECO mode, the change is not recorded in the .eco file.
Part Outline Width	Displays the current width of component outlines. PADS Layout only changes the part outline widths of selected components' decals, not the widths of all decals in the design. **See also:** To Change Part Outline Width
Pad Stack button	Opens the component in the Pad Stack Properties dialog box.
Attributes button	The Object Attributes dialog box opens to assign attributes and values to any selected objects.
Rules button	Opens the Component Rules Dialog Box with the selected components highlighted in the Components list. **See also:** Design Rule Hierarchy

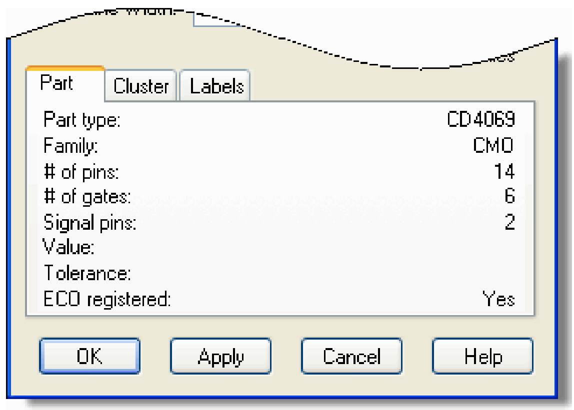
There are three tabs that change the content of the lower half of the dialog box:

Part Tab	Lists all information about the part.
Cluster Tab	Exclude clusters or set one as a base cluster.
Labels Tab	Set Part Label properties.

Part Tab

Use the Part tab in the Component Properties dialog box to view all information about the part. This is not editable.

Figure 45-88. Part Tab



Cluster Tab

Use the Cluster tab in the Component Properties dialog box to exclude clusters or set one as a base cluster.

Figure 45-89. Cluster Tab

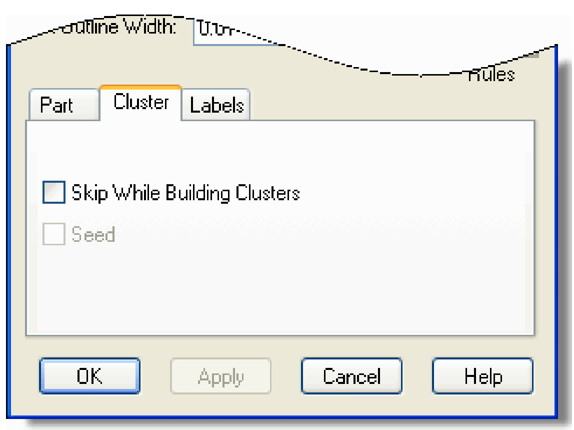


Table 45-82. Cluster Tab Contents

Name	Description
Skip While Building Cluster	Excludes the component from cluster builds. Click to exclude the selected part from cluster build operations.
Seed	Identifies a cluster as a base, or top level cluster, during build and grow operations. PADS Layout searches outward from seed clusters to identify other clusters that you can add to the seed cluster.

Labels Tab

Figure 45-90. Labels Tab

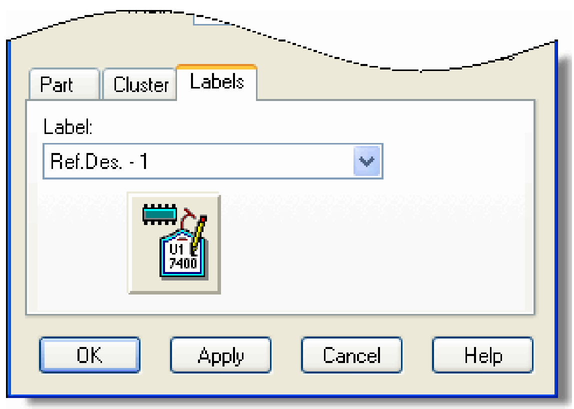


Table 45-83. Labels Tab Contents

Name	Description
Label list	Specifies the label you want to create or edit.
Part Label Properties	Opens the Part Label Properties dialog box.

Related Topics

Modifying Component Properties

Component Properties Dialog Box, Associated Nets

Use the Component Properties dialog box to modify component placement, decal, cluster, label, and associated net information.

Accessing

- Select a component > Right-click > **Properties**

Figure 45-91. Component Properties Dialog Box, Associated Nets,

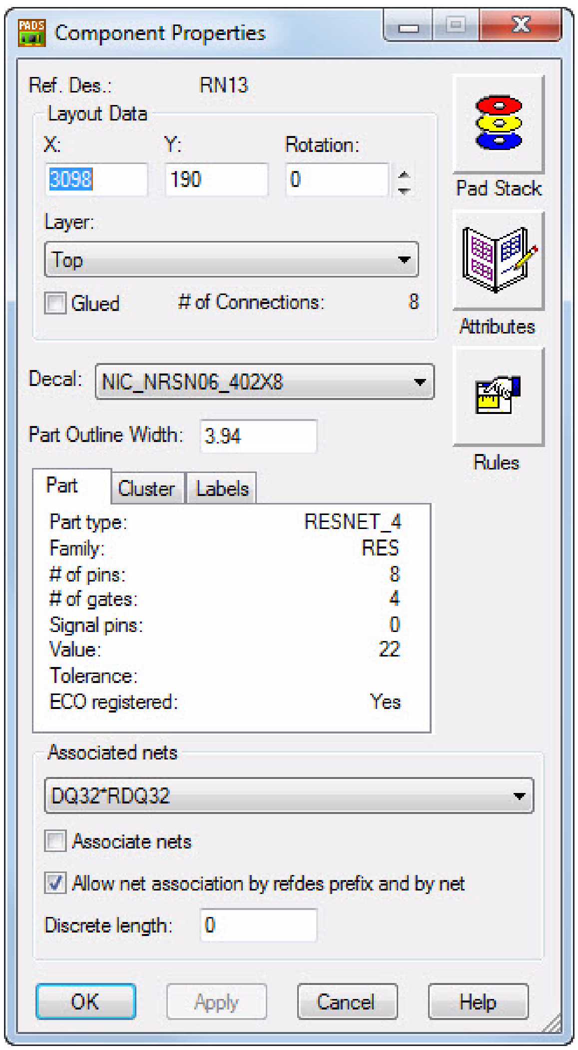


Table 45-84. Component Properties Dialog Box, Associated Nets Contents

Name	Description
Ref. Des.	The name of the reference designator of the selected part.
X,Y coordinates	Displays the current coordinates of the queried part. Type new coordinates to move the part.
Rotation list	Displays the current rotation angle of the part. Type a new rotation angle to change the rotation of the part.
Layer list	Displays the layer on which the part is located. Click a new layer from the list to move the component to another layer.
Glued	Sets whether the component is glued. Click to select the Glued check box to glue the component to the board and prevent component movement. Click to clear Glued to unglue a component. If multiple parts are selected, and some of the parts are glued and others are not, the Glued check box appears gray (in an undefined state). You can click to select the Glued check box to glue all selected parts or click to clear the Glued check box to unglue all selected parts.
# of Connections	Lists the number of connections attached to the part.
Decal list	Lists the current part decal. Click a different decal from the list to change the decal. **Caution:** You can assign decals using this dialog box outside of ECO mode. This allows attribute changes even if the attribute is ECO-registered. For example, you can change the decal for U1 from a DIP 14 with a Geometry.Height attribute set at 200, to a SOIC 14 with a Geometry.Height attribute set at 100. Since you are not in ECO mode, the change is not recorded in the .eco file.
Part Outline Width	Displays the current width of component outlines. PADS Layout only changes the part outline widths of selected components' decals, not the widths of all decals in the design. **See also:** To Change Part Outline Width
Pad Stack button	Opens the component in the Pad Stack Properties dialog box.
Attributes button	The Object Attributes dialog box opens to assign attributes and values to any selected objects.
Rules button	Opens the Component Rules Dialog Box with the selected components highlighted in the Components list. **See also:** Design Rule Hierarchy
Part tab	Lists all information about the part.

Table 45-84. Component Properties Dialog Box, Associated Nets Contents

Name	Description
Clusters tab	Exclude clusters or set one as a base cluster. • **Skip While Building Cluster** Select to exclude the selected part from cluster build operations. • **Seed** Identifies a cluster as a base, or top level cluster, during build and grow operations. PADS Layout searches outward from seed clusters to identify other clusters that you can add to the seed cluster.
Labels tab	Set Part Label properties. • **Label list** Specifies the label you want to create or edit. • **Part label properties button** Opens the Part Label Properties dialog box.
Associated nets list	Name(s) of the associated nets attached to or going through the component. An associated net's name is a sorted list of net names separated by asterisks (*).

Table 45-84. Component Properties Dialog Box, Associated Nets Contents

Name	Description

Associate nets	Select the checkbox to associate the component's nets.
	Tip: This checkbox is also set by the Associate Nets popup command for selected components.
	Restrictions:
	• If a net of a selected component is excluded from net association, it cannot be included in an associated net. (A component or net is excluded from net association if its Associate nets and Allow net association... checkboxes are both cleared.)
	• If a selected component has more than two pins, the following conditions must apply, or the component cannot be an associating component, that is, the associated net can't go through the component:
	  • All pins must connect to a gate.
	  • Each gate must have exactly two pins.
	
	Clear the checkbox to remove the component from net association, that is, to prevent an associated net from going through it.
	Tip: This checkbox is also cleared by the Disable Net Association popup command for selected components.
	Restriction: If the Associate net checkbox in the Net Properties dialog box is checked for the component's nets, you must also clear the Allow association by refdes prefix and by net checkbox.
	
	Clear both checkboxes to prevent the selected components from being an associating componentin any associated net (that is, to prevent any associated net from going through them).

Table 45-84. Component Properties Dialog Box, Associated Nets Contents

Name	Description
Allow net association by refdes prefix and by net	Select the checkbox to allow the component's nets to be included in associated nets automatically by specifying the component's refdes prefix in the Associated Nets dialog box, or by setting the Associate net checkbox in the Net Properties dialog box. Clear the checkbox to prevent association of the selected components' nets *through those components* by net or by refdes prefix. The nets can still be associated by the other components they're attached to. **Tip:** This checkbox is also cleared by the Disable Net Association popup command for selected components. **Restriction:** Components that have the Associate nets checkbox checked are not disabled from association by clearing this check box. Clear both checkboxes to prevent the selected components from being an in any associated net (that is, to prevent any associated net from going through them).
Discrete length	Enter the length to be used for this component when calculating associated net length.

Related Topics

Modifying Component Properties

Component Rules Dialog Box

Use the Component Rules dialog box to define design rules that apply to components.

Restrictions

-
- You can define Component Rules in PADS Layout; however, these rules are used in PADS Router only.

Accessing

- **Setup** menu > **Design Rules** > **Component** button
  or
  Select a component **> right-click > Show Rules**

Figure 45-92. Component Rules Dialog Box

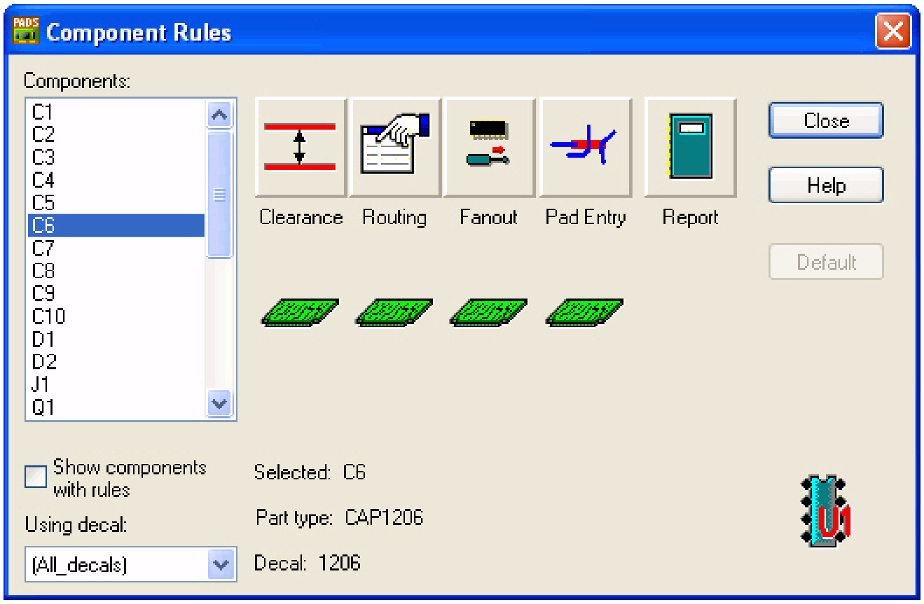


Table 45-85. Component Rules Dialog Box

Name	Description
Components list	Lists all components in the design.
Show components with rules	Specifies to show only components that have rules.
Using Decal	Specifies to display components for a specific decal. **Tip:** Select (All_decals) to display all decals.
Clearance	Opens the Clearance Rules Dialog Box.
Routing	Opens the Routing Rules Dialog Box.
Fanout	Opens the Fanout Rules Dialog Box.
Pad Entry	Opens the Pad Entry Rules Dialog Box.
Report	Opens the Rules Report Dialog Box.
Picture below rule button	The picture below each type of rule button identifies which rules hierarchy level is used for that rule type. The meaning of the picture corresponds to the button in the Hierarchy area of the Rules dialog box. For example, if you select a class in the Class list and a green polygon appears below the Clearance button, then the default values apply to the class.
Selected:	Lists the components selected in the Components list.

Table 45-85. Component Rules Dialog Box (cont.)

Name	Description
Part type:	Lists the part type(s) associated with the component(s) selected in the Components list.
Decal:	Lists the decal(s) associated with the component(s) selected in the Components list.
Default	Removes non-default rules from the selected components, so that only default rules apply.

Related Topics

Creating Component Design Rules

Modifying Component Design Rules

Resetting Component Rules to Default Rules

Design Rule Hierarchy

Conditional Rule Setup Dialog Box

Use the Conditional Rule Setup dialog box to setup clearance and high-speed rules that come into effect only when objects named in the rule are adjacent or on a specific layer. For example, the default clearance between nets is X, but when net A is adjacent to net B, the clearance is Y.

Requirement: The Advanced Rules option is required to setup conditional design rules.

Tip: Conditional rules override design rules in the design hierarchy and high-speed rules.

Accessing

- **Setup** menu > **Design Rules** > **Conditional Rules** button

Figure 45-93. Conditional Rule Setup Dialog Box

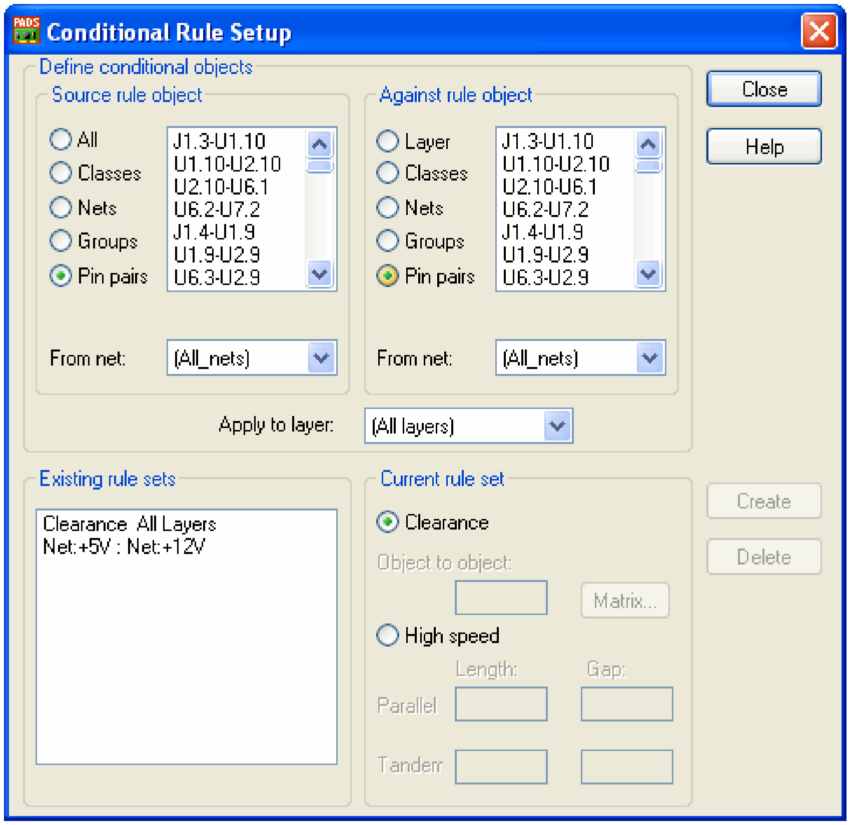


Table 45-86. Conditional Rule Setup Dialog Box Contents

Name	Description
Source rule object	Specifies the object for which to set the rule.
From net	Specifies to display pin pairs for a specific net. **Tip:** Select (All_nets) to display all pin pairs. **Restriction:** Available only if Pin pairs is selected in the Source rule object area.
Against rule object	Specifies the object against which to set the rule. **Tip:** For pin pair objects, you can display pin pairs for a net by selecting the net in the From net list.
From net	Specifies to display pin pairs for a specific net. **Tip:** Select (All_nets) to display all pin pairs. **Restriction:** Available only if Pin pairs is selected in the Against rule object area.

Table 45-86. Conditional Rule Setup Dialog Box Contents (cont.)

Name	Description
Apply to layer	Specifies to apply the conditional rule to a specific layer. Tip: Select (All layers) to apply it to all layers. Restriction: Unavailable if Layer is selected in the Against rule object area.
Existing rule sets list	Lists all rules previously created.
Clearance	Specifies that this is a Clearance rule set. You can set the clearance for all objects in the Object to object box. But this box is unavailable if there are differing values within the Clearance Rules matrix, and you must click Matrix, and enter the values within the Clearance Rules dialog box.
Matrix button	Opens the Clearance Rules Dialog Box.
High speed	Specifies that this is a High Speed rule set. Set the parallel and tandem length and gap in the boxes. **Tip:** For reporting purposes, nets and pin pairs in the Source rule object list are identified as aggressors. If a class is in the Source rule object list, all nets in the class are identified as aggressors.
Create	Creates the rule and displays it in the Existing rule sets list.
Delete	Removes the selected rule from the Existing rule sets list.

Related Topics

Deleting a Conditional Rule

Modifying a Conditional Rule

Design Rule Hierarchy

See the conditional rules topics listed in the Creating Rules for Your Design topic.

Confirm Pin Swap Dialog Box

Use the Confirm Pin Swap dialog box to confirm and proceed with swapping pins that do not have swap IDs or have different swap IDs.

Accessing

- **ECO Toolbar** > **Swap Pin** > select pins without a swap ID or with different swap IDs

Figure 45-94. Confirm Pin Swap Dialog Box

Table 45-87. Confirm Pin Swap Dialog Box Contents

Name	Description
Don’t display again	Prevents this prompt from appearing the next time you swap pins with different or undefined swap IDs. **Tip:** This setting is reset when you close and then return to the ECO Toolbar.
OK	Proceeds with the swap of the pins. **Tip:** If these pins are swappable, the correct process would be to give them identical swap IDs in the Pins tab of the Part Information dialog box.

Related Topics

Swapping a Pin Manually in ECO Mode

Connectivity Checking Setup Dialog Box

During design verification, you can use the Connectivity Check to report isolated routing vias and isolated stitching vias. (An isolated stitching via is a stitching via that is not connected to any hatch outline or copper area.) However, to have the check report isolated stitching vias, you must first set it up to ignore connections to CAM planes.

Accessing

- **Tools menu > Verify Design > Connectivity check > Setup button**

Figure 45-95. Connectivity Checking Setup Dialog Box

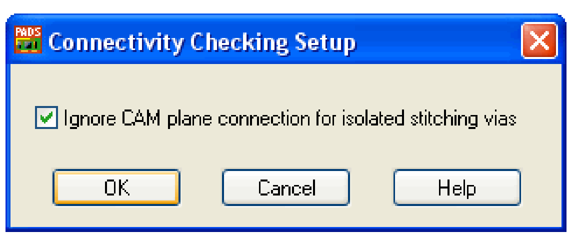


Table 45-88. Connectivity Checking Setup Dialog Box contents

Name	Description
Ignore CAM Plane Connection for isolated stitching vias	Specifies to report isolated stitching vias as errors and marks them in the design, along with the other errors found during checking.

Related Topics

Setting Up Checking for Isolated Stitching Vias

Convert Pin Pairs to Chamfered Paths Dialog Box

Use the Convert Pin Pairs to Chamfered Paths dialog box to convert traces to chamfered copper. Converting traces to a copper chamfered path has two advantages over simply creating a copper chamfered path. You can also use the more powerful interactive router to initially route the trace and when you convert the trace to a chamfered path, the net assignment is automatically made.

Accessing

- Select a pin pair, multiple pin pairs, or a net > Right-click > **Convert to Chamfered Paths**

  Restriction: Unrouted or partially routed pin pairs, or pin pairs belonging to reuse blocks, are excluded from selection.

Figure 45-96. Convert Pin Pairs to Chamfered Paths Dialog Box

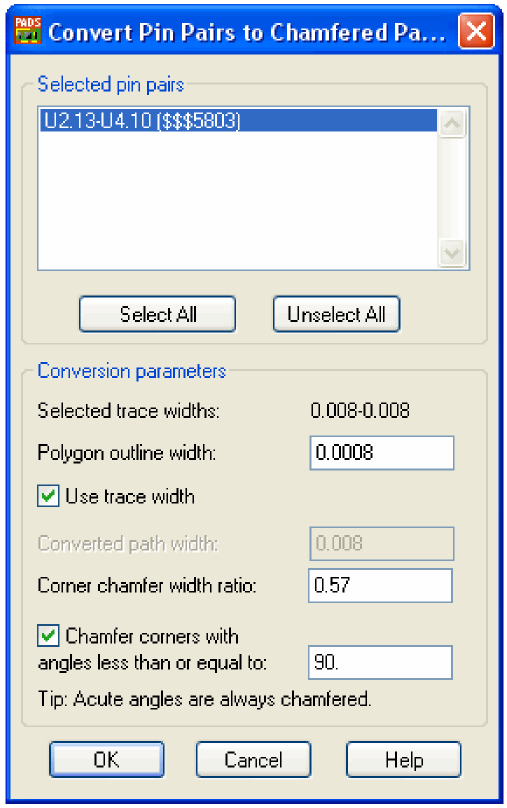


Table 45-89. Convert Pin Pairs to Chamfered Paths Dialog Box contents

Name	Description
Selected pin pairs list	Lists the pin pairs you selected in the design.
Select All	Selects all items listed in the Selected pin pairs list.
Unselect All	Deselects all items listed in the Selected pin pairs list.
Selected trace widths	Lists the range of widths for the selected traces.
Polygon outline width	Specifies a width value for the width of the copper outline. **Tip:** Since copper is created with an outline and a fill, you can specify a very narrow outline width to achieve very sharp corners. Decrease the value for sharper corners and increase the value for more blunt corners. All corners are rounded with a radius equal to one half of the outline width.

Table 45-89. Convert Pin Pairs to Chamfered Paths Dialog Box contents

Name	Description
Use trace width	Specifies to use the trace width as the width of the chamfered path. Where multiple trace widths exist, the actual trace widths are used. The Selected trace widths value at the top of the Conversion parameters area displays the range of trace widths of the items in the Selected pin pairs list. **Tip:** Click to clear this check box to enter a value in the Converted path width box.
Corner chamfer width ratio	Specifies the ratio of the chamfered corner width to the chamfered path width. If the ratio is 1.0, the width of the chamfered corner is the same as the chamfered path. Reduce the radio for a more narrow chamfered corner.
Chamfer corners with angles less than or equal to	Clear the **Chamfer corners with angles less than or equal to** check box to chamfer only angles less than 90 degrees (acute angles). or Select the **Chamfer corner with angles less than or equal to** check box to specify an angle between 90 and 180 degrees as the upper limit beneath which all angles are chamfered. Outside corners less than 90 degrees are always chamfered.

Related Topics

Converting a Trace to a Copper Chamfered Path

Copy to and Move to Dialog Boxes

Use the Copy to dialog box and the Move to dialog box to copy or move a project container or folder to another location in the vault.

Restriction: You cannot copy or move an item to any of the following locations:

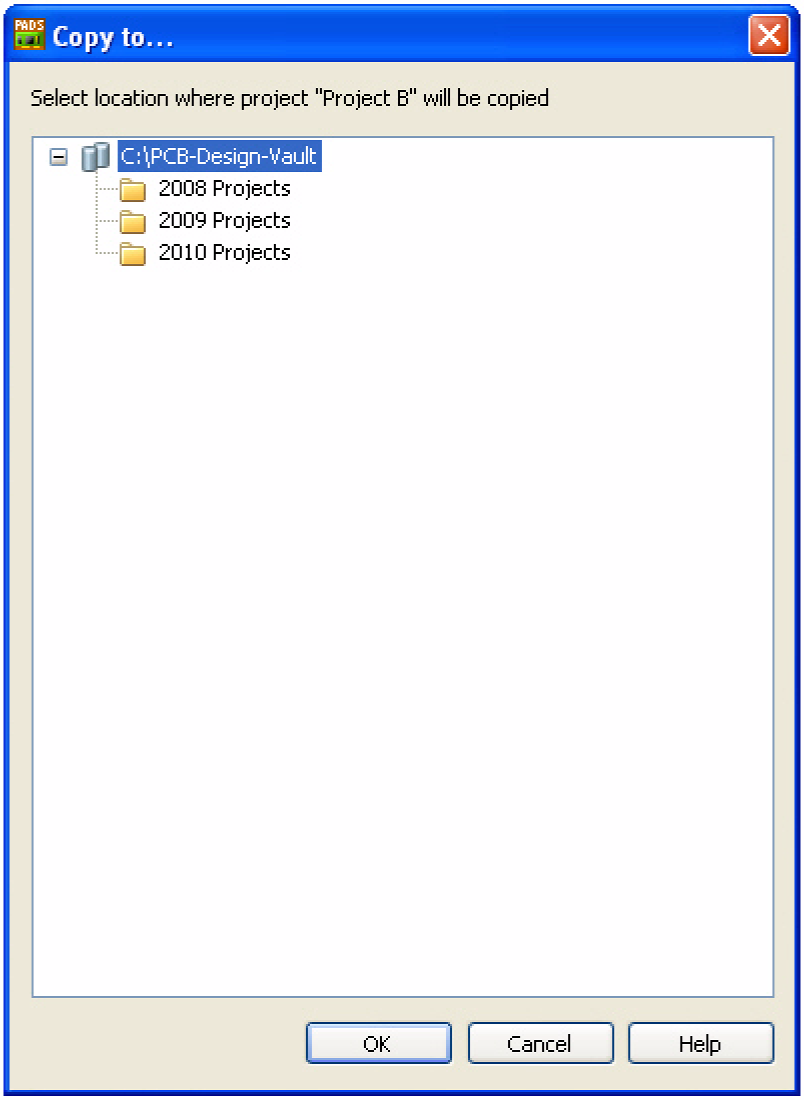
- To the same folder where the folder or project you want to move or copy currently resides
- To the same folder that you want to move or copy
- To a folder containing a project or folder with the same name as the folder or project you want to move or copy

If you select any of these locations in the dialog box tree, the OK button becomes unavailable.

Accessing

- In the **Vault** view, right-click a project or folder, and click **Copy to or Move to.**

Figure 45-97. Copy to Dialog Box



Crash Detected Dialog Box

The Crash Detected dialog box opens at a crash and allows you to save a report of the PADS environment as well as pertinent files into a compressed PADS Dump File. You can then submit this file to Mentor Customer Support. You can attach feedback to this report, and optionally, the BMW media and project files.

Accessing

- This dialog box is inaccessible unless the software crashes and crash detection is enabled in the software .ini file.

Figure 45-98. Crash Detected Dialog Box

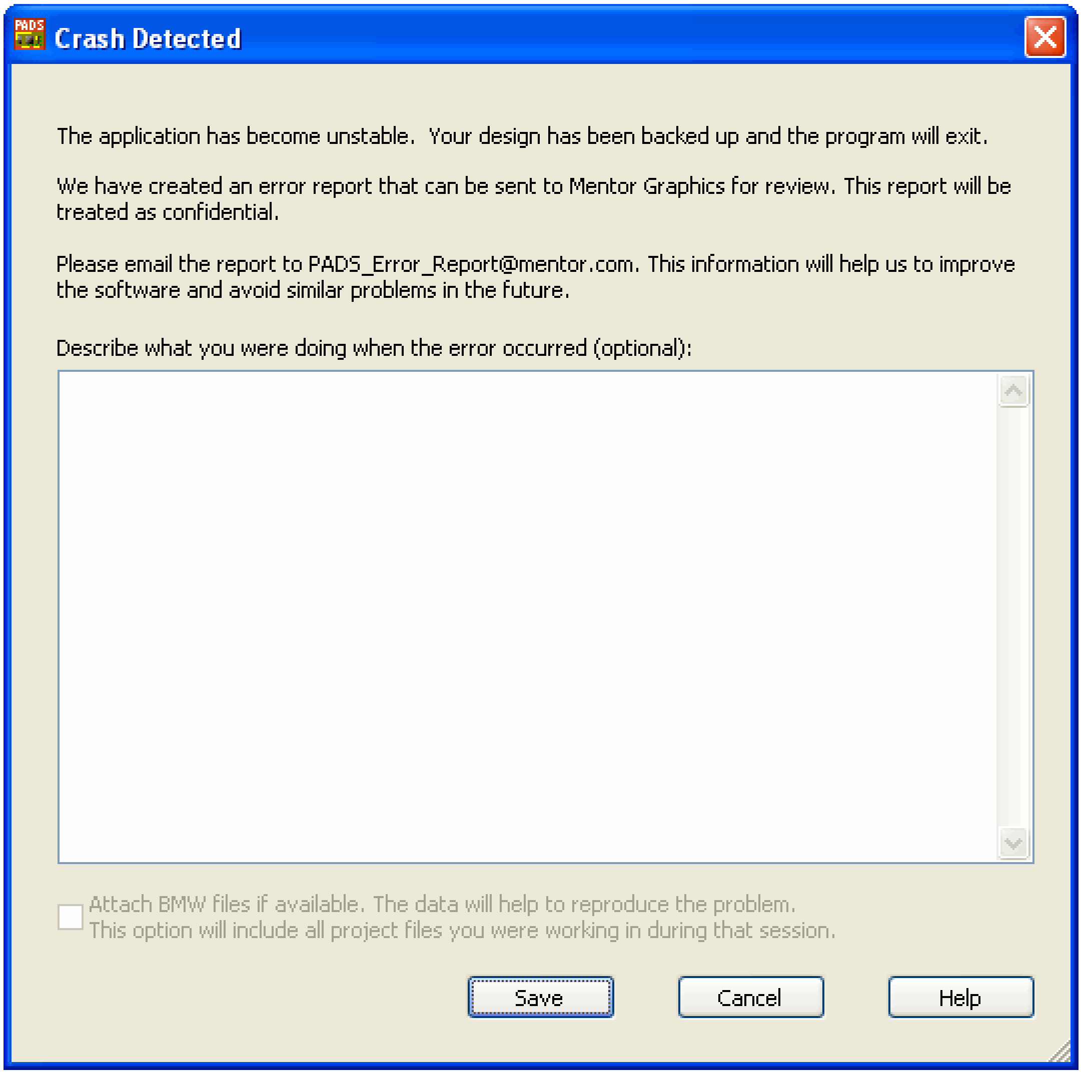


Table 45-90. Crash Detected Dialog Box Contents

Name	Description
Comments box	You can describe what you were doing when the error occurred or anything else you can think of that might help when investigating the crash.
Attach BMW data check box	You can include BMW data and your project files. This will allow customer support to play back what you were doing in your design that led up to the crash. This check box is unavailable if the BMW feature is not enabled. **See also:** BMW and BLT

Table 45-90. Crash Detected Dialog Box Contents

Name	Description
Save button	You must click the Save button if you want to create a report file. When you click the Save button, you are prompted with a Save As dialog box. The file that is created is called a PADS Dump File and is compressed in the .zip format. This is the file that you must send to customer support. It will include the report, the BMW data and the project files.

Create Array Dialog Box

Use the Create Array dialog box to set options for creating an array.

Figure 45-99. Planar tab

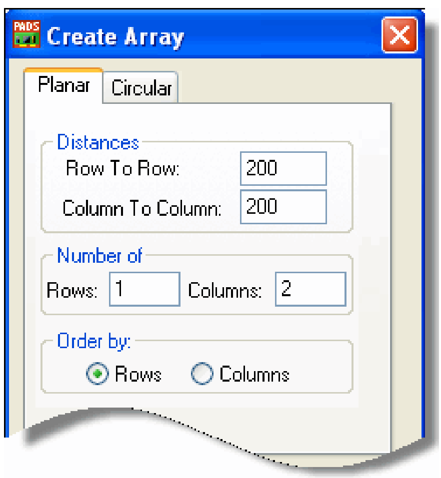


Figure 45-100. Circular tab

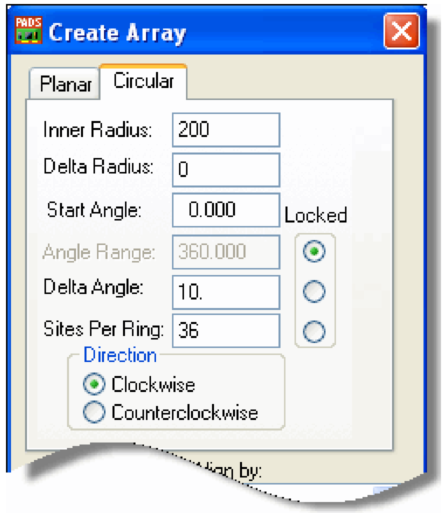


Figure 45-101. Create Array Dialog Box

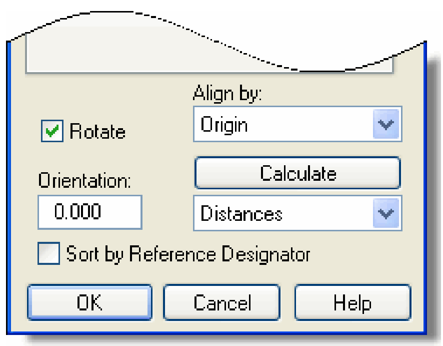


Table 45-91. Create Array Dialog Box Contents

Name	Description
Row To Row	Sets the row-to-row distance in current design units.
Column To Column	Sets the column-to-column distance in current design units.
Number of Rows	Sets the number of rows in the array. Type a value equal to or greater than 1. The Number of Columns is updated automatically based on this value.
Number of Columns	Sets the number of columns in the array. Type a value equal to or greater than 1. The Number of Rows is updated automatically based on this value.

Table 45-91. Create Array Dialog Box Contents (cont.)

<table>
<tr><th>Name</th><th>Description</th></tr>
<tr><td>Order by area</td><td>

Rows—Determines the sequence in which components are placed in the planar array.
When Order by Rows is selected, parts are sorted by selection and placed in array sites by rows, starting from the bottom row. Parts are placed in rows from left to right.
<table>
<tr><td>·6¤</td><td>·7¤</td><td>·8¤</td><td>·9¤</td><td>10¤</td></tr>
<tr><td>·1¤</td><td>·2¤</td><td>·3¤</td><td>·4¤</td><td>·5¤</td></tr>
</table>

Columns—Determines the sequence in which components are placed in the planar array.
When Order by Columns is selected, parts are sorted by selection and placed in array sites by rows, starting from the bottom row. Parts are placed in rows from left to right.
<table>
<tr><td>·4¤</td><td>·8¤</td></tr>
<tr><td>·3¤</td><td>·7¤</td></tr>
<tr><td>·2¤</td><td>·6¤</td></tr>
<tr><td>·1¤</td><td>·5¤</td></tr>
</table>

</td></tr>
<tr><td>Inner Radius</td><td>Sets the radius of the inner ring of the polar grid or circular array in current design units. You cannot use zero or negative values.</td></tr>
<tr><td>Delta Radius</td><td>Sets the radial distance between neighboring rings of the polar grid or circular array in current design units.
For Circular Arrays you cannot use zero or negative values.</td></tr>
<tr><td>Start Angle</td><td>Sets the polar angle, in degrees, of the first grid or circular array site. You can type a value between 0.000 and 359.999.</td></tr>
<tr><td>Angle Range</td><td>Sets the range within which you want to place objects. 360 sets a full circle grid or array; smaller values set sector-shaped grids or arrays.</td></tr>
<tr><td>Delta Angle</td><td>Sets the angular distance between neighboring sites within a ring.</td></tr>
<tr><td>Sites Per Ring</td><td>Sets the number of sites for each ring of the grid or array. Type a value equal to or greater than 2. You cannot use zero or negative values.</td></tr>
</table>

Table 45-91. Create Array Dialog Box Contents (cont.)

Name	Description
Locked	Controls the automatic adjustment of Angle Range, Delta Angle, and Sites Per Ring for Radial Move. Controls the automatic adjustment of Count, Angle, and Angle Range for Polar Step and Repeat. The three settings above, Angle Range, Delta Angle, and Sites per Ring, are interdependent; each value depends on the values in the other two. Set one of the values and lock it. Set one of the unlocked values; the other unlocked value automatically updates. For example, if you set Angle Range to 360 and Sites Per Ring to 36, Delta Angle updates to 10.
Direction	Sets how the sites are placed on the grid or circular array: Clockwise or Counterclockwise.
Rotate	Sets whether to change the orientation of the selected components. To change the orientation select this option and type a value, in degrees, in the Orientation box. Clear Rotate to maintain each component's current orientation.
Orientation	Sets the orientation for the selected components. Type in an orientation value, in degrees, for PADS Layout to apply to each component in the array.
Sort by Reference Designator	Sorts selected items by reference designator when creating or modifying an array or during a Radial Move. When this option is cleared, items are sorted by the order in which they were selected.
Align by	Determines how to align the components in the array. • **Origin**—Snaps the origins of the components to the array sites. • **Midpoint**—Snaps the midpoints of the components to the array sites.
Calculate	Defines an array's parameters so that components are placed as closely to each other as possible without violating the Body-to-Body Clearance Design Rule. You can calculate the following: • **Everything**—For planar arrays, calculates Instances, Number of Columns, and Number of Rows. For Circular arrays, calculates all options except Start Angle and Direction. Calculate always calculates options for a single ring array with an Angle Range of 360 degrees. • **Distances**—Calculates Row To Row and Column To Column values. Planar arrays only. • **Radius, Delta Radius**—Calculates Inner Radius and Delta Radius based on all of the other values in the Circular tab. Circular arrays only.

Related Topics

To Set Up a Polar Grid

Defining Arrays

Create Die Dialog Box

Restriction: This information applies only to the BGA toolkit.

Accessing

- **BGA Toolbar button** > **Die Wizard button**

Figure 45-102. Create Die Dialog Box

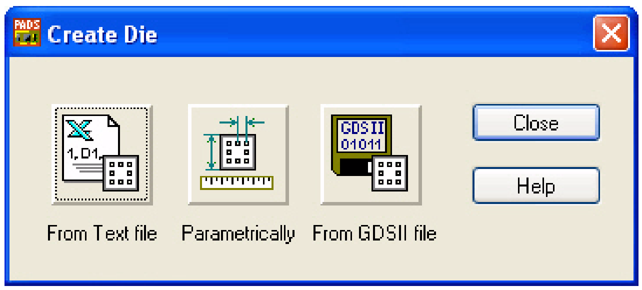


Table 45-92. Create Die Dialog Box contents

Name	Description
From Text File button	Opens the Die Wizard - Create from Text File dialog box
Parametrically button	Opens the Die Wizard - Create Parametrically dialog box
From GDSII File button	Opens the Die Wizard - Create from GDSII File dialog box

Related Topics

To Create a New Die

Create Empty Project Dialog Box

Before you can begin archiving a PCB project, you must create a Project container in the vault. The project will be associated with a working folder containing the PCB project files, and will serve as a container for the project’s archives.

Use the Create empty project dialog box to create a project in the vault.

Accessing

- In the **Vault** view, right-click a vault or a folder, and click **Create empty project.**

Figure 45-103. Create Empty Project Dialog Box

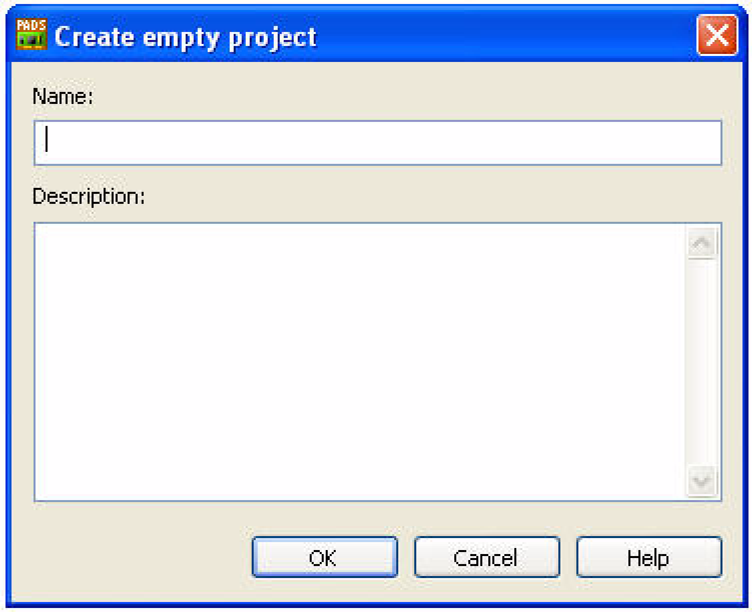


Table 45-93. Create Empty Project Dialog Box Contents

Name	Description
Name Description	Specify a name and description attribute for the project. **Tip:** Create a name and description you can search for with the Find in Vault tool.

Related Topics

Adding a Project Container to the Vault

Custom String Dialog Box

Use the Custom String dialog box to create custom header or footer text for PDF outputs.

Accessing

- **File** menu > **Create PDF** > in a header or footer position list, select **Custom**

Figure 45-104. Custom String Dialog Box

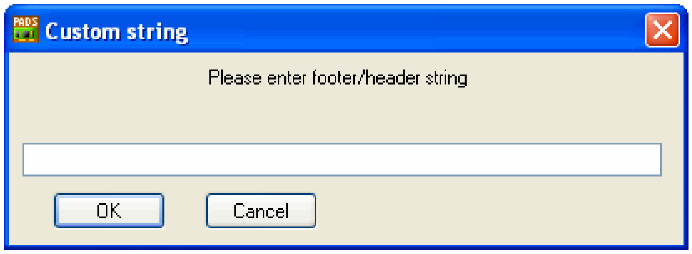


Table 45-94. Custom String Dialog Box Contents

Name	Description
text box	Type your custom text for the selected header or footer position.

Related Topics

PDF Configuration Dialog Box

Customize Dialog Box, Commands Tab

Use the Commands tab to add commands to menus or toolbars, or to create custom menus.

Accessing

- **Tools menu > Customize > Commands tab**

Figure 45-105. Commands Tab

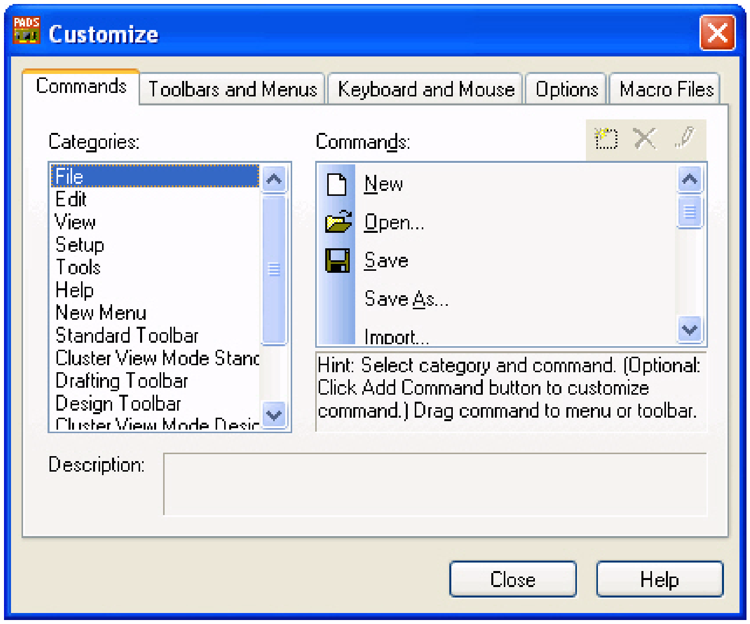


Table 45-95. Command Tab Contents

Name	Description
Categories list	Narrows down the list of commands.
Commands list	List of commands available to add to a menu or toolbar.
	Add a new command, delete a command you've added, or rename a command you've added.

Related Topics

Creating a Custom Command
Creating a Custom Menu
Defining Properties for a New Command
Editing a Custom Command
Adding Items to Toolbars and Menus

Customize Dialog Box, Keyboard and Mouse Tab

Create and customize shortcut keys using the Keyboard and Mouse tab of the Customize dialog box.

Accessing

- **Tools menu** > **Customize** > **Keyboard and Mouse tab**

Figure 45-106. Keyboard and Mouse Tab

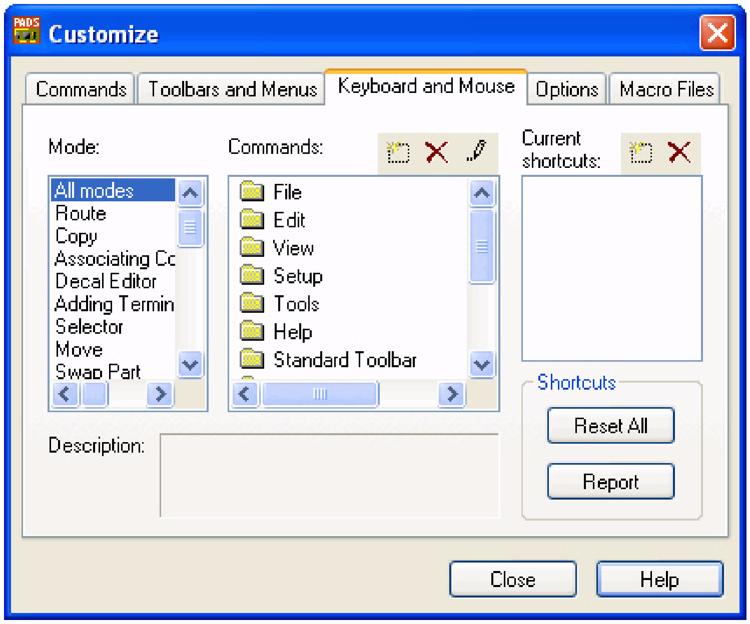


Table 45-96. Keyboard and Mouse Tab Contents

Name	Description
Mode list	Narrows down the list of commands.
Commands list	The list of commands available for which to assign a shortcut.
	Add a new command (opens the Add Command Dialog Box), delete a command you've added, or rename a command you've added (opens the Edit Command dialog box).
Current shortcuts list	The list of shortcuts assigned to the selected command.
	Add a new shortcut (open the Assign Shortcut Dialog Box), or delete a shortcut you've added.
Description	Lists what the selected command does.
Reset	Sets the selected toolbar or shortcut menu to the default settings.
Report	Saves a report of all current shortcut commands.

Related Topics

Customizing Shortcut Keys

Customize Dialog Box, Macro Files Tab

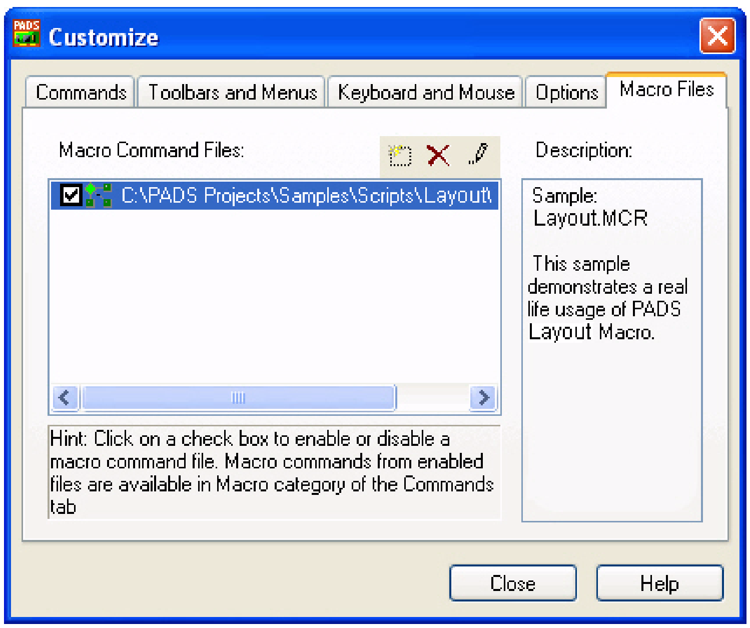
Create commands from macro files and add them to toolbars and menus using the Macro Files tab.

Tip: To create a command from a macro command file, the macro command file (.mcr) must already exist. You can create a macro by recording it in a PADS tool or scripting it in Macro language. For more information, see Creating Macros.

Accessing

- **Tools menu** > **Customize** > **Macro Files tab**

Figure 45-107. Macro Files Tab



Macro Files Tab Contents

Name	Description
Macro Command Files list	The list of macro files you have opened.
	Add a macro to the list (opens the Open Macro dialog box), delete a macro from the list, or edit the location of a macro you've added.
Description	Lists what the selected macro does.

Related Topics

Creating Macros

Customize Dialog Box, Options Tab

Customize the PADS interface by changing the appearance of menus and toolbars using the Options tab of the Customize dialog box.

Accessing

- **Tools menu** > **Customize** > **Options tab**

Figure 45-108. Options Tab

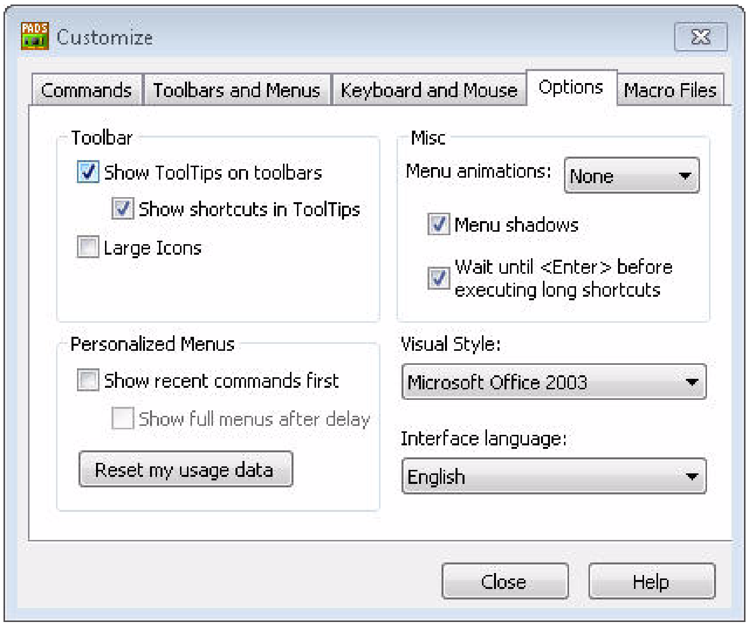


Table 45-97. Options Tab Contents

Name	Description
Show ToolTips on toolbars	Displays the button name over the toolbar button when you hover over it with your pointer.
Show shortcuts in ToolTips	In addition to the name in the ToolTip, displays the shortcut for the button.
Large Icons	Displays icons on the toolbar larger than the default size.
Menu animations list	The type of animation for your menus: None, Unfold, Slide, or Fade.

Table 45-97. Options Tab Contents

Name	Description
Menu shadows	Displays a shadow behind the menu.
Wait until <Enter> before executing long shortcuts	Delays the execution of shortcut keys until you press Enter.
Show recent commands first	Displays your recent menu command selections at the top of the list.
Show full menus after delay	Displays the full menu after a slight pause.
Reset my usage data	Restores the default set of commands to the menus and toolbars. **Tip:** This option does not undo any explicit customizations you made.
Visual Style	Sets the look and feel of your toolbars and title bars.
Interface Language	Specifies the language for all dialog boxes and messages displayed: English, Japanese, Brazilian Portuguese.

Related Topics

Customizing the Appearance of the Screen

Customize Dialog Box, Toolbars and Menus Tab

Use the Toolbars and Menus tab on the Customize dialog box to create custom toolbars and shortcut menus.

Tip: To create a custom main menu, use the Commands tab on the Customize dialog box. See Creating a Custom Menu.

Accessing

- **Tools menu > Customize** > **Toolbars and Menus tab**

Figure 45-109. Toolbars and Menus Tab

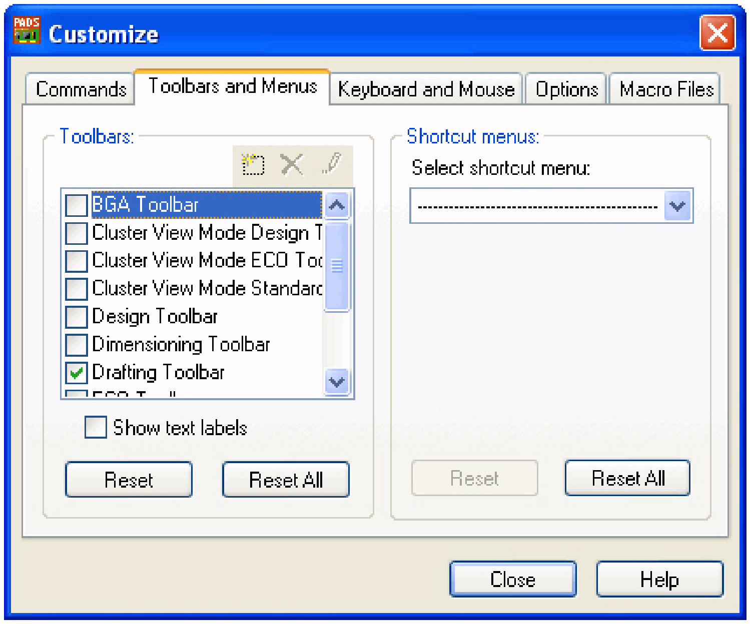


Table 45-98. Toolbars and Menus Tab Contents

Name	Description
Toolbars list	Specify which toolbars to display in the main window.
	Add a new toolbar, delete a toolbar you've added, or rename a toolbar you've added.
Show text labels	Shows the text label on the button in addition to the icon.
Select shortcut menus	Specifies the shortcut menu you want to customize. **Restriction:** PADS Router only.
Reset	Sets the selected toolbar or shortcut menu to the default settings.
Reset All	Sets all toolbars or shortcut menus back to their default settings.

Related Topics

Customizing Toolbars and Shortcut Menus

Decal Attributes Dialog Box

Use this dialog box to assign decal attributes, such as Geometry.Height. This information travels with the decal.

Accessing

- **Tools** menu > **PCB Decal Editor** > **Edit** menu > **Attribute Manager**

Figure 45-110. Decal attributes Dialog Box

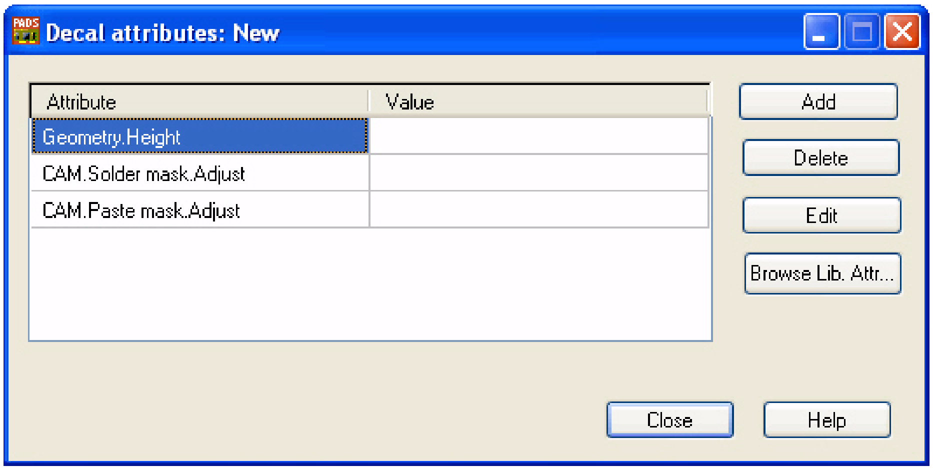


Table 45-99. Decal attributes Dialog Box Contents

Name	Description
Attribute table	• **Attribute column**—Lists the attributes assigned to the decal. • **Value column**—Lists the value of the attributes assigned to the decal. You can specify the units for the value.
Add	Adds a new row to the end of the Attribute table.
Delete	Removes the selected row from the Attribute table.
Edit	Makes the selected cell available for editing.
Browse Lib. Attr	Opens the Browse Library Attributes dialog box.

Decal Label Properties Dialog Box

Use the Decal Label Properties dialog box to modify a decal label or to change the attribute the label displays.

Tip: If you select multiple labels, settings in this dialog box apply to all selected labels.

Accessing

- Select a decal label > right-click > **Properties**

Figure 45-111. Decal Label Properties Dialog Box

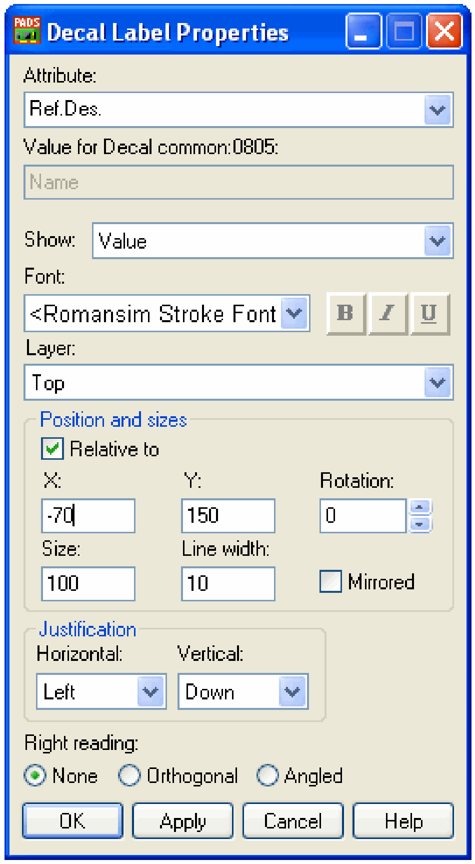


Table 45-100. Decal Label Properties Dialog Box Contents

Name	Description
Attribute	The attributes available to you. If you are creating labels for jumpers, Reference Designator is the only available attribute. **Tip:** Hidden attributes do not appear in the Attribute list unless the attribute was selected for the label before it was set as a hidden attribute.

Table 45-100. Decal Label Properties Dialog Box Contents (cont.)

Name	Description
Value for	The value of the selected attribute. **Tips:** • Unavailable if you selected Reference Designator or Part Type from the Attribute list, if the attribute is read-only, or if you open the Properties dialog box for labels of different attribute types. However, if the labels you select belong to attributes of the same type, you can edit this box. • If the attributes have different values, the box is blank. You can type a new value in the box to apply it to all of the selected attribute labels and their parent objects. • Value is also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.
Show	Controls the visibility of the label. • **None**—Turns visibility off. • **Value**— Displays only the label value. • **Name and Value**—Displays the name and value. • **Full Name and Value**—When labeling a structured attribute, displays the full structured name and value. **Tip:** Labels are invisible regardless of this setting unless you use the Display Colors Setup dialog box to change the color of labels to a color different from that of the background.
Font	The fonts available to you. **Tips:** • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.
Layer	The layers available to you.
Relative to	Places the label at the X and Y location relative to the component or the jumper. If you clear this check box, the label is placed at the X and Y location relative to the design origin.
X,Y	Places the decal label in a specified location.
Rotation	Specifies the rotation angle of the label.

Table 45-100. Decal Label Properties Dialog Box Contents (cont.)

Name	Description
Size	Specifies the size of the font. **Size (pts):** This is font size in points and appears for system fonts **Size (mils):** This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters. 
Line Width	Specifies the line width for stroke fonts only. 
Mirrored	Flips the label - text is considered readable from the bottom side of the board.
Horizontal, Vertical	Sets the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change. **Tips:** • For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**. • Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.
Right reading	Controls whether labels are readable (from left to right, or from bottom to top if the label is rotated). Click the **None**, **Orthogonal**, or **Angled** button to indicate the direction of reading you want.

Related Topics

Modifying Decal Label Properties

Decal Rules Dialog Box

Use the Decal Rules dialog box to define design rules that apply to decals.

Restrictions

-
- You can define Decal Rules in PADS Layout; however, these rules are used in PADS Router only.

Accessing

- **Setup** menu > **Design Rules** > **Decal** button

Figure 45-112. Decal Rules Dialog Box

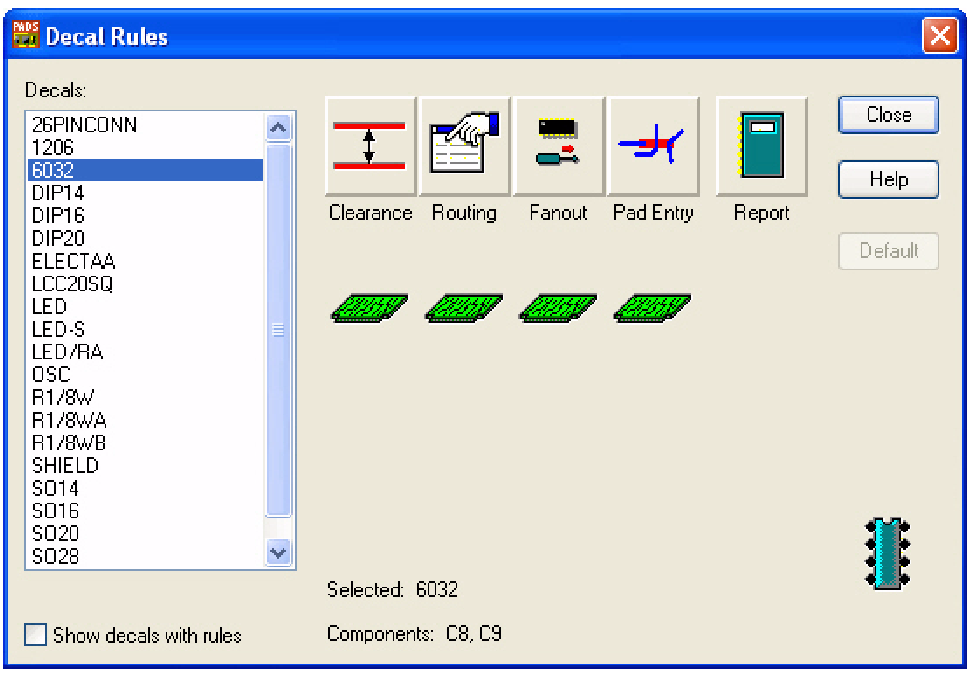


Table 45-101. Decal Rules Dialog Box

Name	Description
Decals list	Lists all decals in the design.
Show decals with rules	Specifies to show only decals that have rules.

Table 45-101. Decal Rules Dialog Box (cont.)

Name	Description
Clearance	Opens the Clearance Rules Dialog Box.
Routing	Opens the Routing Rules Dialog Box.
Fanout	Opens the Fanout Rules Dialog Box.
Pad Entry	Opens the Pad Entry Rules Dialog Box.
Report	Opens the Rules Report Dialog Box.
Picture below rule button	The picture below each type of rule button identifies which rules hierarchy level is used for that rule type. The meaning of the picture corresponds to the button in the Hierarchy area of the Rules dialog box. For example, if you select a class in the Class list and a green polygon appears below the Clearance button, then the default values apply to the class.
Selected:	Lists the decal(s) selected in the Decals list.
Components:	Lists the component(s) associated with decal(s) selected in the Connections list.
Default	Removes non-default rules from the selected decals, so that only default rules apply.

Related Topics

Creating Decal Design Rules

Modifying Decal Design Rules

Resetting Decal Rules to Default Rules

Creating Decal Design Rules in the PCB Decal Editor

Design Rule Hierarchy

Decal Rules Dialog Box (Decal Editor)

Use the Decal Rules dialog box to define design rules that apply to decals in the Decal Editor.

Restriction

- You can define Decal Rules in PADS Layout; however, these rules are used in PADS Router only.

Accessing

- **Tools** menu > **PCB Decal Editor**> **Setup** menu > **Decal Rules**

Figure 45-113. Decal Rules Dialog Box (Decal Editor)

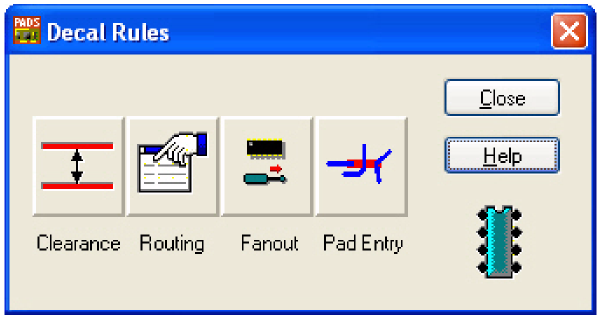


Table 45-102. Decal Rules Dialog Box (Decal Editor)

Name	Description
Clearance	Opens the Clearance Rules dialog box.
Routing	Opens the Routing Rules dialog box.
Fanout	Opens the Fanout Rules dialog box.
Pad Entry	Opens the Pad Entry Rules dialog box.

Related Topics

Creating Decal Design Rules in the PCB Decal Editor

Decal Wizard Dialog Box, BGA/PGA Tab

Use the BGA/PGA wizard to create ball grid array and pin grid array decals. You can create decal patterns for full and depopulated matrix BGAs/PGAs, including staggered arrays.

Accessing

- **Tools** menu > **PCB Decal Editor** > **Drafting Toolbar** button > **Wizard** button > **BGA/PGA** tab

The BGA/PGA tab controls change depending on your Device type selection. The two differences are:

- BGA/PGA Tab - Through hole Controls
- BGA/PGA Tab - SMD Controls

Figure 45-114. BGA/PGA Tab - Through hole Controls

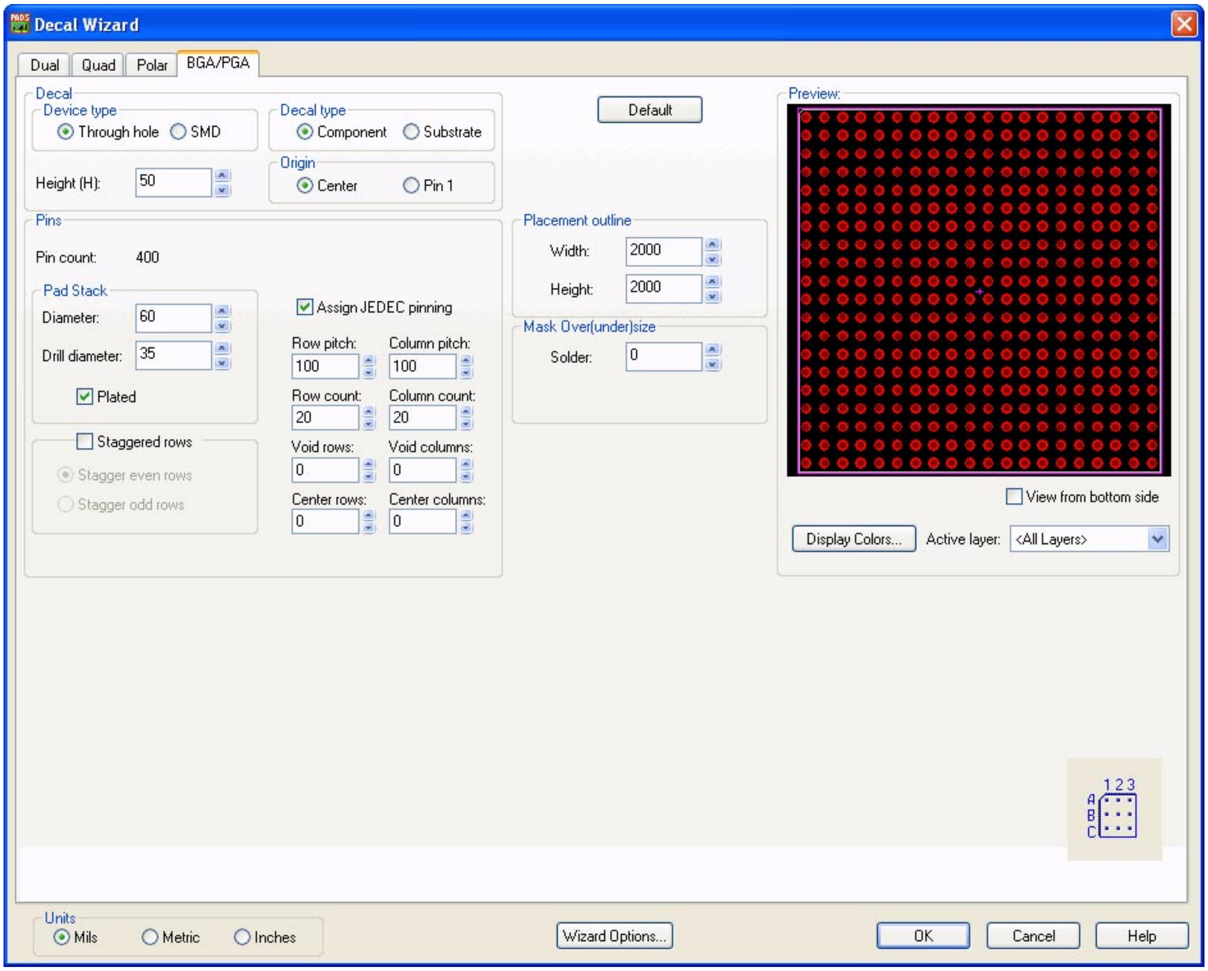


Figure 45-115. BGA/PGA Tab - SMD Controls

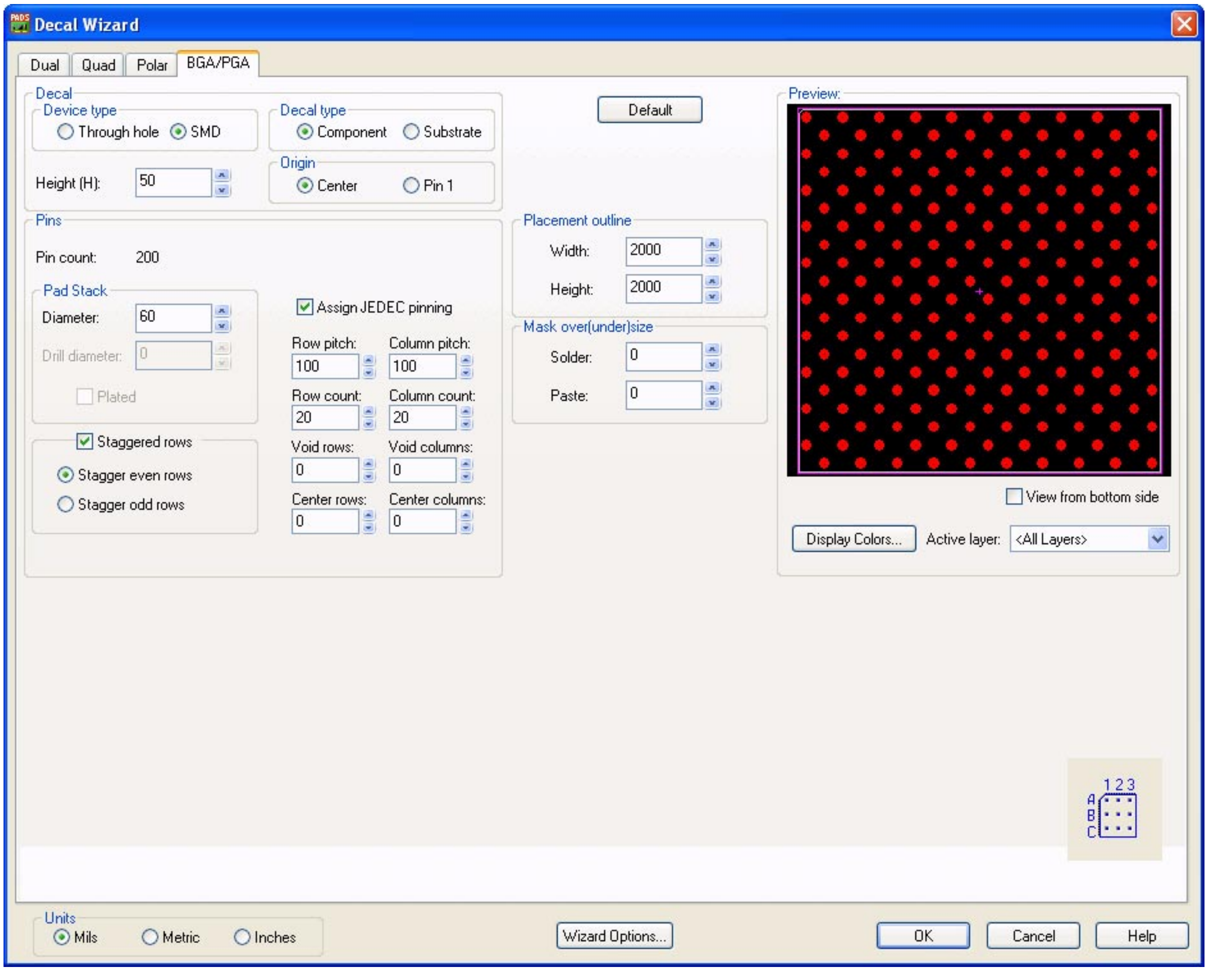


Table 45-103. BGA/PGA Tab Contents

Name	Description
Decal area	
Device type	Specifies whether the device will be a through hole or surface mount device. This setting is used to generate the correct type of pad stack for the device pins.
Decal type	Specifies whether the decal is a component or substrate. This setting mirrors the column numbering. Column numbering is left to right for component type and right to left for substrate type.
Height(H)	Specifies the height of the component. This value is added to the Geometry.Height attribute.

Table 45-103. BGA/PGA Tab Contents (cont.)

Name	Description
Origin	Specifies the origin of the decal: the center of the decal or pin 1.
Pins area	
Pin count	Displays the number of pins in the decal. You cannot edit this value; it depends on row count, column count, and the stagger pitch value.
Diameter	Sets the diameter of the pins in current units
Drill diameter	Sets the drill hole diameter in current units. **Restriction:** This control is only available when the Device type is set to Through hole.
Plated	Flags the through holes for plating. **Restriction:** This control is only available when the Device type is set to Through hole.
Staggered rows	Enables staggering of pins. Stagger even rows or odd rows.
Assign JEDEC pinning	Assigns an alphanumeric name to each pin in an array following the JEDEC standard. Pin rows are lettered from top to bottom starting with A. The letters I, O, Q, S, X, and Z are not used. For arrays with more than 20 rows, row 21 is designated AA. Subsequent rows are designated AB, AC, etc. Pin columns are numbered starting with 1. Column numbering is left to right for component type and right to left for substrate type.
Row pitch	Specifies the distance between rows of pins in current units
Row count	Sets the number of pin rows in the decal
Void rows	Specifies the number of rows in the decal that will be depopulated (void). Void rows are calculated from the center of the decal. When you set void rows, you must also set void columns. **Tip:** If the number of pin rows is an even number, the number of void rows should be an even number. If the number of pin rows is an odd number, the number of void rows should be an odd number.

Table 45-103. BGA/PGA Tab Contents (cont.)

Name	Description
Center rows	Specifies the number of pin rows in the decal to center within void rows. When you set center rows, you must also set center columns. **Tip:** If the number of void rows is an even number, the number of center rows should be an even number. If the number of void rows is an odd number, the number of center rows should be an odd number.
Column pitch	Specifies the distance between columns of pins in current units
Column count	Sets the number of pin columns in the decal
Void columns	Specifies the number of columns in the decal that will be depopulated (void). Void columns are calculated from the center of the decal. When you set void columns, you must also set void rows. **Tip:** If the number of pin columns is an even number, the number of void columns should be an even number. If the number of pin columns is an odd number, the number of void columns should be an odd number.
Center columns	Specifies the number of pin columns in the decal to center within void columns. When you set center columns, you must also set center rows. **Tip:** If the number of void columns is an even number, the number of center columns should be an even number. If the number of void columns is an odd number, the number of center columns should be an odd number.
Default	Sets all decal options to their default settings
Placement outline area **Restriction:** This area is unavailable if you've chosen not to create the outline in the Decal Wizard Options.	
Width	Specifies the width of the placement outline. The placement outline options are set in the Decal Wizard Options.
Height	Specifies the height of the placement outline. The placement outline options are set in the Decal Wizard Options.
Mask over(under)size area	

Table 45-103. BGA/PGA Tab Contents (cont.)

Name	Description
Solder	Specifies the oversize or undersize of the solder mask. The solder mask options are set in the Decal Wizard Options. **Restriction:** This area is unavailable if you've chosen not to create the solder mask outline in the Decal Wizard Options.
Paste	Specifies the oversize or undersize of the paste mask. The paste mask options are set in the Decal Wizard Options. **Restrictions:** • This control is only available when the Device type is set to SMD. • This area is unavailable if you've chosen not to create the paste mask outline in the Decal Wizard Options.
Preview area - displays a preview of the decal based on your current settings. The view displays what is inside the Placement outline.	
View from bottom side	Displays the decal from the bottom side in the Preview area.
Display Colors	Click to open the Display Colors Setup dialog box to change the colors of the preview window.
Active layer	Specifies the active layer to display in the preview window.
Units	Specifies whether the decal units are Mils, Metric, or Inches.
Wizard Options	Opens the Decal Wizard Options Dialog Box.

Related Topics

Creating a Basic Decal Automatically

Decal Wizard Dialog Box, Dual Tab

Use the Dual wizard to create both DIP and SMD decals.

Accessing

- **Tools** menu > **PCB Decal Editor** > **Drafting Toolbar** button > **Wizard** button > **Dual** tab

The Dual tab controls change depending on your Device type selection. The two differences are:

- Dual Tab - Through hole Controls
- Dual Tab - SMD Controls

Figure 45-116. Dual Tab - Through hole Controls

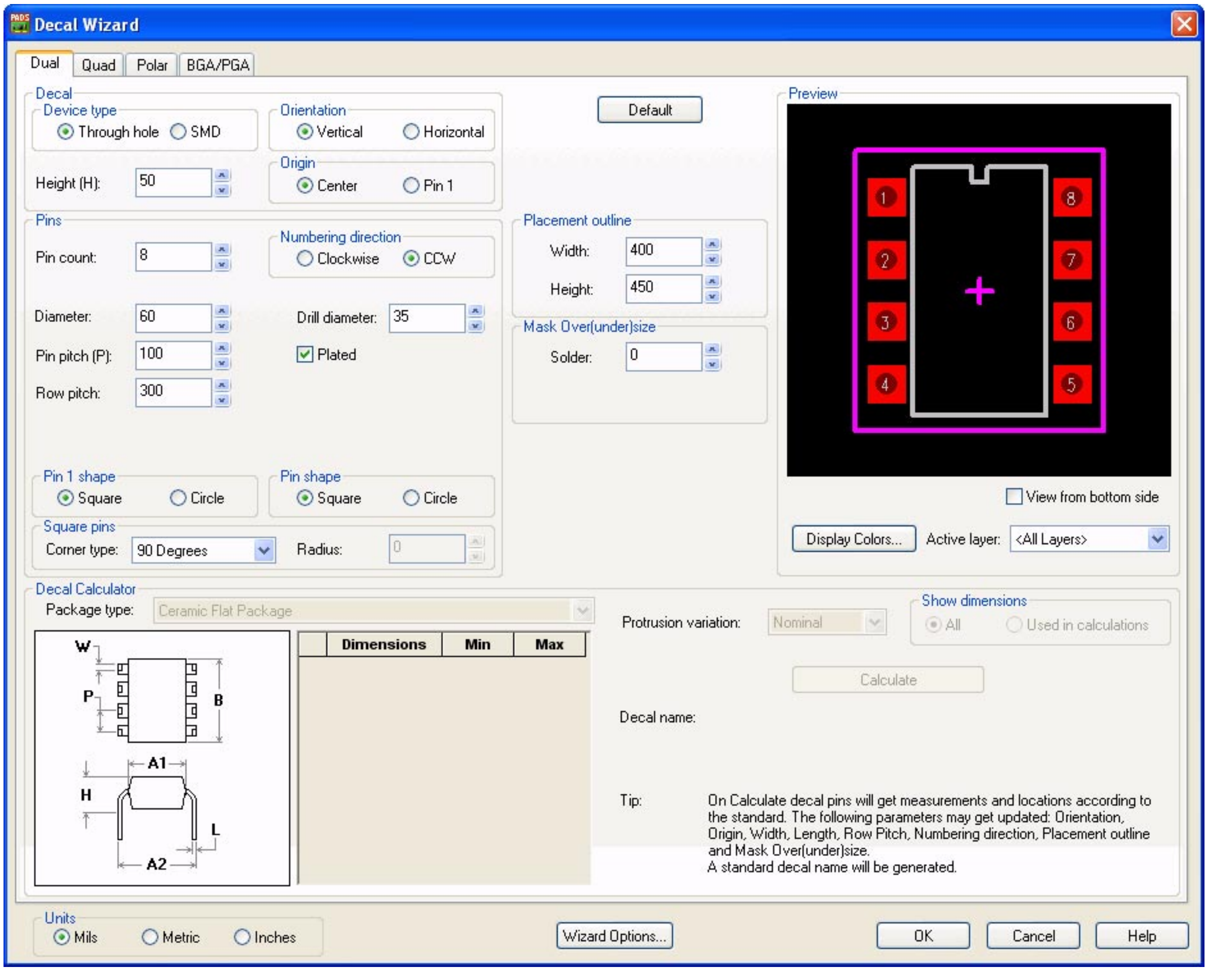


Figure 45-117. Dual Tab - SMD Controls

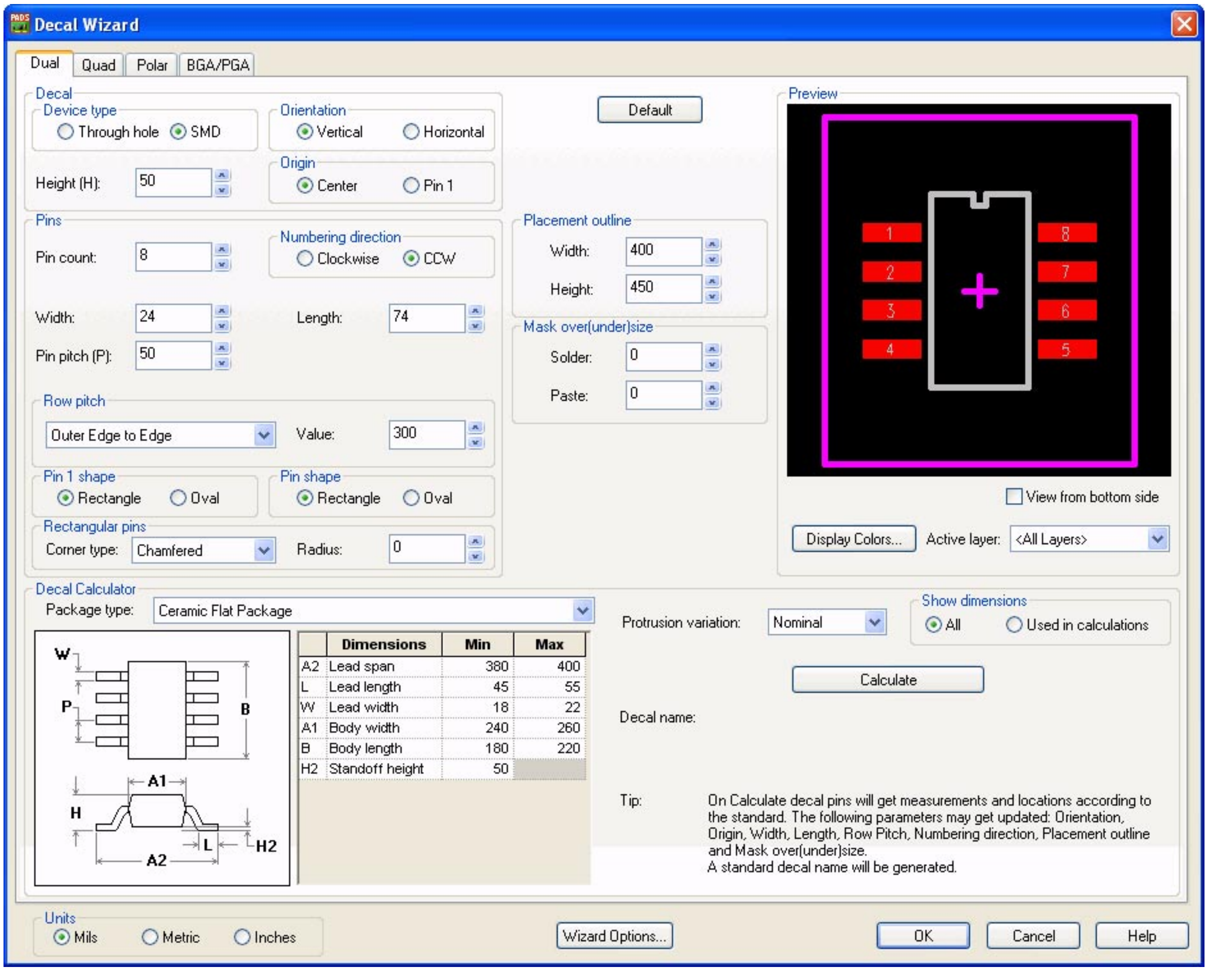


Table 45-104. Quad Tab Contents

Name	Description
Decal area	
Device type	Specifies whether the device will be a through hole or surface mount device. This setting is used to generate the correct type of pad stack for the device pins.
Orientation	Sets a vertical or horizontal orientation for the decal.
Height(H)	Specifies the height of the component. This value is added to the Geometry.Height attribute.
Origin	Specifies the origin of the decal: the center of the decal or pin 1.
Pins area	

Table 45-104. Quad Tab Contents (cont.)

Name	Description
Pin Count	Sets the number of pins in the decal.
Diameter	Sets the diameter of the pins in current units. **Restriction:** This control is only available when the Device type is set to Through hole.
Width	Sets the width of each pin in current units. **Restriction:** This control is only available when the Device type is set to SMD.
Pin Pitch	Sets the center-to-center spacing between pins, in current units.
Row Pitch	Sets the center-to-center spacing between rows of pins, in current units. **Restriction:** This control is only available when the Device type is set to Through hole. The area, of the same name, that applies to SMD devices is documented below.
Numbering direction	Specifies the direction in which to number pins: clockwise or counterclockwise (CCW).
Drill Diameter	Sets the drill hole diameter in current units. **Restriction:** This control is only available when the Device type is set to Through hole.
Plated	Plates pins. **Restriction:** This control is only available when the Device type is set to Through hole.
Length	Sets the length of each pin in current units. **Restriction:** This control is only available when the Device type is set to SMD.
Row pitch area **Restriction:** This control is only available when the Device type is set to SMD.	
List	Specifies the locations used to measure the pitch. • Center to Center—between centers of pins on opposite sides of the decal. • Inner Edge to Edge—between inner edges of pins on opposite sides of the decal. • Outer Edge to Edge—between outer edges of pins on opposite sides of the decal.
Value	Specifies the value of the pitch or pin rows.
Pin 1 shape	Specifies the shape of pin 1. Values change depending on whether the device type is through hole or SMD.

Table 45-104. Quad Tab Contents (cont.)

Name	Description
Pin shape	Specifies the shape of all pins except for pin 1. Values change depending on whether the device type is through hole or SMD.
Square/Rectangular pins area	
Corner type	Specifies the corner type used when pin shapes are square or rectangular.
Radius	Specifies the radius used for chamfered or rounded corners. **Restriction:** This control is not available for 90 Degrees.
Default button	Sets all decal options to their default settings.
Placement outline area **Restriction:** This area is unavailable if you've chosen not to create the outline in the Decal Wizard Options.	
Width	Specifies the width of the placement outline. The placement outline options are set in the Decal Wizard Options.
Height	Specifies the height of the placement outline. The placement outline options are set in the Decal Wizard Options.
Mask over(under)size area **Tip:** Half the oversize is added to either side of the pad.	
Solder	Specifies the oversize or undersize (use a negative value) of the solder mask. The solder mask options are set in the Decal Wizard Options. **Restriction:** This area is unavailable if you've chosen not to create the solder mask outline in the Decal Wizard Options.
Paste	Specifies the oversize or undersize (use a negative value) of the paste mask. The paste mask options are set in the Decal Wizard Options. **Restrictions:** • This control is only available when the Device type is set to SMD. • This area is unavailable if you've chosen not to create the paste mask outline in the Decal Wizard Options.
Preview area - displays a preview of the decal based on your current settings. The view displays what is inside the Placement outline.	

Table 45-104. Quad Tab Contents (cont.)

Name	Description
View from bottom side	Displays the decal from the bottom side in the Preview area.
Display Colors	Click to open the Display Colors Setup dialog box to change the colors of the preview window.
Active layer	Specifies the active layer to display in the preview window.
Decal Calculator area - generates the appropriate decal footprint based on the package dimensions. Uses the IPC-7351A standard and parameters that are set in the Decal Wizard Options - Package types tab**.** Restriction: The decal calculator is not available for through hole devices.	
Package type	Specifies the type of package and updates the package dimension diagram and variables on the parameter spreadsheet.
Package dimension diagram	Displays the usage of the variables for the package type. You can find the variable parameters in the parameter spreadsheet and also in various locations in the dialog box. For example, the H variable (height) is not found in the spreadsheet because it is located at the top of the Decal Wizard dialog box. See the Show dimensions area for a useful feature to display only those settings in the dialog box that are used by the Decal Calculator.
Package parameter spreadsheet	Specify the exact package dimensions and tolerances. Refer to the dimension diagram to the left of the spreadsheet to interpret the spreadsheet variables. Some variables are not found in the spreadsheet but are located in other parts of the dialog box. For example, the H variable (height) is not found in the spreadsheet because it is located at the top of the Decal Wizard dialog box.
Protrusion variation	Specifies the amount that the decal pins should protrude from under the component leads to achieve different amounts of solder welding. Choose from the three land pattern material conditions: minimum, nominal, and maximum. See the Decal Wizard Options - Package types tab for settings of these material conditions.
Show dimensions	Specify “All” to make all dialog box settings available or “Used in calculations” to gray out those settings not used by the Decal Calculator.

Table 45-104. Quad Tab Contents (cont.)

Name	Description
Calculate button	Click to generate the decal according to the Decal Calculator parameters and settings. **Tip:** The orientation is set to vertical, the origin is set to center, pin numbering is set to CCW. And the width, length, row pitch, placement outline and Mask over(under)size are calculated.
Decal name	Displays the generated name for the new decal in accordance with the IPC-7351A standard. **Restriction:** If the decal name is generated and then you change any decal parameter that was used in the name generation, the decal name is cleared and you must click the Calculate button to regenerate the new, correct name.
Units	Specifies whether the decal units are Mils, Metric, or Inches.
Wizard Options	Opens the Decal Wizard Options Dialog Box.

Related Topics

Creating a Basic Decal Automatically

Decal Wizard Dialog Box, Polar Tab

Use the Polar decal wizard to create through-hole or SMD decals with pins evenly distributed on the array of the specified radius.

Accessing

- **Tools** menu > **PCB Decal Editor** > **Drafting Toolbar** button > **Wizard** button > **Polar** tab

The Polar tab controls change depending on your Device type selection. The two differences are:

- Polar Tab - Through hole Controls
- Polar Tab - SMD Controls

Figure 45-118. Polar Tab - Through hole Controls

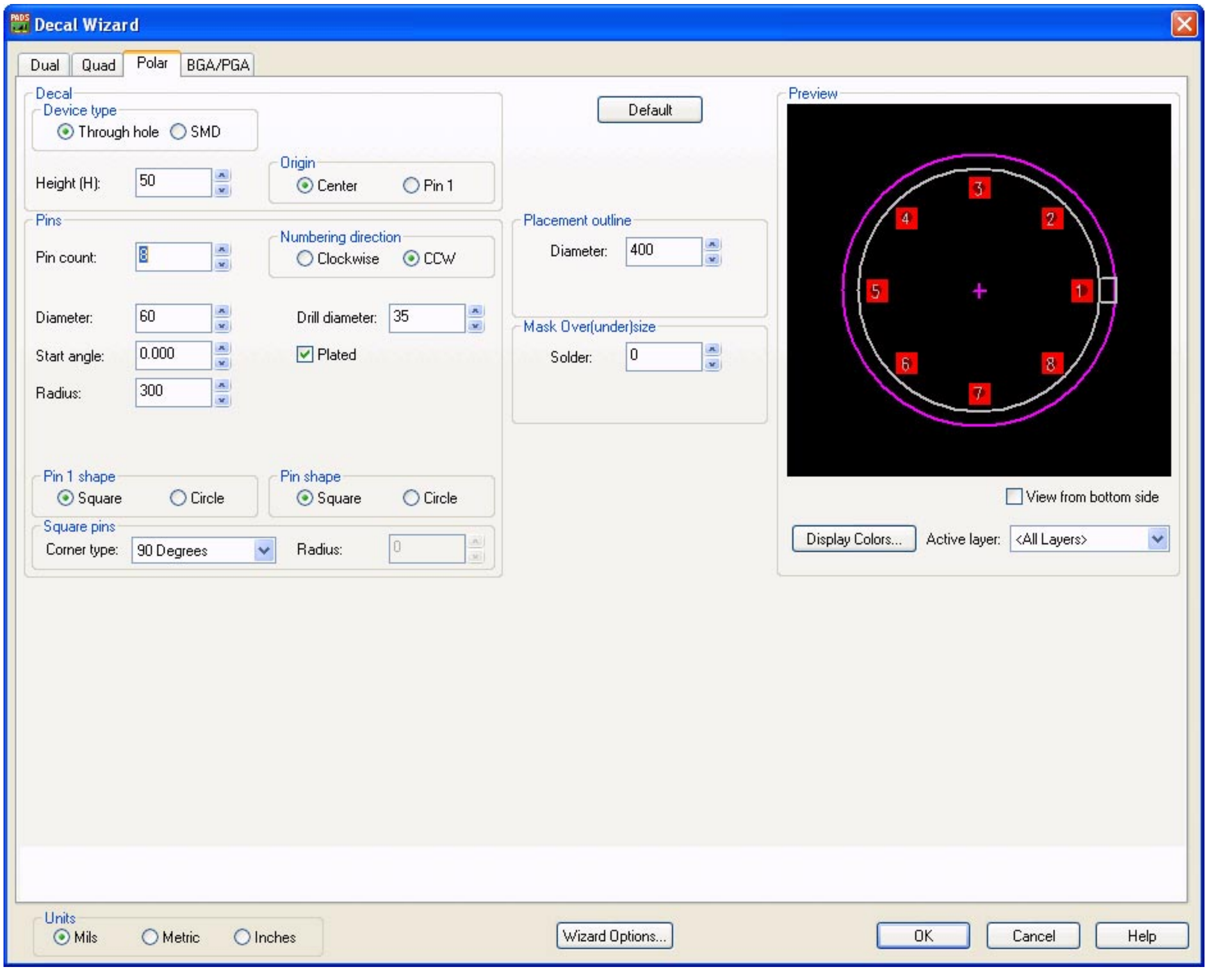


Figure 45-119. Polar Tab - SMD Controls

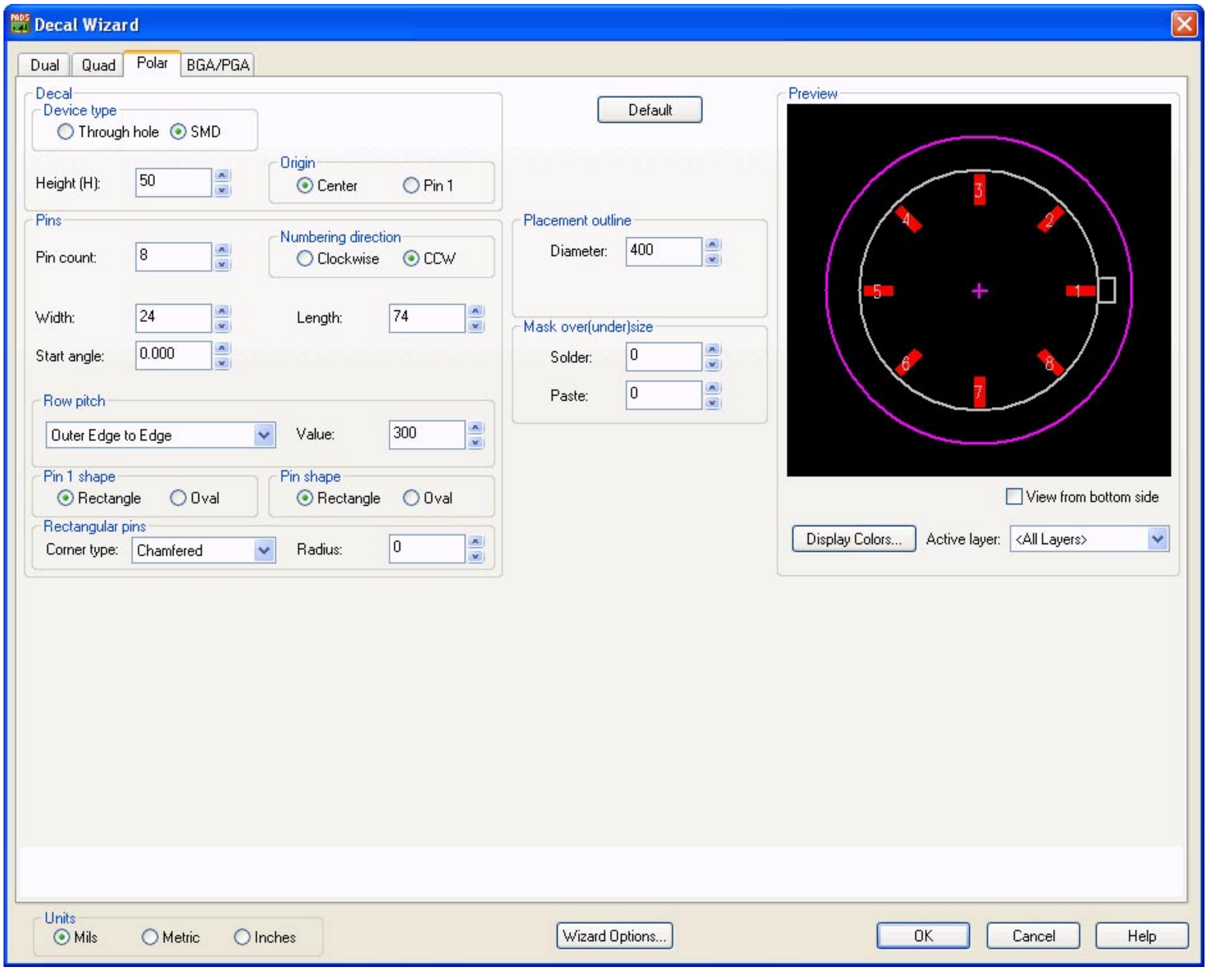


Table 45-105. Quad Tab Contents

Name	Description
Decal area	
Device type	Specifies whether the device will be a through hole or surface mount device. This setting is used to generate the correct type of pad stack for the device pins.
Height(H)	Specifies the height of the component. This value is added to the Geometry.Height attribute.
Origin	Specifies the origin of the decal: the center of the decal or pin 1.
Pins area	
Pin count	Sets the number of pins in the decal.

Table 45-105. Quad Tab Contents (cont.)

Name	Description
Diameter	Sets the diameter of the through hole pins. **Restriction:** This control is only available when the Device type is set to Through hole.
Width	Specifies the width of the surface mount pins. **Restriction:** This control is only available when the Device type is set to SMD.
Start angle	Sets the location, by angle, of the first pin on the circle.
Radius	Sets the radius of the circle for the array in current units. **Restriction:** This control is only available when the Device type is set to Through hole.
Numbering direction	Specifies the direction in which to number pins: clockwise or counterclockwise (CCW).
Drill diameter	Sets the drill hole diameter of the through hole pins. **Restriction:** This control is only available when the Device type is set to Through hole.
Plated	Marks the through holes for plating. **Restriction:** This control is only available when the Device type is set to Through hole.
Length	Specifies the length of the surface mount pins. **Restriction:** This control is only available when the Device type is set to SMD.
Row pitch area **Restriction:** This control is only available when the Device type is set to SMD.	
List	Specifies the locations used to measure the pitch. • Center to Center—between centers of pins on opposite sides of the decal. • Inner Edge to Edge—between inner edges of pins on opposite sides of the decal. • Outer Edge to Edge—between outer edges of pins on opposite sides of the decal.
Value	Specifies the value of the pitch or pin rows.
Pin 1 shape	Specifies the shape of pin 1. Values change depending on whether the device type is through hole or SMD.
Pin shape	Specifies the shape of all pins except for pin 1. Values change depending on whether the device type is through hole or SMD.
Square/Rectangular pins area	

Table 45-105. Quad Tab Contents (cont.)

Name	Description
Corner type	Specifies the corner type used when pin shapes are square or rectangular.
Radius	Specifies the radius used for chamfered or rounded corners. **Restriction:** This control is not available for 90 Degrees.
Default button	Sets all decal options to their default settings.
Placement outline area **Restriction:** This area is unavailable if you've chosen not to create the outline in the Decal Wizard Options.	
Diameter	Specifies the diameter of the placement outline. The placement outline options are set in the Decal Wizard Options.
Mask over(under)size area	
Solder	Specifies the oversize or undersize of the solder mask. The solder mask options are set in the Decal Wizard Options. **Restriction:** This area is unavailable if you've chosen not to create the solder mask outline in the Decal Wizard Options.
Paste	Specifies the oversize or undersize of the paste mask. The paste mask options are set in the Decal Wizard Options. **Restrictions:** • This control is only available when the Device type is set to SMD. • This area is unavailable if you've chosen not to create the paste mask outline in the Decal Wizard Options.
Preview area - displays a preview of the decal based on your current settings. The view displays what is inside the Placement outline.	
View from bottom side	Displays the decal from the bottom side in the Preview area.
Display Colors	Click to open the Display Colors Setup dialog box to change the colors of the preview window.
Active layer	Specifies the active layer to display in the preview window.
Units	Specifies whether the decal units are Mils, Metric, or Inches.

Table 45-105. Quad Tab Contents (cont.)

Name	Description
Wizard Options	Opens the Decal Wizard Options Dialog Box.

Related Topics

Creating a Basic Decal Automatically

Decal Wizard Dialog Box, Quad Tab

Use the QUAD wizard to create packages with leads extending from each of the four sides. For example, quad flat packs or plastic leaded chip carriers.

Accessing

- **Tools** menu > **PCB Decal Editor** > **Drafting Toolbar** button > **Wizard** button > **Quad** tab

Figure 45-120. Quad Tab

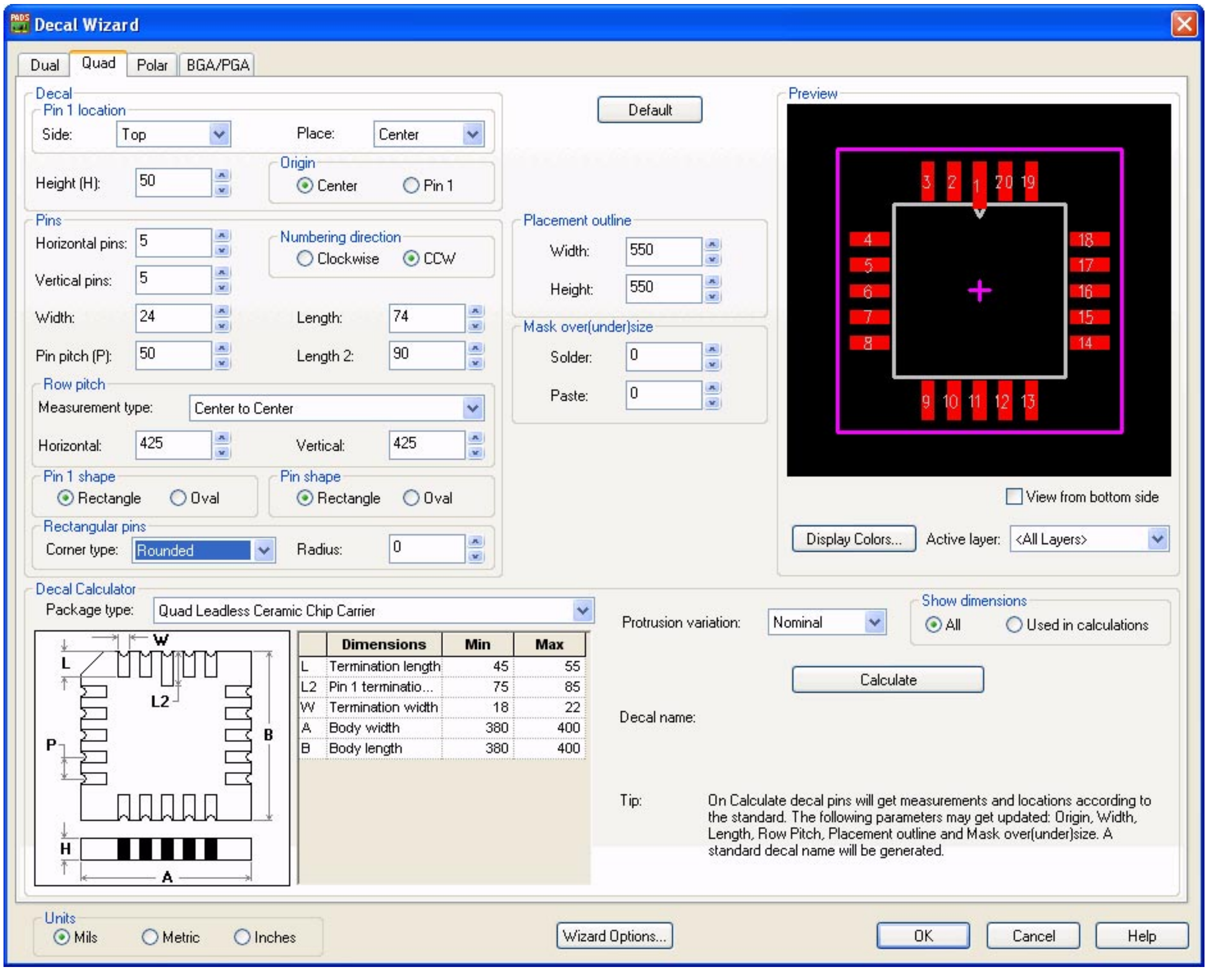


Table 45-106. Quad Tab Contents

Name	Description
Decal area	
Pin 1 location	Specify the side and place to locate pin 1.
Height(H)	Specifies the height of the component. This value is added to the Geometry.Height attribute.
Origin	Specifies the origin of the decal: the center of the decal or pin 1.
Pins area	
Horizontal pins	Sets the number of horizontal pins in each of the two rows.
Vertical pins	Sets the number of vertical pins in each of the two rows.

Table 45-106. Quad Tab Contents (cont.)

Name	Description
Width	Sets the width of each pin in current units.
Pin Pitch(P)	Sets the center-to-center spacing between pins, in current units.
Numbering direction	Specifies the direction in which to number pins: clockwise or counterclockwise (CCW).
Length	Sets the length of each pin in current units.
Length 2	**Sets the length of pin 1, which is different from the other pins.** **Restriction:** This control is only available for the Quad Leadless Ceramic Chip Carrier (with longer pin 1).
Row pitch area	
Measurement type	Specifies the locations used to measure the pitch. • Center to Center—between centers of pins on opposite sides of the decal. • Inner Edge to Edge—between inner edges of pins on opposite sides of the decal. • Outer Edge to Edge—between outer edges of pins on opposite sides of the decal.
Horizontal	Specifies the pitch of horizontal pins.
Vertical	Specifies the pitch of vertical pins.
Pin 1 shape area - Specifies the shape of pin 1.	
Pin shape area - Specifies the shape of all pins except for pin 1.	
Square/Rectangular pins area	
Corner type	Specifies the corner type used when pin shapes are square or rectangular.
Radius	Specifies the radius used for chamfered or rounded corners. **Restriction:** This control is not available for 90 Degrees.
Default button	Sets all decal options to their default settings.
Placement outline area **Restriction:** This area is unavailable if you've chosen not to create the outline in the Decal Wizard Options.	
Width	Specifies the width of the placement outline. The placement outline options are set in the Decal Wizard Options.

Table 45-106. Quad Tab Contents (cont.)

Name	Description
Height	Specifies the height of the placement outline. The placement outline options are set in the Decal Wizard Options.
Mask over(under)size area	
Solder	Specifies the oversize or undersize of the solder mask. The solder mask options are set in the Decal Wizard Options. **Restriction:** This area is unavailable if you've chosen not to create the solder mask outline in the Decal Wizard Options.
Paste	Specifies the oversize or undersize of the paste mask. The paste mask options are set in the Decal Wizard Options. **Restriction:** This area is unavailable if you've chosen not to create the paste mask outline in the Decal Wizard Options.
Preview area - displays a preview of the decal based on your current settings. The view displays what is inside the Placement outline.	
View from bottom side	Displays the decal from the bottom side in the Preview area.
Display Colors	Click to open the Display Colors Setup dialog box to change the colors of the preview window.
Active layer	Specifies the active layer to display in the preview window.
Decal Calculator area - generates the appropriate decal footprint based on the package dimensions. Uses the IPC-7351A standard and parameters that are set in the Decal Wizard Options - Package types tab**.**	
Package type	Specifies the type of package and updates the package dimension diagram and variables on the parameter spreadsheet.
Package dimension diagram	Displays the usage of the variables for the package type. You can find the variable parameters in the parameter spreadsheet and also in various locations in the dialog box. For example, the H variable (height) is not found in the spreadsheet because it is located at the top of the Decal Wizard dialog box. See the Show dimensions area for a useful feature to display only those settings in the dialog box that are used by the Decal Calculator.

Table 45-106. Quad Tab Contents (cont.)

Name	Description
Package parameter spreadsheet	Specify the exact package dimensions and tolerances. Refer to the dimension diagram to the left of the spreadsheet to interpret the spreadsheet variables. Some variables are not found in the spreadsheet but are located in other parts of the dialog box. For example, the H variable (height) is not found in the spreadsheet because it is located at the top of the Decal Wizard dialog box.
Protrusion variation	Specifies the amount that the decal pins should protrude from under the component leads to achieve different amounts of solder welding. Choose from the three land pattern material conditions: minimum, nominal, and maximum. See the Decal Wizard Options - Package types tab for settings of these material conditions.
Show dimensions	Specify “All” to make all dialog box settings available or “Used in calculations” to gray out those settings not used by the Decal Calculator.
Calculate button	Click to generate the decal according to the Decal Calculator parameters and settings. **Tip:** The the origin is set to center, pin numbering is set to CCW. And the width, length, row pitches, placement outline and Mask over(under)size are calculated.
Decal name	Displays the generated name for the new decal in accordance with the IPC-7351A standard. **Restriction:** If the decal name is generated and then you change any decal parameter that was used in the name generation, the decal name is cleared and you must click the Calculate button to regenerate the new, correct name.
Units	Specifies whether the decal units are Mils, Metric, or Inches.
Wizard Options	Opens the Decal Wizard Options Dialog Box.

Related Topics

Creating a Basic Decal Automatically

Decal Wizard Options Dialog Box, Global Tab

Use the Decal Wizard Options Global tab to enable or disable supplemental documentation layers and settings.

Accessing

- **PCB Decal Editor** > **Tools** menu > **Wizard Options** > **Global tab**
- **PCB Decal Editor** > **Drafting Toolbar** button > **Wizard Options button > Global tab**
- **PCB Decal Editor** > **Drafting Toolbar** button > **Wizard** button > **Wizard Options** > **Global tab**

Figure 45-121. Decal Wizard Options Dialog Box, Global Tab

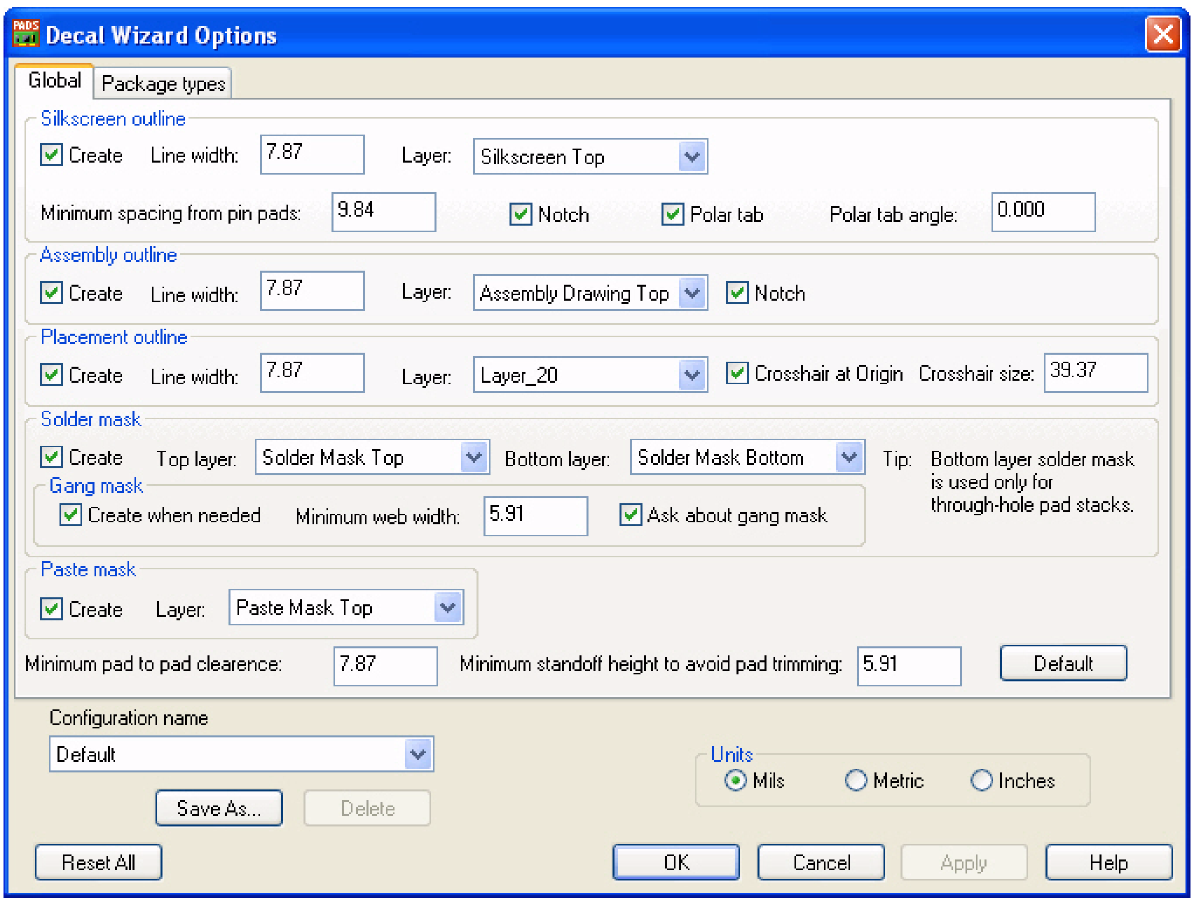


Table 45-107. Decal Wizard Options Dialog Box, Global Tab Contents

Name	Description
Silkscreen outline area	
Create	Specifies to create the decal silkscreen.
Line width	Specifies the width of the line used to draw the silkscreen.
Layer	Specifies the layer on which to draw the silkscreen.

Table 45-107. Decal Wizard Options Dialog Box, Global Tab Contents

Name	Description
Minimum spacing from pin pads	Specifies the gap between pad edges and the silkscreen outline edge.
Notch	Specifies to create a notch in rectangular silkscreen shapes.
Polar tab	Specifies to add a tab to polar (circular) silkscreen shapes.
Polar tab angle	Specifies the location of the polar tab in degrees away from pin 1.
Assembly outline area	
Create	Specifies to create the assembly outline. **Restrictions:** • The assembly drawing is generated only if you use the Decal Calculator to generate the decal. • The assembly drawing is generated only when you click the Calculate button in the Decal Wizard. If you change body width or length parameters after you click the Calculate button, the assembly drawing won't match the body of the component unless you click the Calculate button to generate a new assembly drawing with the new parameters.
Line width	Specifies the width of the line used to draw the assembly outline.
Layer	Specifies the layer on which to draw the assembly outline.
Notch	Specifies to create a notch in the assembly outline shape.
Placement outline area Tips: • The placement outline dimensions are defined in this dialog, but the calculation of the dimensions happens in the Decal Calculator, which puts the resulting values into the corresponding dialog fields. • The geometry of the placement outline is a rectangle surrounding the component with a clearance (the Courtyard excess value of the package type set in the Environment area of the Package types tab.) The clearance is calculated from the edges of pads or the component body, to the centerline of the placement outline.	
Create	Specifies to create the placement outline. **Tip:** Clearing this checkbox makes the Placement outline area unavailable in the Decal Wizard.
Line width	Specifies the width of the line used to draw the placement outline.
Layer	Specifies the layer on which to draw the placement outline.

Table 45-107. Decal Wizard Options Dialog Box, Global Tab Contents

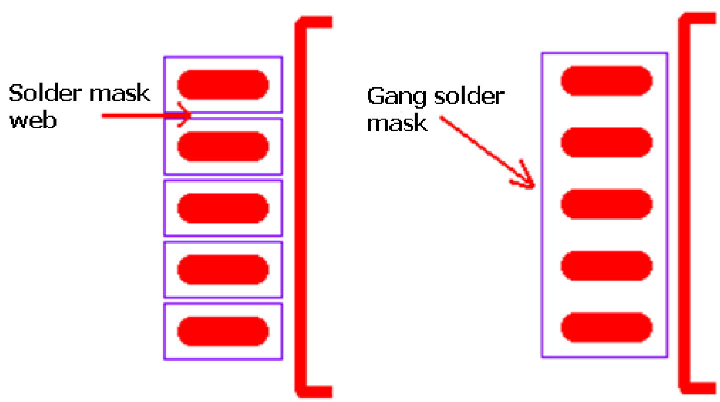
Name	Description
Crosshair at Origin	Specifies to create a crosshair geometry at the origin of the decal. **Tip:** The crosshair is created with the same Line width as the placement outline.
Crosshair size	Specifies the size of the crosshair. **Tip:** The width of the lines used in the crosshair are the same as the line width for the placement outline.
Solder mask area	
Create	Specifies to create the solder mask. **Tip:** Clearing this checkbox makes the Solder field unavailable in the Mask over(under)size area of the Decal Wizard.
Top Layer	Specifies the layer to use for top solder mask. **Restriction: Layers assigned to** top and bottom solder mask and paste mask may not be the same.
Bottom Layer	Specifies the layer to use for bottom solder mask. **Restriction: Layers assigned to** top and bottom solder mask and paste mask may not be the same.
Gang mask area	
Create when needed	Specifies to create gang solder mask when the width of the solder mask web becomes less than the minimum web width value.  **Tip:** Gang mask is a copper shape associated with the component. The shape of the gang mask takes on the shape of the pins. If the pins are oval shaped, the corners of the gang shape will be rounded. If one pin is longer than others, the gang will only extend beyond the regular pin length for the one pin.

Table 45-107. Decal Wizard Options Dialog Box, Global Tab Contents

Name	Description
Minimum web width	Specifies the minimum value of solder mask web allowed. Gang solder mask is created when the width of the web is less than this value and when you select the *Create when needed* check box.
Ask about gang mask	Specifies to prompt you upon clicking OK in the Decal Wizard, when gang mask conditions exist. You can choose to accept the conditions or a gang mask will not be created. It will make individual (overlapping) solder mask pads.
Paste mask area	
Create	Specifies to create paste mask. **Tip:** Clearing this checkbox makes the Paste field unavailable in the Mask over(under)size area of the Decal Wizard.
Layer	Specifies the layer on which to create paste mask. **Restriction: Layers assigned to** top and bottom solder mask and paste mask may not be the same.
Minimum pad to pad clearance	Specifies the minimum spacing allowed between decal pads. **Tip:** Pads are created regardless. But you are given a warning that the minimum spacing is violated.
Minimum standoff height to avoid pad trimming	Specifies the minimum standoff height - the gap between the component body and the board. When this value is greater than the Standoff height (H2) value of SOIC and QFP type packages, the pads will be trimmed to avoid coming in contact with the component body.
Default	Resets all Global tab settings to their default values.
Configuration name	Specifies the saved configuration to use. When the settings of the dialog box are changed, you are prompted to save it to a configuration file. Configuration names can be a maximum of 30 characters. **Restriction:** The Default configuration cannot be changed. **Tip:** For more information, see the The Decal Wizard Options Configuration File in the *Concepts Guide*.

Table 45-107. Decal Wizard Options Dialog Box, Global Tab Contents

Name	Description
Save As	Allows you to save the current settings in a configuration for reuse. Opens the Save Configuration dialog box. **Restriction:** Configuration names can be a maximum of 30 characters. **Tip:** Configuration files are stored in the “UserDir” folder as set in the powerpcb.ini file and has the extension .dwc. For more information, see the The Decal Wizard Options Configuration File in the *Concepts Guide*.
Units	Specify whether the values are Mils, Metric, or Inches.
Reset All	Resets both Global and Package types tab (including all package types) settings to their default values.

Related Topics

Decal Wizard Options Dialog Box, Package Types Tab

Decal Wizard Options Dialog Box, Package Types Tab

Use the Decal Wizard Options Global tab to set the defaults for each of the Package types used by the Decal Calculator.

Accessing

- **PCB Decal Editor > Tools** menu > **Wizard Options > Package types tab**
- **PCB Decal Editor** > **Drafting Toolbar** button > **Wizard Options button > Package types tab**
- **PCB Decal Editor** > **Drafting Toolbar** button > **Wizard** button > **Wizard Options > Package types tab**

Figure 45-122. Decal Wizard Options Dialog Box, Package Types Tab

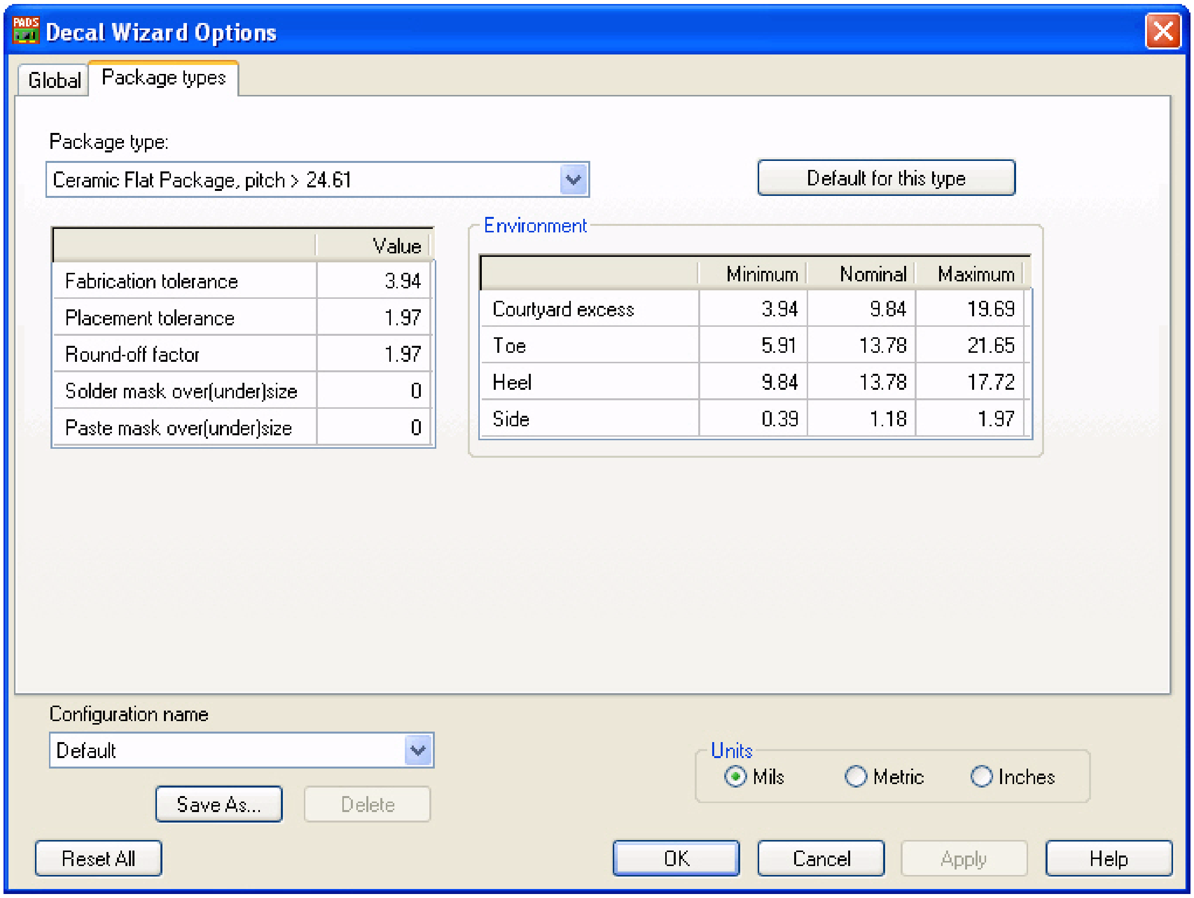


Table 45-108. Decal Wizard Options Dialog Box, Package Types Tab Contents

Name	Description
Package type list	Lists all the supported package types (and their pitch ranges) available to the Decal Calculator in the Decal Wizard. Select a package type from the list to view or modify the values used by the Decal Calculator to generate the decal. **Tip:** If you open this dialog box from the Decal Wizard, this list item will correspond to the package type and the pitch value you've chosen in the Decal Wizard.
Fabrication tolerance	Specifies the tolerance of the manufactured component and its land pattern. **Caution:** If you add a tolerance to your decals, inform your manufacturer in case they also compensate by over-sizing the land area and unnecessarily duplicate the tolerance.

Table 45-108. Decal Wizard Options Dialog Box, Package Types Tab Contents

Name	Description
Placement tolerance	Specifies the tolerance of the assembly process.
Round-off factor	Specifies the factor used to round calculated values.
Solder mask over(under)size	Specifies a positive value to oversize or a negative value to undersize the solder mask.
Paste mask over(under)size	Specifies a positive value to oversize or a negative value to undersize the paste mask.
Environment area **Tip:** This area changes according to each different package type.	
Courtyard excess	Specifies the extra clearance to apply to the placement outline in excess of the rectangle surrounding the component. The clearance is added between the edges of pads and the centerline of the placement outline. Specify the values for all three material conditions.
Toe	Specifies the extra protrusion of the pad beyond the toe of the lead. Specify the values for all three material conditions. **Restriction:** This is replaced by the single Periphery parameter for two package types.
Heel	Specifies the extra protrusion of the pad beyond the heel of the lead. Specify the values for all three material conditions. **Restriction:** This is replaced by the single Periphery parameter for two package types.
Side	Specifies the extra protrusion of the pad beyond the sides of the lead. Specify the values for all three material conditions. **Restriction:** This is replaced by the single Periphery parameter for two package types.
Periphery	Specifies one set of parameters instead of using Toe, Heel and Side parameters for both Pull-back Small Outline, No-Lead and Pull-back Quad Flat, No-Lead package types.
Default for this type	Resets the values to the default for the current package type listed.

Table 45-108. Decal Wizard Options Dialog Box, Package Types Tab Contents

Name	Description
Configuration name	Specifies the saved configuration to use. When the settings of the dialog box are changed, you are prompted to save it to a configuration file. Configuration names can be a maximum of 30 characters. **Restriction:** The Default configuration cannot be changed. **Tip:** For more information, see the The Decal Wizard Options Configuration File in the *Concepts Guide*.
Save As	Allows you to save the current settings in a configuration for reuse. Configuration names can be a maximum of 30 characters. **Tip:** Configuration files are stored in the "UserDir" folder as set in the powerpcb.ini file and has the extension .dwc. For more information, see the The Decal Wizard Options Configuration File in the *Concepts Guide*.
Units	Specify whether the values are Mils, Metric, or Inches.
Reset All	Resets both Global and Package types tab settings to their default values.

Related Topics

Decal Wizard Options Dialog Box, Global Tab

Default Rules Dialog Box

Use the Default Rules dialog box to define design rules that apply to all objects in the design, except for objects to which you assigned rules with a higher priority.

Accessing

- **Setup** menu > **Design Rules** > **Default** button

Figure 45-123. Default Rules Dialog Box

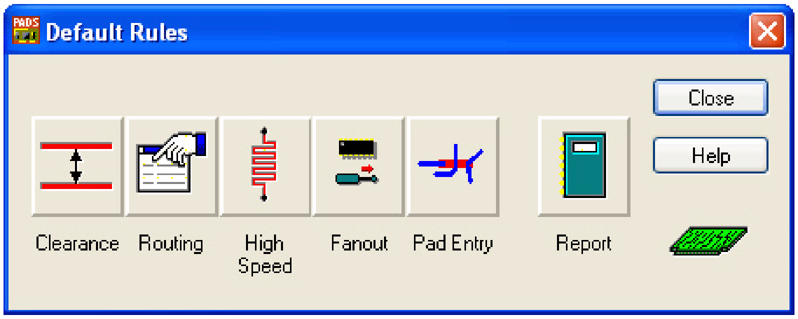


Table 45-109. Default Rules Dialog Box

Name	Description
Clearance	Opens the Clearance Rules Dialog Box.
Routing	Opens the Routing Rules Dialog Box.
High Speed	Opens the HiSpeed Rules Dialog Box.
Fanout	Opens the Fanout Rules Dialog Box.
Pad Entry	Opens the Pad Entry Rules Dialog Box.
Report	Opens the Rules Report Dialog Box.

Related Topics

Creating Default Rules

Design Rule Categories

Design Rule Hierarchy

Define CAM Documents Dialog Box

Use the Define CAM Documents dialog box to define and store up to 250 CAM documents.

Tip: You must save the design file before the changes made to the CAM documents become part of the design file.

Accessing

- **File Menu > CAM**

Figure 45-124. Define CAM Documents Dialog Box

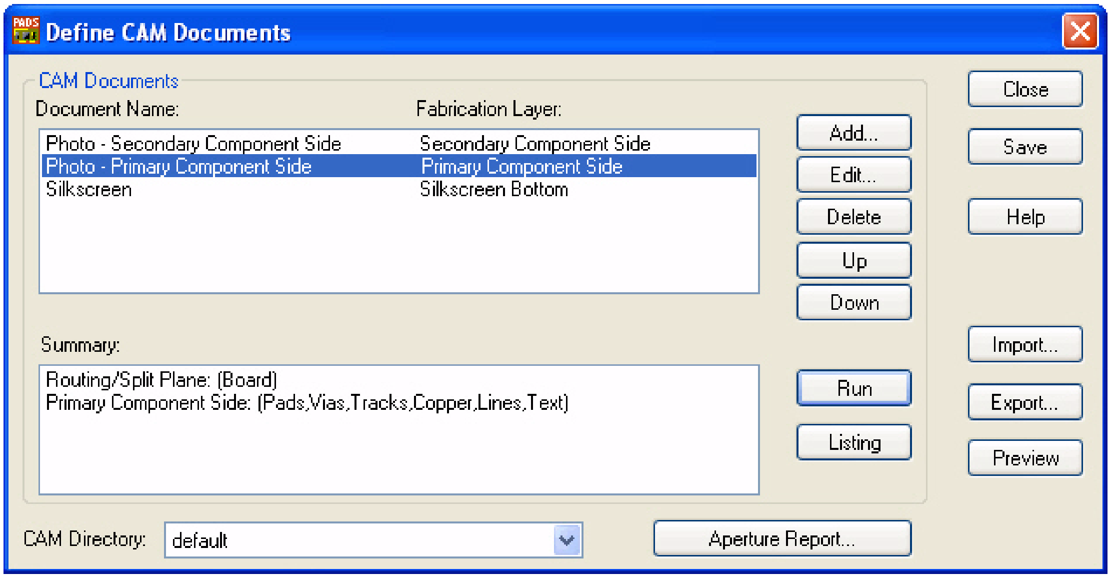


Table 45-110. Define CAM Documents Dialog Box contents

Name	Description
Document Name	The name of the CAM document.
Fabrication Layer	The layer on which you will be using CAM350 for post processing.
Add button	Opens the Add Document dialog box. Create a CAM Document configuration and return to the Define CAM Documents dialog box to save the configuration and Run the configuration against the design.
Edit button	Opens the Edit Document dialog box where you can edit the document settings.
Delete button	Removes the selected CAM document.
Up button	Moves the selected CAM document up one space.
Down button	Moves the selected CAM document down one space.
Summary	Displays a summary of the selected CAM document.
Run	Runs the selected CAM document configurations against your current design.
Listing	Opens the Listing file name dialog box where you can save a list of the CAM documents, which can also be printed and saved with your design documentation.

Table 45-110. Define CAM Documents Dialog Box contents (cont.)

Name	Description
CAM Directory	Specifies the location of the CAM output file. **Tips:** • The default location is \PADS Projects\Cam\default. • To create a new directory, select <Create> from the list.
Aperture Report	Produces a report of the apertures used in a CAM Document. **Requirement:** To produce an aperture report, you must have run the CAM Document(s) photo plot configuration against your design. If the CAM Document is set to print, or pen output, it will not produce the report.
Import	Opens the CAM import file name dialog box where you can recall an exported configuration to use as the CAM configuration for the .pcb file.
Export	Saves the configuration to a separate file that you can later import to use for similar .pcb files. **Tip:** If you name the file default.cam, it is used as the default CAM configuration for each .pcb file.
Preview	Opens the CAM Preview dialog box where you can see the results of a CAM Document configuration before you run the configuration against your design.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Defining CAM Documents

Define Name of Merged Net Dialog Box

You use the Define Name of Merged Net dialog box to rename the net of two pins connected using the Add Connection or Add Route ECO tools.

Accessing

- **ECO Toolbar** button > **Add Connection** button > click two pins with different netnames
- **ECO Toolbar** button > **Add Route** button > create a route between two pins with different netnames

Figure 45-125. Define Name of Merged Net Dialog Box

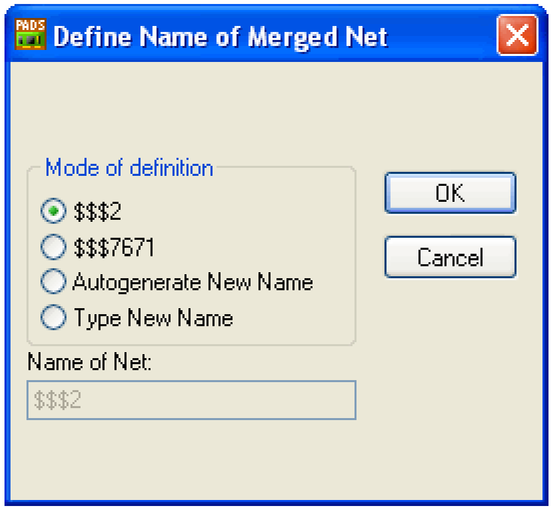


Table 45-111. Define Name of Merged Net Dialog Box Contents

Name	Description
Mode of definition area	This area allows you to choose the net name of the new merged net and provides you with four options. The first two options are the names of the two nets prior to their merging. **Autogenerate New Name**—assigns a new $$$<number> name to the merged net. **Type New Name**—allows you to type a new name in the Name of Net box.
Name of Net	Displays the autogenerated new name of the net, or allows you to type the name of the merged net. **Restriction:** This box is only available if you click Type New Name in the Mode of definition area.

Related Topics

Adding a Connection in ECO Mode

Adding a Route in ECO Mode

Define Name of New Net Dialog Box

You use the Define Name of New Net dialog box to name a new net created by the Delete Connection ECO tool or the Die Flag Wizard.

Tip: If you have deleted a connection bridging two pin pairs using the Delete Connection ECO tool, a prompt first appears and specifies which of the two pin pairs will be considered the new net. See Deleting a Connection in ECO Mode for more info.

Accessing

- **ECO Toolbar** button > **Delete Connection** button > click a connection bridging two pin pairs.

  or

- In the Die Flag Wizard dialog box, click **New Net**.

Figure 45-126. Define Name of New Net Dialog Box

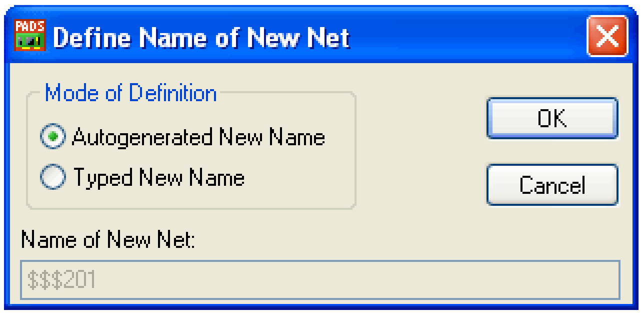


Table 45-112. Define Name of New Net Dialog Box Contents

Name	Description
Mode of Definition area	This area allows you to choose the net name of the new net. **Autogenerate New Name**—assigns a new $$$<number> name to the merged net. **Type New Name**—allows you to type a new name in the Name of Net box.
Name of New Net	Displays the autogenerated new name of the net, or allows you to type the name of the new net. **Restriction:** This box is only available if you click Type New Name in the Mode of definition area.

Delete Part Dialog Box

In ECO mode, use the Delete Part dialog box to delete parts, resulting single-pin nets, and optionally any attached traces. Two Delete Part dialog boxes appear in succession. The first dialog confirms the deletion and allows you to delete traces that are connected to the component, and the second dialog box informs you of any single-pin nets that will be deleted as a result from the deletion of the component.

Accessing

- **ECO Toolbar** button > **Delete Component** button > select one or multiple components
- **ECO Toolbar** button > select one or multiple components > press **Delete**

Example 45-1. Delete Part Dialog Box 1

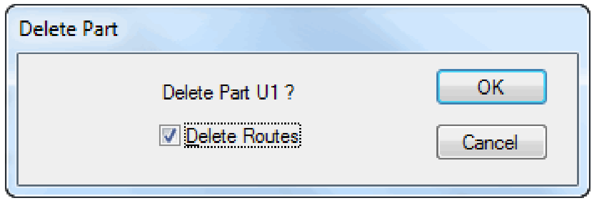


Table 45-113. Delete Part Dialog Box 1 Contents

Name	Description
Delete Part/Selected Parts	Prompts to confirm the deletion of the selected part(s). Specifies the reference designator of the part if you've only selected a single part.
Delete Routes	Select the check box to also delete all traces routed to and from this component. **Exception:** Despite this setting, the traces of any single-pin nets that result from the deletion of the component will always be deleted. **Tip:** If you have routed to and from a pad (both traces sharing the pad), you will lose both connections. But if you keep the traces when the component is removed, the connection will be continuous and will show a tack in place of the pad.

Figure 45-127. Delete Part Dialog Box 2

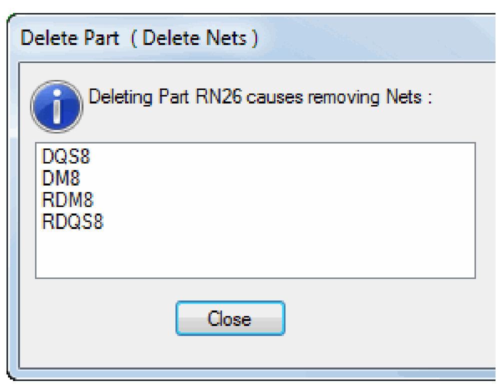


Table 45-114. Delete Part Dialog Box 2 Contents

Name	Description
Single-pin net list	When the selected component is deleted, single-pin nets might result. These nets are always deleted, including both unrouted connections and traces.

Related Topics

Deleting a Component in ECO Mode

Derive SBP Function from Netlist Dialog Box

Use the Derive SBP Function from Netlist dialog box to import the SBP functions from a BGA, PADS Layout, or PADS Logic netlist ASCII file.

Restriction: This information applies only to the BGA toolkit.

See also: Exporting ASCII Files

Tip: SBP numbers are used as they are currently defined in the SBP Properties dialog box.

Accessing

- **BGA Toolbar** button > **Wire Bond Wizard** button > **SBP Naming** button > **Derive from Netlist** button > open a file

Figure 45-128. Derive SBP Function from Netlist Dialog Box

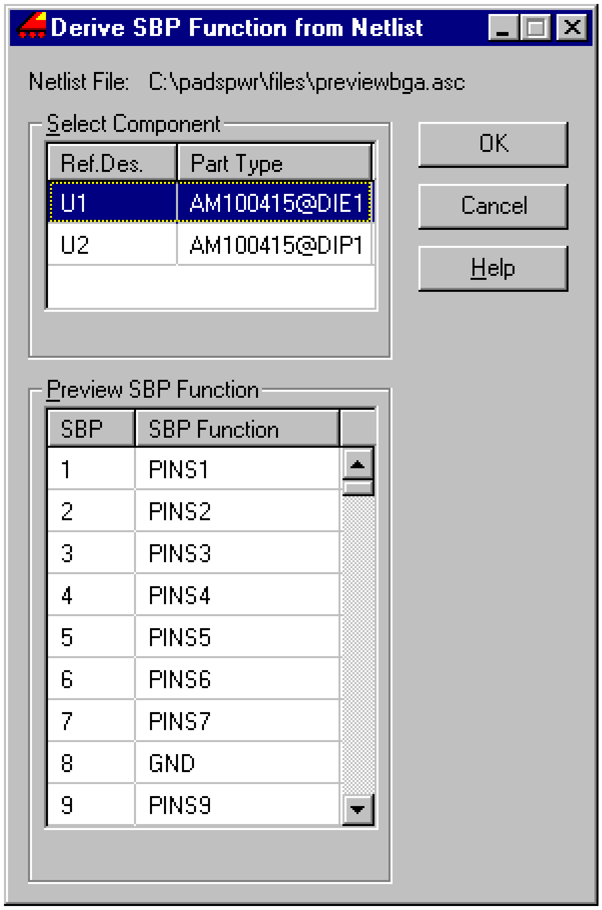


Table 45-115. Derive SBP Function from Netlist Dialog Box contents

Name	Description
Netlist File	The name of the file you are using.
Select Component area	Displays the reference designator and part type for all components in the net list from the imported ASCII file. Select a die component from the list. After selecting the component, the names in the SBP Function column of the Preview SBP Function area are updated with the net names assigned to pins of the selected components in the imported file.
Preview SBP Function area	This area contains a list with the columns SBP and SBP Function. When you select a component in the Select Component area, Wire Bond Wizard updates the names in the SBP Function column. These names are updated to the net names assigned to the pins of the selected component in the imported ASCII file.

Related Topics

To Import the SBP Functions

DFT Audit Dialog Box, Assignment Tab

Use the Assignment tab to prevent or favor assigning test points to components or to via types. By default, all pins on a net are available for test pin assignment and are evenly weighted as test point candidates.

Tip: To reduce design iterations, consult with your automated test engineers when setting DFT Audit options.

Accessing

- **Tools menu > DFT Audit > Assignment tab**

Figure 45-129. Assignment Tab

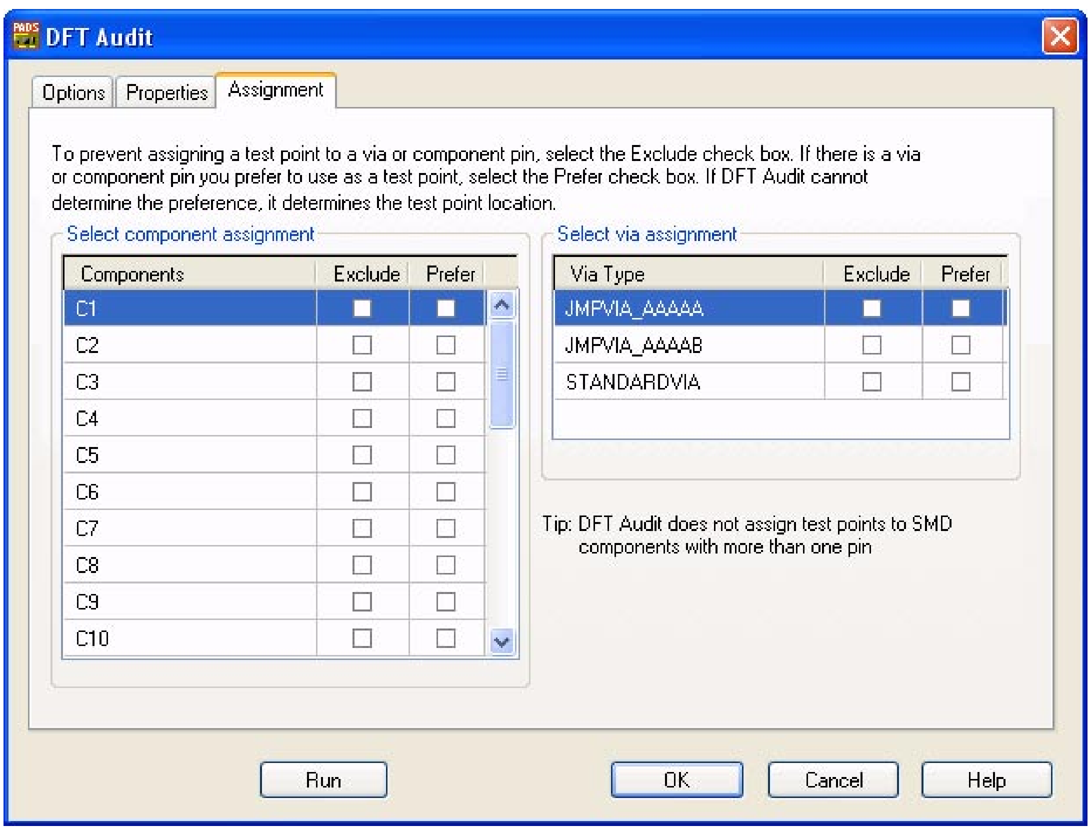


Table 45-116. Assignment Tab contents

Name	Description
Components column	Lists the components available to you.
Exclude column	Specifies to prevent the use of the component as a test point. **Tip:** apply even weighting, clear the **Exclude** and **Prefer** check boxes.
Prefer column	Specifies to favor the use of the component as a test point. **Tip:** apply even weighting, clear the **Exclude** and **Prefer** check boxes.
Via Type column	Lists the vias available to you.
Exclude column	Specifies to prevent the use of the via as a test point. **Tip:** apply even weighting, clear the **Exclude** and **Prefer** check boxes.
Prefer column	Specifies to favor the use of the via as a test point. **Tip:** apply even weighting, clear the **Exclude** and **Prefer** check boxes.
Run button	Starts the automatic audit process.

Related Topics

Setting Test Point Assignment Eligibility

DFT Audit Dialog Box, Options Tab

Several options for placing test points are available to you on the Options tab of the DFT Audit dialog box.

Tip: To reduce design iterations, consult with your automated test engineers when setting DFT Audit options.

Accessing

- **Tools menu > DFT Audit > Options tab**

Figure 45-130. Options Tab

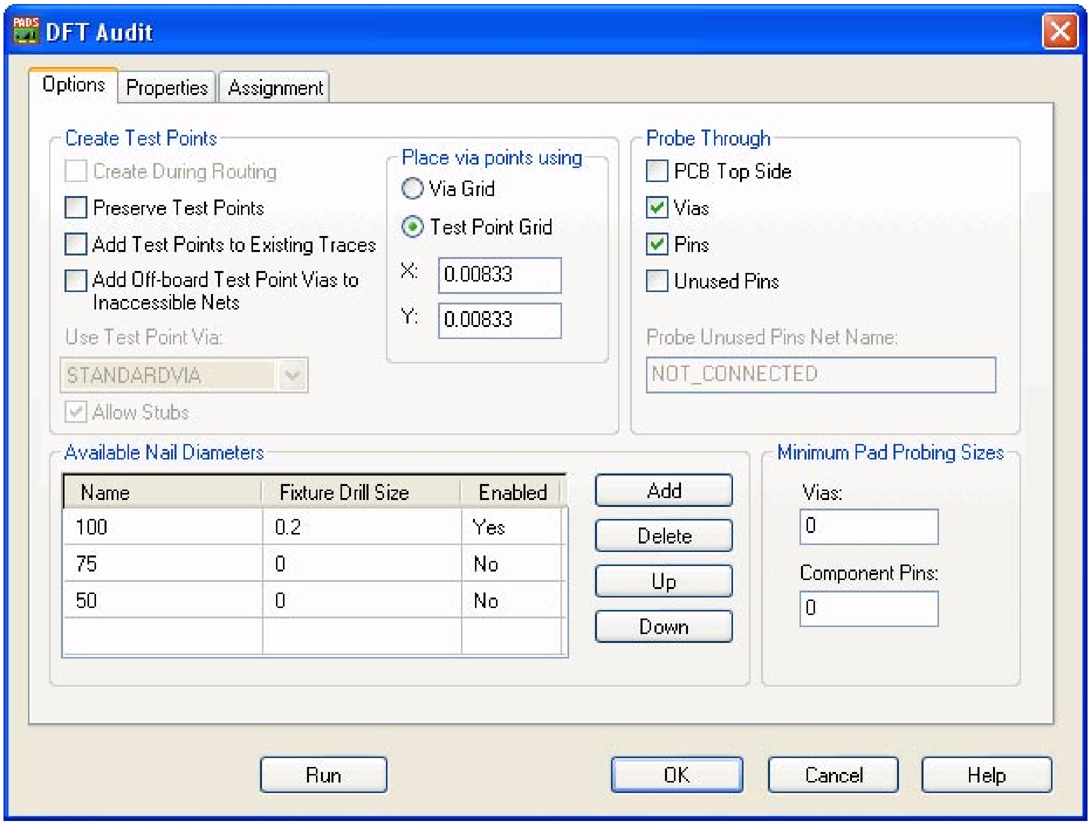


Table 45-117. Options Tab contents

Name	Description
Create During Routing	When using PADS Router Link, select the check box to enable PADS Router to create test points when routing nets. When using PADS Layout, this check box has no function.
Preserve Test Points	Prevents existing vias and component pins assigned as test points from being reassigned, removed, deleted, shoved, or modified.
Add Test Points to Existing Traces	Adds test point vias to inaccessible nets that are already routed. The design is automatically passed to PADS Router where it places test points and may, for example, plow other traces out of the way to make room for a new test point. Specify the type of via to insert from the Use Test Point Via list. You can also specify whether or not to allow short trace stubs when making nets accessible.

Table 45-117. Options Tab contents (cont.)

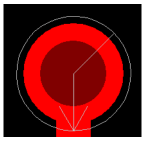
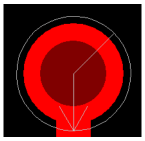
Name	Description
Add Off-board Test Points to Inaccessible Nets	Adds test point vias to nets that PADS Router cannot make accessible. PADS Layout places these test point vias outside the board outline and you can manually place them on the board. Specify the type of via to insert from the Use Test Point Via list.
Use Test Point Via list	Specifies the via type to use for test points. This list is enabled when you select any of the following check boxes: • Add Test Points to Existing Traces • Add Off-board Test Points to Inaccessible Nets This setting also determines the via type used by the Add Test Point command and End Test Point command.
Allow Stubs	Specifies to allow stubs.
Place Via Points Using area	Specifies the grid type to use for via test point placement: Via Grid or Test Point Grid.
PCB Top Side	Specifies to probe the board from the top and bottom sides of the PCB. **Tip:** Pins are always available for probing from the bottom side, whether or not the PCB Top Side check box is selected. If you want to probe **only** the top side, see Probing the PCB Top Side Only.
Vias	Specifies to use vias as test points. **Tip:** When the via is flagged as a test point, and Show Test Points is checked on the Routing/ General page of the Options dialog box, an arrow is drawn on it in the design: 
Pins	Specifies to use pins as test points. **Tip:** When the pin is flagged as a test point, and Show Test Points is checked on the Routing/ General page of the Options dialog box, an arrow is drawn on it in the design: 

Table 45-117. Options Tab contents (cont.)

Name	Description
Unused Pins	Specifies to provide access to unused pins during automated testing, and then type the net name used for all the unused pins in the design. If the unused pin is an SMD pad, DFT Audit attaches a test point via by adding one single pin net to each unused SMD pin. If the unused pin is a through hole component pin, DFT Audit assigns the unused pin as test point.
Available Nail Diameters table	The nail diameters are listed in order of preference, where the first entry is the most preferred. You can only use a maximum of fifteen integer characters in the nail diameter name. • **Name**—Specifies the probe size. The name is used to identify unique probe types. During the placement or assignment of test points, DFT Audit assigns values you specify in the Name cell as via or pin attributes. When you change these values, attributes for all vias and pins with these values are updated. • **Fixture Drill Size**—The diameter of the drilled hole in the fixture. **Recommendation:** Fixture drill size is used to calculate all rules relevant to the probe size, and its diameter should usually be slightly larger than the probe size. • **Enabled**—Specifies whether or not to use the associated nail during automated testing.
Add button	Adds a row to the bottom of the table.
Delete button	Removes the selected row from the table.
Up button	Moves the selected row up one.
Down button	Moves the selected row down one.
Maximum Pad Probing Sizes	Set minimum pad probing sizes for both vias and component pins to ensure that there is sufficient pad area for probe contact.
Run button	Starts the automatic audit process.

Related Topics

Placing Test Points

DFT Audit Dialog Box, Properties Tab

Several test point properties are available to you on the Properties tab of the DFT Audit dialog box.

Tip: To reduce design iterations, consult with your automated test engineers when setting DFT Audit options.

Accessing

- **Tools menu > DFT Audit > Properties tab**

Figure 45-131. Properties Tab

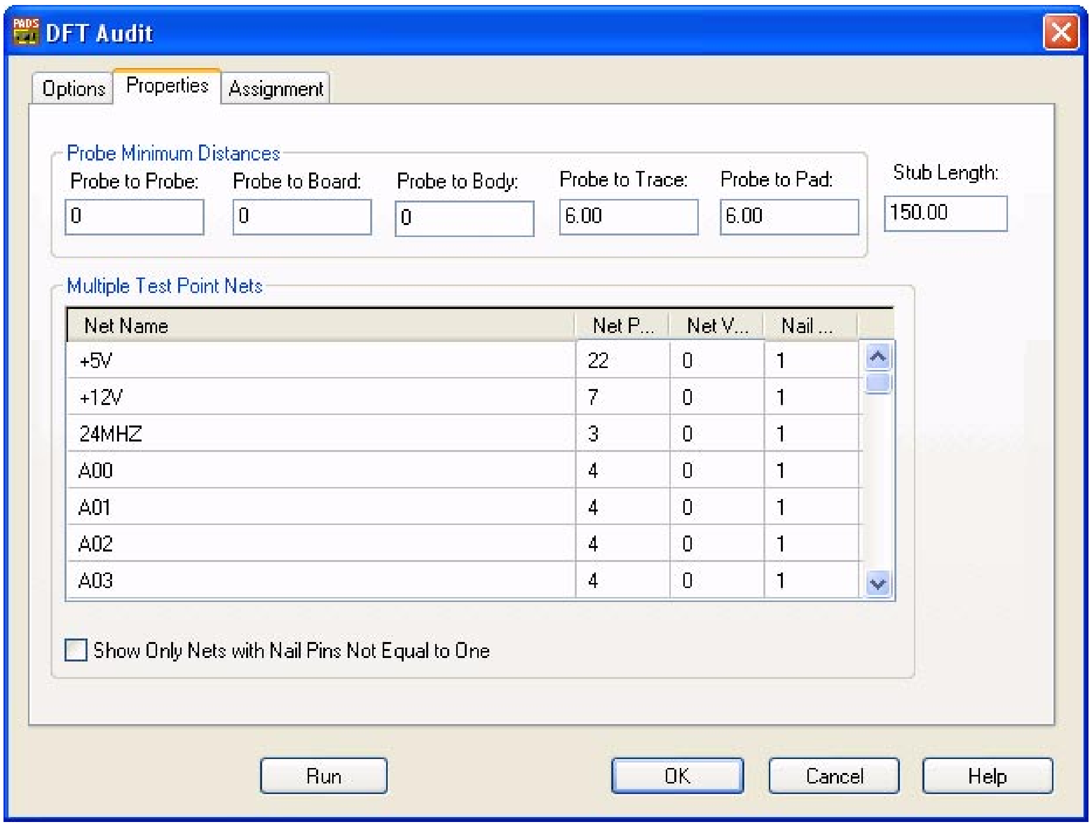


Table 45-118. Properties Tab contents

Name	Description
Probe Minimum Distances area	Specifies the minimum distances between the probe and other design objects using options in the Probe Minimum Distances area. **Tip:** The clearances needed between a probe and another design object are mostly based on the physical constraints of the Automated Test Equipment (ATE) used by In Circuit Testing (ICT) procedures. The probes extending out of the ATE fixture must make contact with the PCB without any obstacle. This means that test points must keep a fixed distance from component bodies, pads, mounting holes, and board edge, and must also have the minimum spacing between them.
Stub Length	Specifies the maximum length of trace stubs required to make a net accessible to a test probe.
Multiple Test Point Nets table	Displays the Net name, net pins, and net vias. Specifies the nail pins. **Tips:** • If you do not want any nail pins on a net, double-click the Nail Pins cell for the net and type zero (0). • To sort the list by a different column, click the column header at the top of the list.
Show Only Nets With Pins Not Equal to One	Specifies to display only nets with no nail pin or with more than one nail pin in the table.
Run button	Starts the automatic audit process.

Related Topics

Setting Test Point Properties

Die Flag Wizard Dialog Box

Use the Die Flag Wizard dialog box to:

- Define the number of rings to create.
- Make selections that define the layers, shapes, dimensions, net connections, and other properties of the die flag and rings that you are creating.

Restriction: This information applies only to the BGA toolkit.

Accessing

- **BGA Toolbar** button > **Die Flag Wizard button** > **select die**

Figure 45-132. Die Flag Wizard Dialog Box

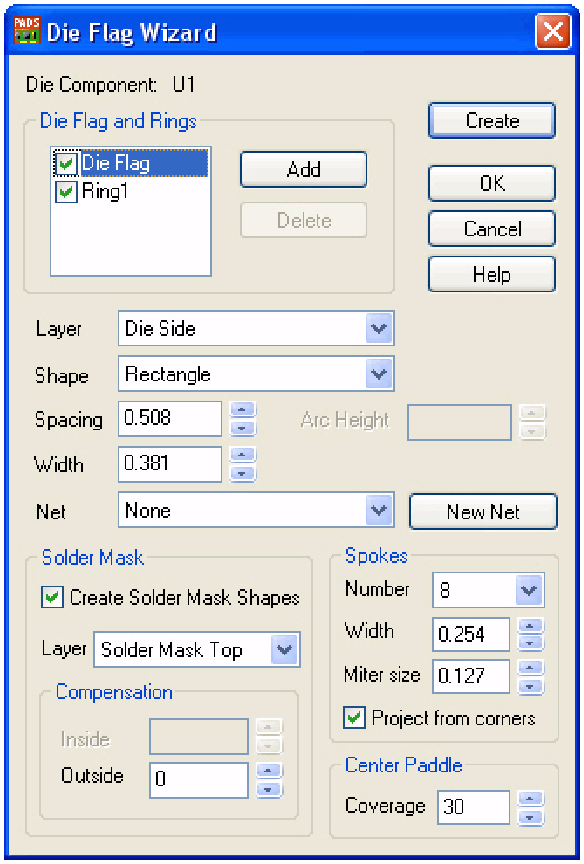


Table 45-119. Die Flag Wizard Dialog Box contents

Name	Description
Die Flag and Rings list	Lists all currently defined die flag shapes. Shapes have the predefined names **Die Flag**, **Ring 1**, **Ring 2**, and so on. These names appear only in the dialog box; they are not the names of the design copper shapes. To view or modify settings for a shape, either the die flag or a ring, select the shape. Settings that you change in the dialog box apply only to the currently selected shape. To create the selected shape, click the check box. If the shape's check box is cleared, Die Flag Wizard does not create that shape. It does, however, account for the shape's spacing while creating other shapes.

Table 45-119. Die Flag Wizard Dialog Box contents (cont.)

Name	Description
Add button	Inserts a new ring around the selected shape (die flag or ring). The settings for this new ring are the same as for the selected shape. There is no limit to the number of rings you can add; however, only one die flag is allowed.
Delete button	Deletes the currently selected ring from the list of shapes that Die Flag Wizard will create. You cannot delete the die flag.
Layer list	Lists all layers in the design. Specify a layer for the currently selected shape.
Shape list	Specifies the shape of the die flag or ring: • Rectangle • Rounded rectangle • Chamfered rectangle • Arced
Spacing	Specifies the distance between the inner edge of the die flag ring and the die outline. For other rings, this selection specifies the distance between the inner edge of the selected ring and the outer edge of the ring contained within it. The value for the spacing can be positive, negative, or zero.
Width	Assigns a width, in current design units, for the selected ring (including the die flag ring). Enter a width greater than zero.
Arc Height	Assigns a height for the arced shape. Enter a value greater than or equal to zero.
Net list	Lists net names. Select a net name to assign to the currently selected shape.
New Net button	Opens the Define Name of New Net Dialog Box, where you can add the name of a new net to the list of available nets.
Create Solder Mask Shapes	Creates a solder mask shape for the selected shape if the box is checked.
Solder Mask Layer	Selects a layer upon which to create the solder mask. The Layer list contains the names of the design layers associated with a solder mask.
Inside Compensation	Defines the size of the overlap with or retraction of the solder mask shape from the inside of the conductive shape. To make the solder mask overlap the inside of the conductive shape, enter a positive value. To make the solder mask retract from the inside of the conductive shape, enter a negative value.

Table 45-119. Die Flag Wizard Dialog Box contents (cont.)

Name	Description
Outside Compensation	Defines the size of the overlap with or retraction of the solder mask shape from the outside of the conductive shape. To make the solder mask overlap the outside of the conductive shape, enter a positive value. To make the solder mask retract from the outside of the conductive shape, enter a negative value.
Number	Lists the number of spokes for the die flag. Available selections are: **4**, **8**, **12**, and **16**.
Width	Select the spoke width for the die flag.
Miter Size	Select the size of the straight-line segments to miter acute angles where the die flag spokes connect to the rings or paddles. The value must be greater than zero. It cannot exceed the distance between the outer edge of the center paddle and the inner edge of the die flag ring. **Tip:** To avoid acid traps, acute angles are not allowed.
Project from Corners	Select whether to project the spoke configuration from the corner. When **Project from corners** is cleared, the spokes are offset from the corner.
Coverage	Defines the size of the die flag's center paddle. It is the ratio of the area of the center paddle to the area of the entire die flag. The entire die flag encompasses the center paddle, spokes, and die flag ring. Select a value between zero and 100. A value of zero means that the center paddle is not generated. A value of 100 means that the die flag is generated as one, solid shape without a defined center paddle, spoke, or ring.
Create button	Creates all shapes that are checked in the list box, saves their settings, and closes the Die Flag Wizard dialog box.

Related Topics

To Create a Die Flag and Rings

Die Flag Wizard

Die Wizard - Create from GDSII File Dialog Box

The Die Wizard – Create from GDSII File dialog box adds a new die to the current design or library.

Restriction: This information applies only to the BGA toolkit.

Use the dialog box to:

- Define a die outline from a GDSII file
- Define a set of CBPs from a GDSII file
- Define the numbering of CBPs
- Define the functions for pads from a GDSII file
- Define preferences for die component creation

The Die Wizard has 5 tabs:

- Die Size
- CBP
- Pad #
- Pad Functions
- Die Prefs

Accessing

- **BGA Toolbar** button > **Die Wizard button** > **From GDSII File** button

Figure 45-133. Die Wizard - Create from GDSII File Dialog Box

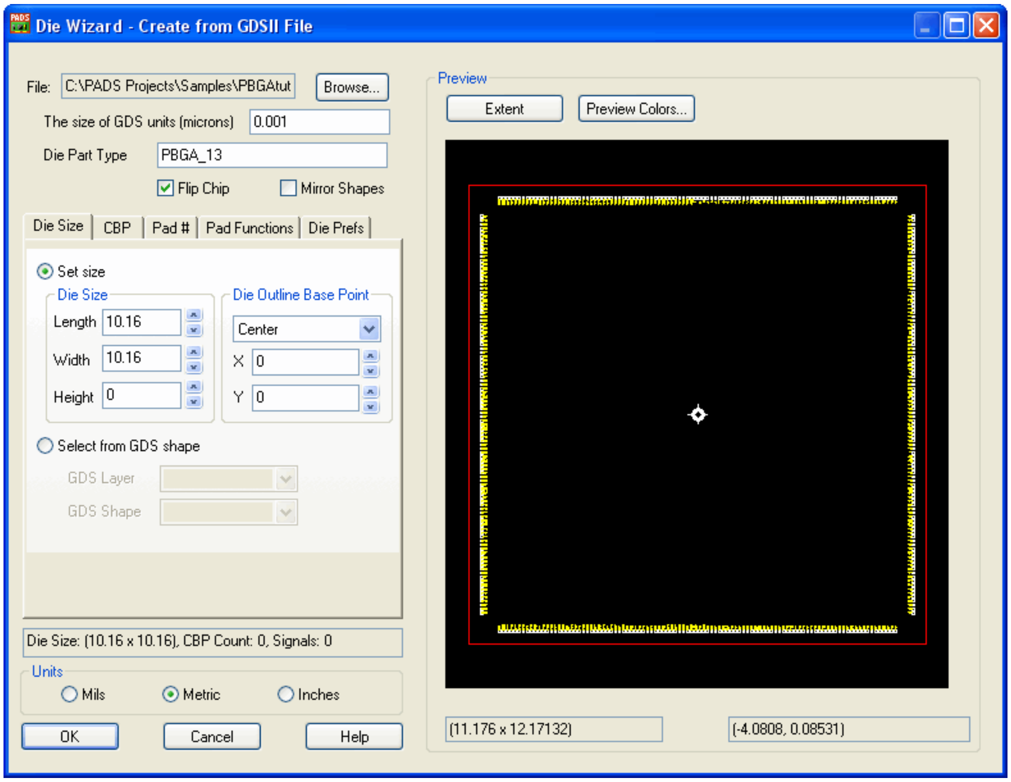


Table 45-120. Die Wizard - Create from GDSII File Dialog Box contents

Name	Description
File	Displays the name of the text file you want to use to create the die. **Tip:** Click Browse to open the Open dialog box and chose the file you want.
The Size of GDS Units (microns)	Displays the size of the GDS file units in microns. Modify the value if necessary.
Die Part Type	The name of the type of die part you want to use to create the die.
Flip Chip	Selects the IC die for facedown mounting. Flip chip components contain only pins (SBPs).
Mirror Shapes	Mirrors, or flips, the shapes (die geometrical data) on the die display.

Table 45-120. Die Wizard - Create from GDSII File Dialog Box contents (cont.)

Name	Description
tabs	• Die Size • CBP • Pad # • Pad Functions • Die Prefs
Die Data Status area	Displays the die size, number of chip bond pads, and number of signals.
Units area	Sets the global unit type to use to convert system units into one of these commonly used sets of measurements: • **Mils—**Expressed in mils (1mil = 2.54*10^-5 m). • **Metric—**Expressed in millimeters (1mm = 1.0*10^-3 m). • **Inches—**Expressed in inches (1" = 2.54*10^-2 m). All values are expressed on the die display in the units you choose. **Tip:** You cannot choose **Microns** to use as system units. Use **Metric** if the values in the imported file are expressed in microns.
Extent button	Displays the design in the largest x and y area that is occupied by all items within the die definition.
Preview Colors button	Opens the Preview Colors dialog box so you can select colors that help you preview your design.
Show All	When **Show All** is selected, the die display area contains both the shapes that represent the die items, such as the die outline and CBPs, and the rest of the shapes from the GDSII file. When clear, the die display contains only the shapes selected as die items.
Preview window	Displays the die design. Position the cursor and click to zoom in or right-click to zoom out.
Size of Preview Window display	Displays the size of the die display in the current unit measurement.
Cursor Position display	Displays the position of the cursor in the die display.

Figure 45-134. Die Wizard - Create from GDSII File, Die Size tab

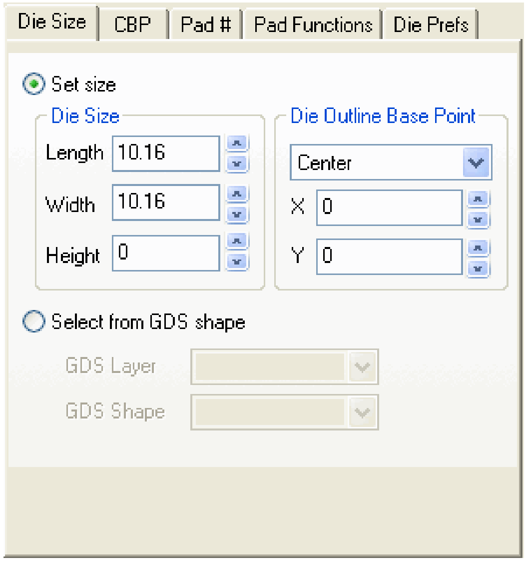


Table 45-121. Die Size tab contents

Name	Description
Die Size Tab	• **Set Size**—Select to set the die size and die outline position manually. • **Select from GDS Shape**—Select to choose a GDS shape for the die outline. You can modify the Height control when you click **Select from GDS Shape**, but you cannot modify the other parameters in the **Set Size** area.
Length	Type or select the length of the die, which corresponds to the X size.
Width	Type or select the width of the die, which corresponds to the Y size.
Height	Type or select the height of the die, which defines the thickness of the die.
Die Point list	(center, lower left, upper left, upper right, or lower right) for which you want to specify the coordinates for the base point, as expressed in X and Y.
X	Type or select the die point along the x-axis you want as the base point for X.
Y	Type or select the die point along the y-axis you want as the base point for Y.
GDS Layer	Shows the layers defined in the GDSII file. Use to select the layer on which the die outline shape is located.

Table 45-121. Die Size tab contents (cont.)

Name	Description
GDS Shape	Shows the names of all the GDS shapes available for selection as the die outline shape. Only closed, filled shapes from the GDSII file that are also on the selected layer in the GDS Layer list appear. The shapes appear in the die display area using the Shapes on Selected Layers color.

Figure 45-135. Die Wizard - Create from GDSII File, CBP tab

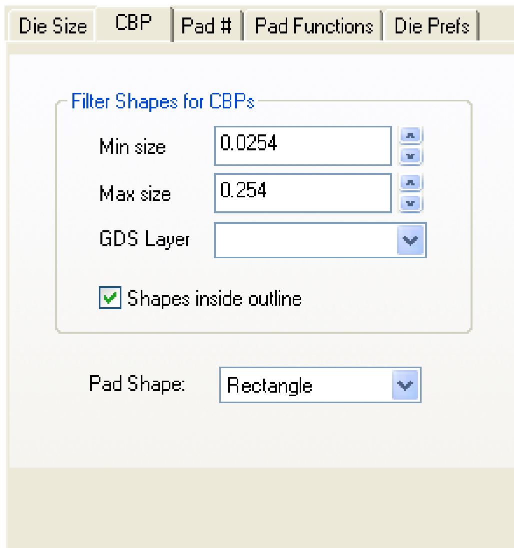


Table 45-122. CBP contents

Name	Description
Min Size	Sets the minimum size you want to use to filter GDS shapes for CBPs, expressed in the current system units.
Max Size	Sets the maximum size to use to filter GDS shapes for CBPs.
GDS Layer	Use to select the layer on which the GDS shape for CBPs is located.
Shapes Inside Outline	Includes only the shapes found inside the die outline in the filtering.
Pad Shape	Defines the shape of the pads: rectangle or oval. • **Rectangle**—CBP shapes are derived from GDS shapes as circumscribed rectangles. • **Oval**—CBP shapes are derived from GDS shapes as circumscribed circles.

Figure 45-136. Die Wizard - Create from GDSII File, Pad # tab

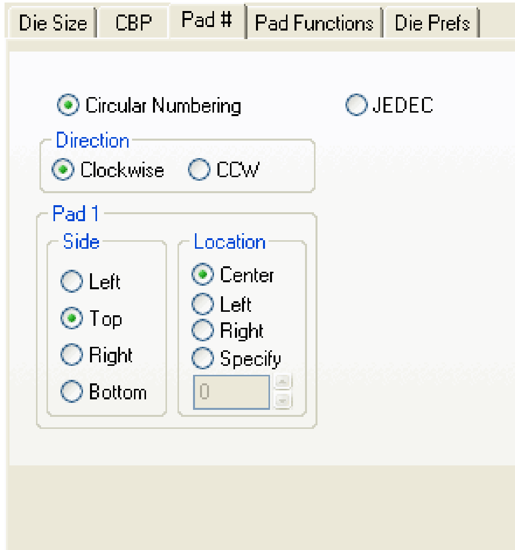


Table 45-123. Pad # tab contents

Name	Description
Numbering Mode	• **Circular—**Circular numbering. • **JEDEC—**JEDEC numbering. Pin rows are lettered from top to bottom starting with A and pin columns are numbered from left to right starting with 1. The letters I, O, Q, S, X, and Z are not used. For arrays with more than 20 rows, row 21 is designated AA and subsequent rows are designated AB, AC, etc.
Direction area	Specifies the numbering direction you want to use: **Clockwise—**Numbering begins with pad 1 and continues in a clockwise direction. **CCW—**Numbering begins with pad 1 and continues in a counterclockwise (CCW) direction.
Pad 1 Side area	Specifies the side of the design to use for the position of pad 1: **Left—**On the left side of the design. **Top—**On the top of the design. **Right—On** the right side of the design. **Bottom—O**n the bottom of the design.

Table 45-123. Pad # tab contents (cont.)

Name	Description
Pad 1 Location area	Specifies the location on the side of the design to use for the position of pad 1: **Center**—Numbers the center pad of the specified side of the design as pad 1. **Left**—Numbers the leftmost pad of the specified side of the design as pad 1. **Right**—Numbers the rightmost pad of the specified side of the design as pad 1. **Specify**—Numbers the pad you select on the specified side of the design as pad 1. Type or select the pad number you want to use for pad 1.

Figure 45-137. Die Wizard - Create from GDSII File, Pad Functions tab

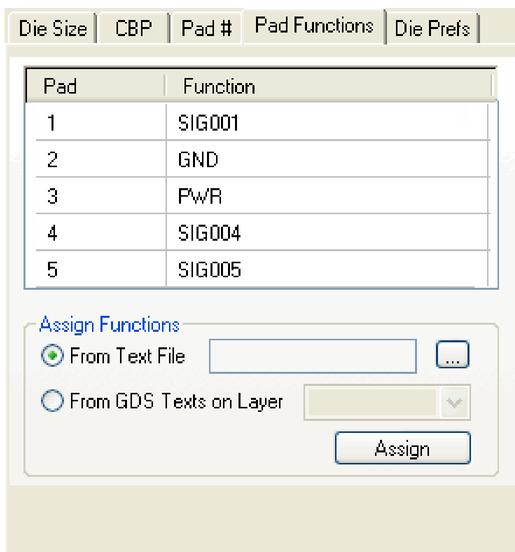


Table 45-124. Pad Functions tab contents

Name	Description
Pad column	Shows each component bond pad number. Click the **Pad** column header to sort by pad number in either ascending or descending order.
Function column	Shows all function names that correspond to component bond pad numbers. Click the **Function** column header to sort by function name in either ascending or descending order. Double-click a function to change a function name.

Table 45-124. Pad Functions tab contents (cont.)

Name	Description
Assign Functions area	Specifies how you want to use to assign functions: • **From Text File—**After selecting the option, click **Browse** to select the file you want to use to assign pad functions. • **From GDS Texts on Layer—**Select the GDS layer you want to use to assign pad functions.
Assign button	Assigns the new function names to all pads.

Figure 45-138. Die Wizard - Create from GDSII File, Die Prefs tab

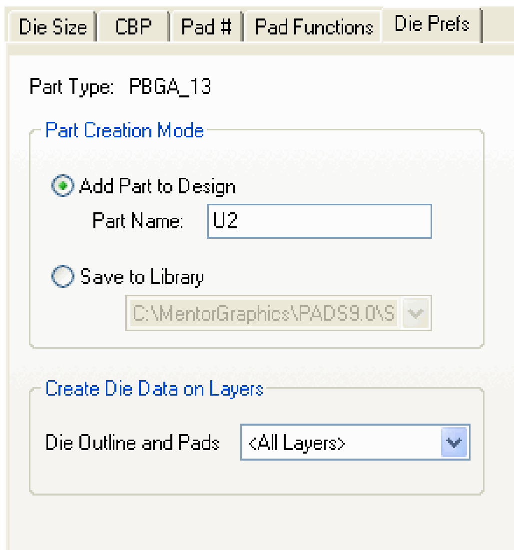


Table 45-125. Die Prefs tab contents

Name	Description
Part Type	Identifies the die part type to add to the design.
Part Creation Mode area	Sets the layer on which the die outline and pads appear. Select a layer from the list. • **Add Part to Design—**Adds the part to the currently open design. Indicate the reference designator that is automatically assigned to the new die component in the design in the Part Name box. To change the reference designator, click on the part name and enter a new name. • **Save to Library—**Saves the part in a specified library. Select the library in which to save the part.
Die Outline and Pads	• Sets the layer on which the die outline and pads appear. Select a layer from the list.

Related Topics

To Create a Die from a GDSII File

Die Wizard - Create from Text File Dialog Box

The Die Wizard – Create from Text File dialog box adds a new die to the current design or library.

Restriction: This information applies only to the BGA toolkit.

Use the dialog box to:

- Define a die outline.
- Modify the shapes of CBPs from a text file.
- Define the numbering of CBPs from a text file.
- Modify the functions for pads from a text file.
- Define preferences for die component creation.

The Die Wizard has 5 tabs:

- Die Size
- CBP
- Pad #
- Pad Functions
- Die Prefs

Accessing

- **BGA Toolbar** button > **Die Wizard button** > **From Text File** button

Figure 45-139. Die Wizard - Create from Text File Dialog Box

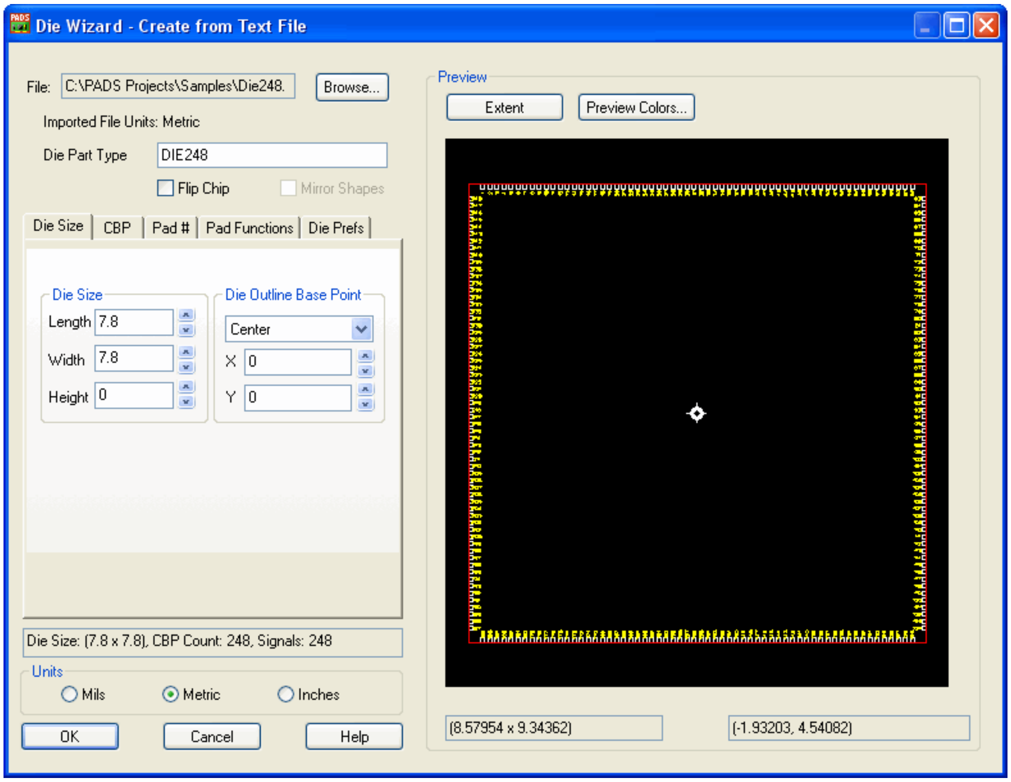


Table 45-126. Die Wizard - Create from Text File Dialog Box contents

Name	Description
File	Displays the name of the text file you want to use to create the die. **Tip:** Click Browse to open the Open dialog box and chose the file you want.
Imported File Units (Metric)	Displays the size of the imported file units in microns. Modify the value if necessary.
Die Part Type	The name of the type of die part you want to use to create the die.
Flip Chip	Selects the IC die for facedown mounting. Flip chip components contain only pins (SBPs).
Mirror Shapes	Mirrors, or flips, the shapes (die geometrical data) on the die display.

Table 45-126. Die Wizard - Create from Text File Dialog Box contents (cont.)

Name	Description
tabs	• Die Size • CBP • Pad # • Pad Functions • Die Prefs
Die Data Status area	Displays the die size, number of chip bond pads, and number of signals.
Units area	Sets the global unit type to use to convert system units into one of these commonly used sets of measurements: • **Mils—**Expressed in mils (1mil = 2.54*10^-5 m). • **Metric—**Expressed in millimeters (1mm = 1.0*10^-3 m). • **Inches—**Expressed in inches (1" = 2.54*10^-2 m). All values are expressed on the die display in the units you choose. **Tip:** You cannot choose **Microns** to use as system units. Use **Metric** if the values in the imported file are expressed in microns.
Extent button	Displays the design in the largest x and y area that is occupied by all items within the die definition.
Preview Colors button	Opens the Preview Colors dialog box so you can select colors that help you preview your design.
Preview window	Displays the die design. Position the cursor and click to zoom in or right-click to zoom out.
Size of Preview Window display	Displays the size of the die display in the current unit measurement.
Cursor Position display	Displays the position of the cursor in the die display.

Figure 45-140. Die Wizard - Create from Text File, Die Size tab

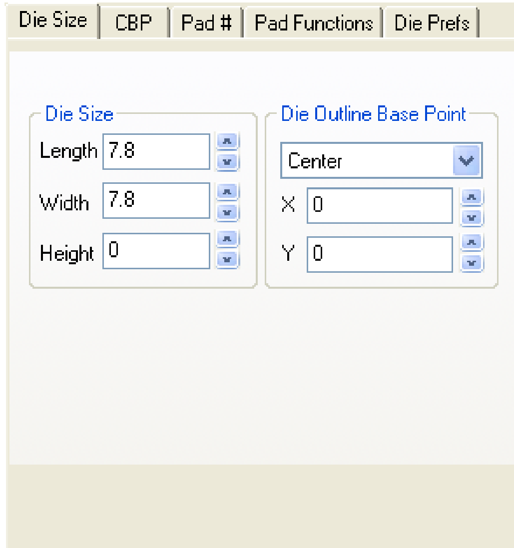


Table 45-127. Die Size tab contents

Name	Description
Length	Type or select the length of the die, which corresponds to the X size.
Width	Type or select the width of the die, which corresponds to the Y size.
Height	Type or select the height of the die, which defines the thickness of the die.
Die Outline Base Point list	Center, lower left, upper left, upper right, or lower right - for which you want to specify the coordinates for the base point, as expressed in X and Y.
X	Type or select the die point along the x-axis you want as the base point for X.
Y	Type or select the die point along the y-axis you want as the base point for Y.

Figure 45-141. Die Wizard - Create from Text File, CBP tab

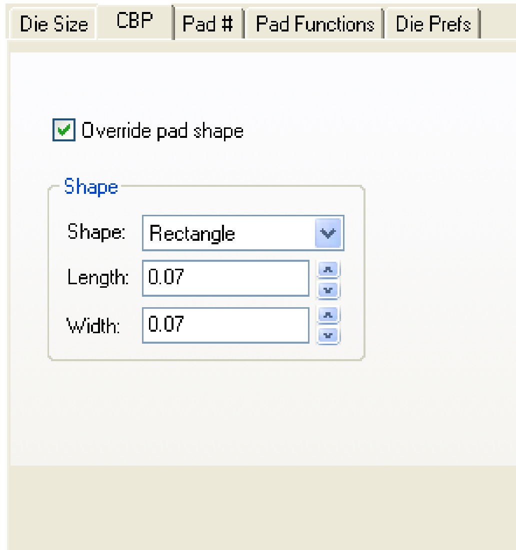


Table 45-128. CBP tab contents

Name	Description
Override pad shape	Overrides the values from the file using the values you enter in **Shape, Length, and Width** to define the shape for all pads.
Shape	Defines the shape of the pads: rectangle or oval.
Length	**Specifies** the value for the pad length.
Width	**Specifies** the value for the pad width.

Figure 45-142. Die Wizard - Create from Text File, Pad # tab

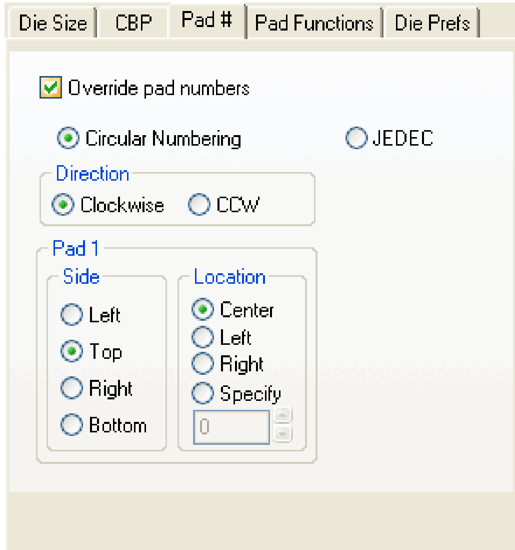


Table 45-129. Pad # tab contents

Name	Description
Override Pad numbers	Overrides the values from the file using the values you enter to define the pad numbering.
Numbering Mode	• **Circular—**Circular numbering. • **JEDEC—**JEDEC numbering. Pin rows are lettered from top to bottom starting with A and pin columns are numbered from left to right starting with 1. The letters I, O, Q, S, X, and Z are not used. For arrays with more than 20 rows, row 21 is designated AA and subsequent rows are designated AB, AC, etc.
Direction area	Specifies the numbering direction you want to use: **Clockwise—**Numbering begins with pad 1 and continues in a clockwise direction. **CCW—**Numbering begins with pad 1 and continues in a counterclockwise (CCW) direction.
Pad 1 Side area	Specifies the side of the design to use for the position of pad 1: **Left—**On the left side of the design. **Top—**On the top of the design. **Right—On** the right side of the design. **Bottom—O**n the bottom of the design.

Table 45-129. Pad # tab contents (cont.)

Name	Description
Pad 1 Location area	Specifies the location on the side of the design to use for the position of pad 1: **Center—**Numbers the center pad of the specified side of the design as pad 1. **Left—**Numbers the leftmost pad of the specified side of the design as pad 1. **Right—**Numbers the rightmost pad of the specified side of the design as pad 1. **Specify—**Numbers the pad you select on the specified side of the design as pad 1. Type or select the pad number you want to use for pad 1.

Figure 45-143. Die Wizard - Create from Text File, Pad Functions tab

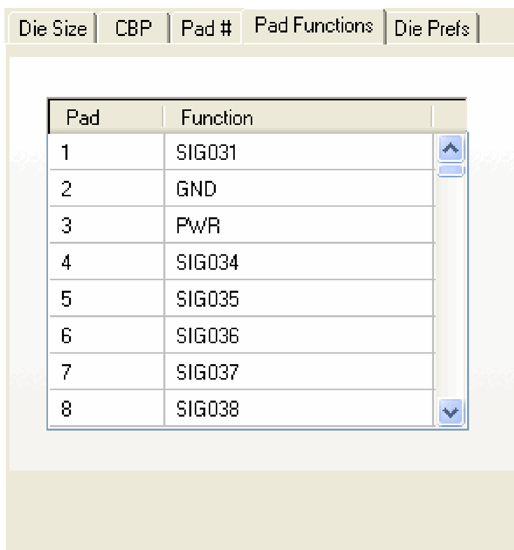


Table 45-130. Pad Functions tab contents

Name	Description
Pad column	Shows each component bond pad number. Click the **Pad** column header to sort by pad number in either ascending or descending order.
Function column	Shows all function names that correspond to component bond pad numbers. Click the **Function** column header to sort by function name in either ascending or descending order. Double-click a function to change a function name.

Figure 45-144. Die Wizard - Create from Text File, Die Prefs tab

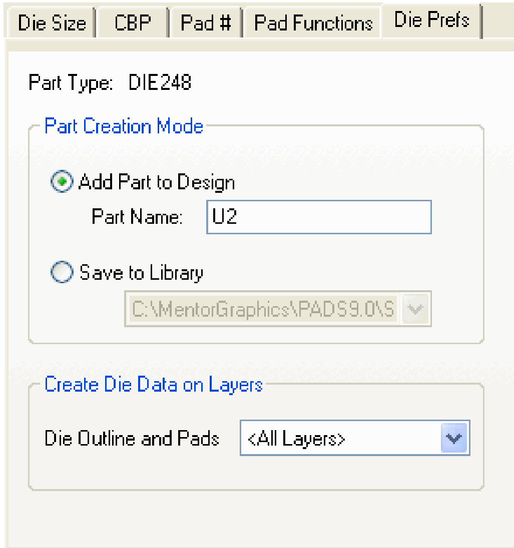


Table 45-131. Die Prefs tab contents

Name	Description
Part Type	Identifies the die part type to add to the design.
Part Creation Mode area	Sets the layer on which the die outline and pads appear. Select a layer from the list. • **Add Part to Design—**Adds the part to the currently open design. Indicate the reference designator that is automatically assigned to the new die component in the design in the Part Name box. To change the reference designator, click on the part name and enter a new name. • **Save to Library—**Saves the part in a specified library. Select the library in which to save the part.
Die Outline and Pads	Specifies the layer on which you want to create the die data.

Related Topics

To Create a Die from a Text File

Die Wizard - Create Parametrically Dialog Box

The Die Wizard – Create Parametrically dialog box adds a new die to the current design or library.

Restriction: This information applies only to the BGA toolkit.

Use the dialog box to:

- Define a die outline.
- Define a set of CBPs.
- Define the numbering of CBPs.
- Define the functions for pads.
- Define preferences for die component creation.
- Add new die to the current design or to the part library.

On the CBP tab, there are two ways to set up pad counts:

- Set up the total pad count for a die with automatic distribution of pad counts evenly along the sides, using **Total**, **GND %**, and **PWR %**.
- Set up a specific pad count for each side of a die, using **Side**, **Total Pads**, **GND**, and **PWR**.
- Define preferences for die component creation

The Die Wizard has 5 tabs:

- Die Size
- CBP
- Pad #
- Pad Functions
- Die Prefs

Accessing

- **BGA Toolbar** button > **Die Wizard button** > **Parametrically** button

Figure 45-145. Die Wizard - Create Parametrically Dialog Box

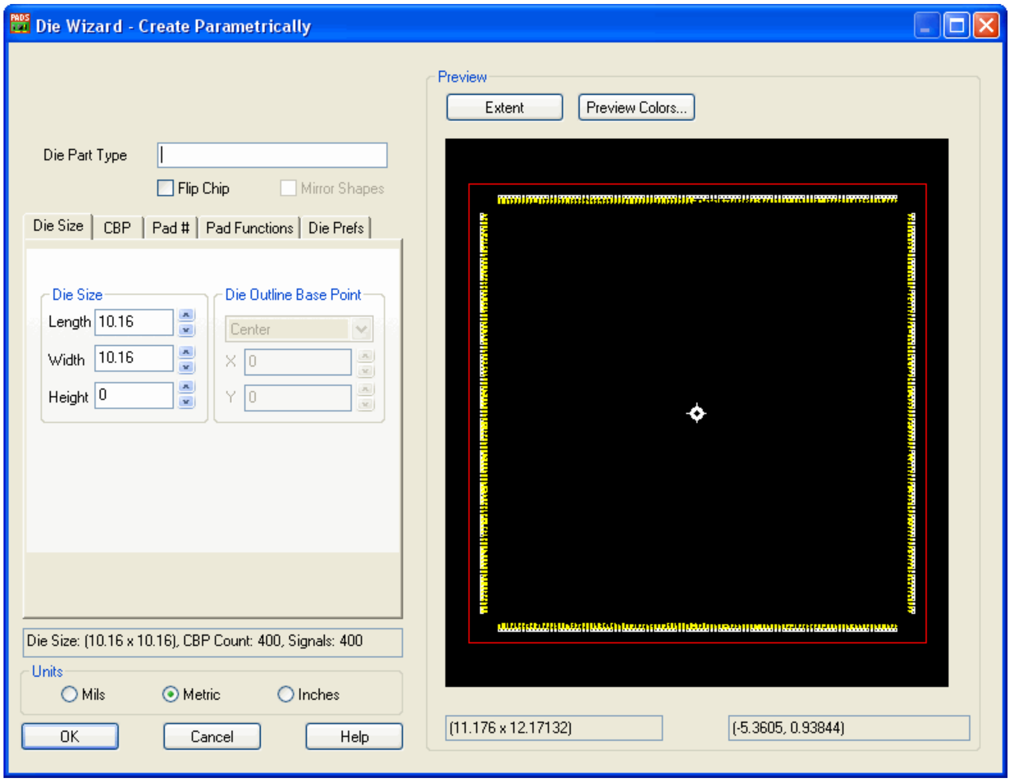


Table 45-132. Die Wizard - Create Parametrically Dialog Box contents

Name	Description
Die Part Type	The name of the type of die part you want to use to create the die.
Flip Chip	Selects the IC die for facedown mounting. Flip chip components contain only pins (SBPs).
Mirror Shapes	Mirrors, or flips, the shapes (die geometrical data) on the die display.
tabs	• Die Size • CBP • Pad # • Pad Functions • Die Prefs
Die Data Status area	Displays the die size, number of chip bond pads, and number of signals.

Table 45-132. Die Wizard - Create Parametrically Dialog Box contents (cont.)

Name	Description
Units area	Sets the global unit type to use to convert system units into one of these commonly used sets of measurements: • **Mils—**Expressed in mils (1mil = 2.54*10^-5 m). • **Metric—**Expressed in millimeters (1mm = 1.0*10^-3 m). • **Inches—**Expressed in inches (1" = 2.54*10^-2 m). All values are expressed on the die display in the units you choose. **Tip:** You cannot choose **Microns** to use as system units. Use **Metric** if the values in the imported file are expressed in microns.
Extent button	Displays the design in the largest x and y area that is occupied by all items within the die definition.
Preview Colors button	Opens the Preview Colors dialog box so you can select colors that help you preview your design.
Preview window	Displays the die design. Position the cursor and click to zoom in or right-click to zoom out.
Size of Preview Window display	Displays the size of the die display in the current unit measurement.
Cursor Position display	Displays the position of the cursor in the die display.

Figure 45-146. Die Wizard - Create Parametrically, Die Size tab

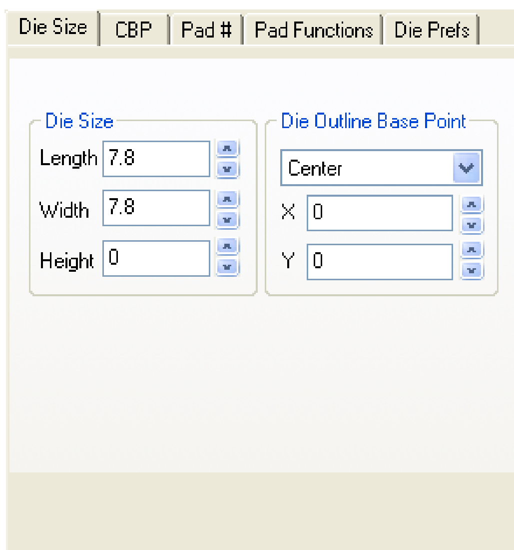


Table 45-133. Die Size tab contents

Name	Description
Length	Type or select the length of the die, which corresponds to the X size.
Width	Type or select the width of the die, which corresponds to the Y size.
Height	Type or select the height of the die, which defines the thickness of the die.
Die Point list	(center, lower left, upper left, upper right, or lower right) for which you want to specify the coordinates for the base point, as expressed in X and Y.
X	Type or select the die point along the x-axis you want as the base point for X.
Y	Type or select the die point along the y-axis you want as the base point for Y.

Figure 45-147. Die Wizard - Create Parametrically, CBP tab

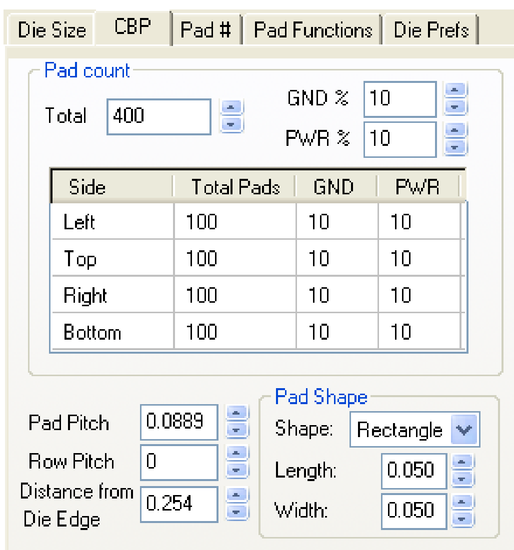


Table 45-134. CBP contents

Name	Description
Total	Type or select the total pad count for the die. If you modify the total pad count, the pads are evenly distributed along the four sides.

Table 45-134. CBP contents (cont.)

Name	Description
GND %	Type or select the percentage of the total pin count to allocate as ground pads. If you modify the value, the ground pads for each side are derived from the **Total** count.
PWR %	Type or select the percentage of the total pin count to allocate as power pads. If you modify the value, the power pads for each side are derived from the **Total** count.
Side column	Lists the sides of the die.
Total Pads column	View or double-click to change the total number of pads for each side of the die. If you modify a value, the values in **Total**, **GND %**, and **PWR %** adjust accordingly.
GND column	View or double-click to change the total number of ground pads for each side of the die. If you modify a value, the values in **Total**, **GND %**, and **PWR %** adjust accordingly.
PWR column	View or double-click to change the total number of power pads for each side of the die. If you modify a value, the values in **Total**, **GND %**, and **PWR %** adjust accordingly.
Pad Pitch	Defines the distance between pads. The distance is measured from the left side of one pad to the left side of the next adjacent pad.
Row Pitch	Defines the distance between rows to control creation of staggered row patterns. For a single row in a straight line around the sides of the die, enter **0**. To create staggered row patterns, enter a specific positive or negative number.
Distance from Die Edge	Defines the distance between the row of the pads and the die edge.
Pad Shape	Sets the shape of the thermal relief: **Round**, **Square**, **Rectangular**, or **Oval Pads**.
Pad Length	Type or select the value for the pad length.
Pad Width	Type or select the value for the pad width.

Figure 45-148. Die Wizard - Create Parametrically, Pad # tab

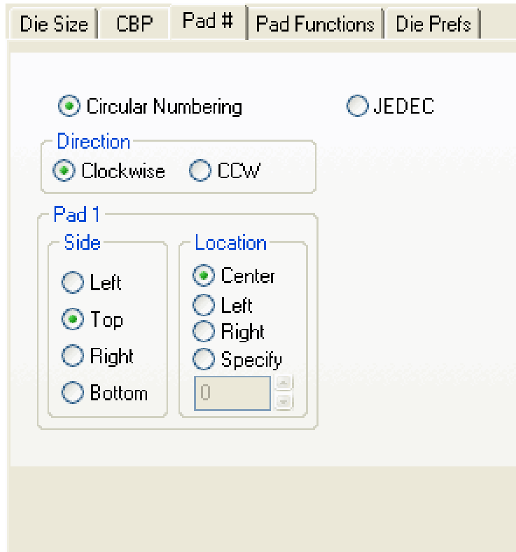


Table 45-135. Pad # tab contents

Name	Description
Numbering Mode	• **Circular—**Circular numbering. • **JEDEC—**JEDEC numbering. Pin rows are lettered from top to bottom starting with A and pin columns are numbered from left to right starting with 1. The letters I, O, Q, S, X, and Z are not used. For arrays with more than 20 rows, row 21 is designated AA and subsequent rows are designated AB, AC, etc.
Direction area	Specifies the numbering direction you want to use: **Clockwise—**Numbering begins with pad 1 and continues in a clockwise direction. **CCW—**Numbering begins with pad 1 and continues in a counterclockwise (CCW) direction.
Pad 1 Side area	Specifies the side of the design to use for the position of pad 1: **Left—**On the left side of the design. **Top—**On the top of the design. **Right—On** the right side of the design. **Bottom—O**n the bottom of the design.

Table 45-135. Pad # tab contents (cont.)

Name	Description
Pad 1 Location area	Specifies the location on the side of the design to use for the position of pad 1: **Center**—Numbers the center pad of the specified side of the design as pad 1. **Left**—Numbers the leftmost pad of the specified side of the design as pad 1. **Right**—Numbers the rightmost pad of the specified side of the design as pad 1. **Specify**—Numbers the pad you select on the specified side of the design as pad 1. Type or select the pad number you want to use for pad 1.

Figure 45-149. Die Wizard - Create Parametrically, Pad Functions tab

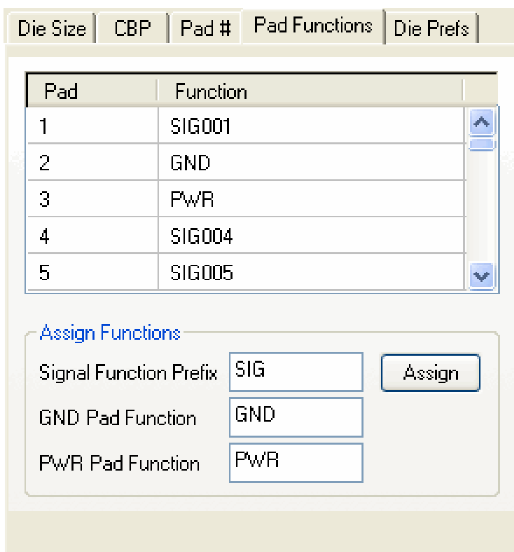


Table 45-136. Pad Functions tab contents

Name	Description
Pad column	Shows each component bond pad number. Click the **Pad** column header to sort by pad number in either ascending or descending order.
Function column	Shows all function names that correspond to component bond pad numbers. Click the **Function** column header to sort by function name in either ascending or descending order. Double-click a function to change a function name.

Table 45-136. Pad Functions tab contents (cont.)

Name	Description
Signal Function Prefix	Type or view the function prefix you want to use to derive the signal pad names.
GND Pad Function	Type or view the function name you want to use for the ground pads.
PWR Pad Function	Type or view the function name you want to use for the power pads.
Assign button	Assigns the new function names to all pads.

Figure 45-150. Die Wizard - Create Parametrically, Die Prefs tab

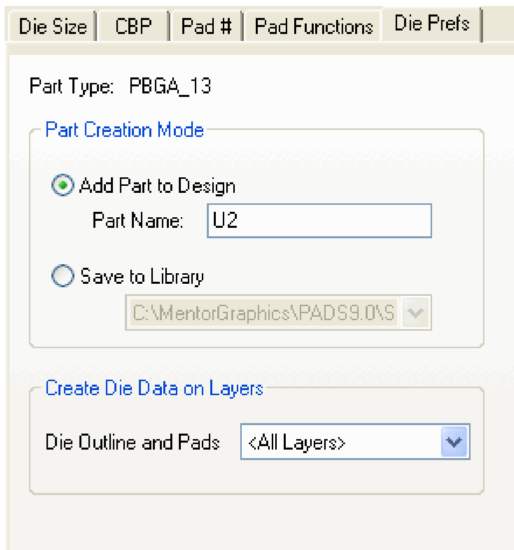


Table 45-137. Die Prefs tab contents

Name	Description
Part Type	Identifies the die part type to add to the design.
Part Creation Mode area	Sets the layer on which the die outline and pads appear. Select a layer from the list. • **Add Part to Design—**Adds the part to the currently open design. Indicate the reference designator that is automatically assigned to the new die component in the design in the Part Name box. To change the reference designator, click on the part name and enter a new name. • **Save to Library—**Saves the part in a specified library. Select the library in which to save the part.

Table 45-137. Die Prefs tab contents (cont.)

Name	Description
Die Outline and Pads	Specifies the layer on which you want to create the die data.

Related Topics

To Create a Die Parametrically

Die Wizard Preview Colors Dialog Box

Use the Die Wizard Preview Colors dialog box to select colors for previewing your die design.

Restriction: This information applies only to the BGA toolkit.

Setting colors on this dialog box does not change the colors you set in the Display Colors Setup dialog box.

Accessing

- **BGA Toolbar** button > **Die Wizard button** > **any** button > **Preview Colors** button

Figure 45-151. Die Wizard Preview Colors Dialog Box

Table 45-138. Die Wizard Preview Colors Dialog Box contents

Name	Description
Selected Color area	Select the color you want to apply to one or more items on the dialog box.
Background	Sets the background color in the die display area.
Highlight	Sets the highlight color in the die display area.
Die Outline	Sets the color of the die outline in the die display area.
CBP	Sets the color of the CBPs in the die display area.
CBP #	Sets the color of the CBP numbers in the die display area.
All Shapes	Sets the color of all GDS shapes in the die display area. The GDSII shapes area of the dialog box is active only when you are creating a die from a GDSII file.
Shapes on Selected Layers	Sets the color of the GDS shapes in the GDSII file that appear on the selected GSD Layer, in the die display area. The color appears only when the Die Size tab or the Pad Functions tab is active. The GDSII shapes area of the dialog box is active only when you are creating a die from a GDSII file.

Related Topics

To Set Die Preview Colors

Differential Pairs Dialog Box

Use the Differential Pairs dialog box to identify nets, associated nets, or pin pairs that behave electrically as differential pairs, and to define differential pair design rules. You can set different properties for differential pairs, which affects how they are routed. Differential pair properties determine the gap between the traces in the controlled gap area, the minimum and maximum trace lengths and widths, and how to respond to obstacles in the controlled gap area.

Restriction: You can define Differential Pairs Rules in PADS Layout, but these rules are used in PADS Router only. Layout does, however, respect the diff pair gap as a trace to trace clearance.

Accessing

- **Setup** menu > **Design Rules** > **Differential Pairs**

Tip: You can set Differential Pairs rules for Nets, Pin pairs, or Associated Nets, depending on the tab you select.

Figure 45-152. Differential Pairs Dialog Box

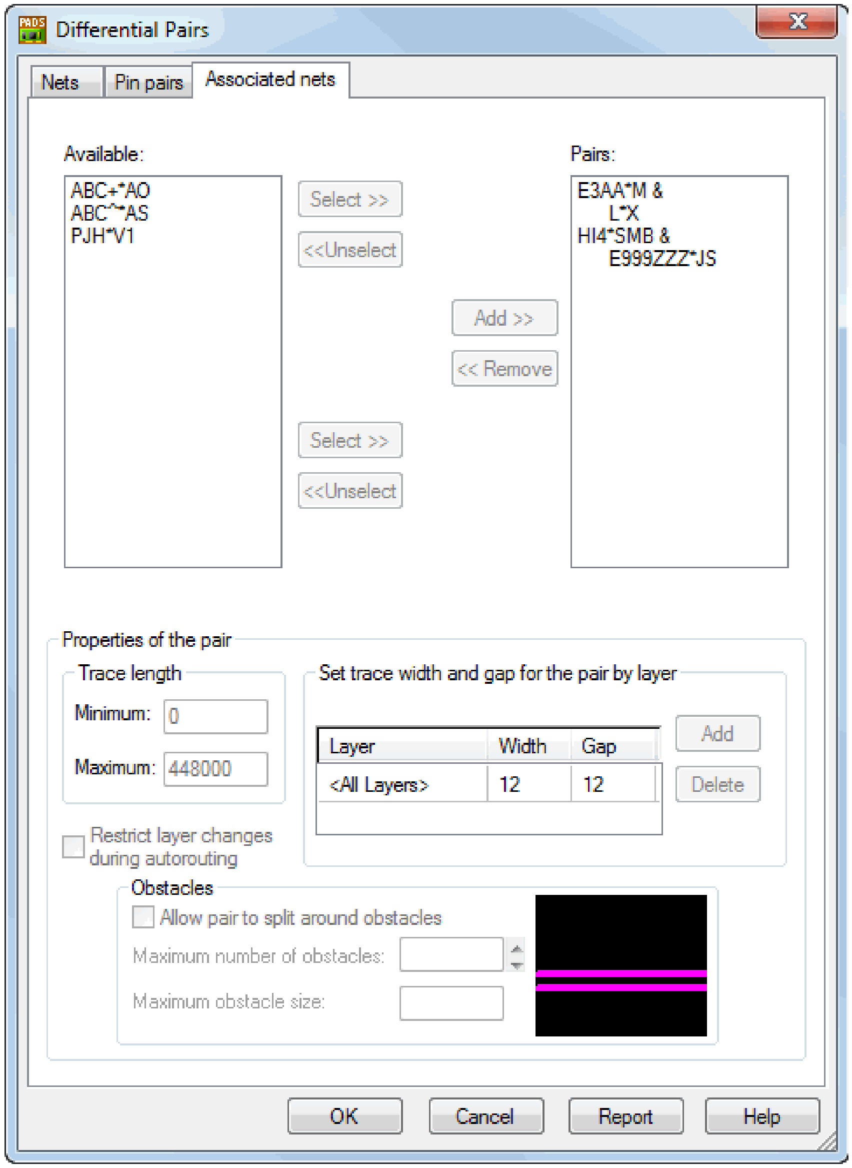


Table 45-139. Differential Pairs Dialog Box Contents

Name	Description
Available list	Lists the nets, pin pairs, or associated nets available for differential pair creation. **Tip:** Nets, associated nets, or pin pairs cannot be part of more than one differential pair. The Available list displays only items that have not been assigned to a differential pair.

Table 45-139. Differential Pairs Dialog Box Contents (cont.)

Name	Description
Select>>	Use the top and bottom Select buttons to move the first and second items making up a diff pair to a holding area, from which you can add them to the Pairs list.
<<Unselect	Moves the first and second items back to the Available list.
Add>>	Moves the selected items to the Pairs list. **Tip:** This button is unavailable unless two items are selected and in the holding area.
<<Remove	Moves the selected pair from the Pairs list back to the Available list.
Pairs list	Lists the created differential pairs.
Trace length area	Specifies the minimum and maximum length for a trace.
Restrict layer changes during autorouting check box	Forces the selected pair to be routed on a single layer. **Tip:** This setting does not restrict layer changes when routing interactively.
Set trace width and gap for the pair by layer table	Specifies the width and gap per layer. • **Layer**—The layer to set width and gap values. **Tips:** • Setting the differential pair width and gap per layer enables you to better control impedance. • If you select multiple differential pairs, and a layer setting does not belong to all of the selected pairs, the Layer column is unavailable. • **Width**—Specifies the width value for differential pairs on the specified layer. • **Gap**—Specifies the gap value for differential pairs on the specified layer. **Important:** The Gap also specifies the trace to trace clearance. This clearance takes precedence over any other trace to trace clearance in the rules hierarchy. • **Add** button—Adds a row to the bottom of the table to specify width and gap values for another layer. • **Delete** button—Removes the selected row from the table. **Restriction:** You cannot remove the <All Layers> row.

Table 45-139. Differential Pairs Dialog Box Contents (cont.)

Name	Description
Obstacles	• **Allow pair to split around obstacles**—Specifies to enable routing around an obstacle in the controlled gap area by temporarily exceeding the pair routing gap. **Tip:** This setting applies to autorouting and does not restrict splitting around obstacles when routing interactively. • **Maximum number of obstacles**—Specifies the maximum number of obstacles to route around in the Maximum number of obstacles box. Obstacles in the start zone or end zone are not counted. • **Maximum obstacle size**—Specifies the maximum spacing allowed between traces around obstacles in the Maximum obstacle size box. The size applies to the obstacle's longest horizontal or vertical dimension. Obstacle size in the start zone or end zone is not checked.
Preview area	Shows the way in which the differential pairs will split around objects based on your selections.

Related Topics

Creating Differential Pair Design Rules

Deleting a Differential Pair Design Rule

Dimension Properties Dialog Box

The Dimension Properties dialog box reflects the type of dimension, Vertical, Horizontal, Aligned, that is currently selected.

Accessing

- Select a dimension > right-click > **Properties**.

Figure 45-153. Dimension Properties Dialog Box

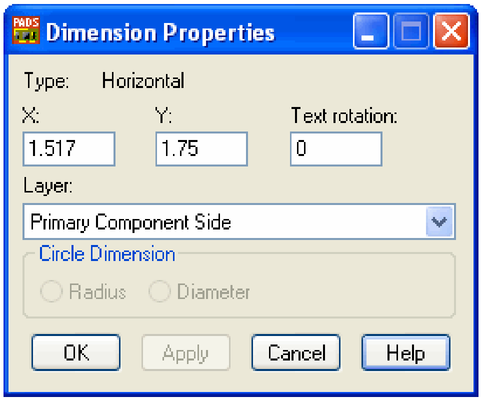


Table 45-140. Dimension Properties Dialog Box contents

Name	Description
Type	Displays the type of the selected dimension.
X/Y	Specifies the x and y coordinates of the dimension object. The coordinates are calculated from the bottom of one of the extension lines or from the radius point of an arc. Type a new value to change the location.
Text Rotation	Specifies the current rotation value. Positive values rotate entries in a counterclockwise direction. Negative values rotate entries in a clockwise direction. Type a new value to change the rotation.
Layer list	Specifies the new layer to which to assign the entire dimension object, even if different layers were assigned for text and lines in the Dimensioning / General page of the Options dialog box.
Circle Dimension	Allows modification of radial dimensions to indicate measurements from the radius point or diameter. Click one of these buttons to change the measurement type: • **Radius**—Measures the circle dimension from the radius point. • **Diameter**—Measures the circle dimension from the diameter.

Related Topics

Dimensioning Process

Dimension Text Properties Dialog Box

The Dimension Text Properties dialog box displays information about the selected text string and provides several areas for modifying the string. It remains open until you click OK or Cancel. Selecting another text object while the dialog box is open updates the information for the selected object.

After a dimension object is created, you might need to change the location of the text string to accommodate existing dimensions. Use either Dynamic Drag or the Move command to move the object.

Accessing

- Select dimension text > right-click > **Properties**

Figure 45-154. Dimension Text Properties Dialog Box

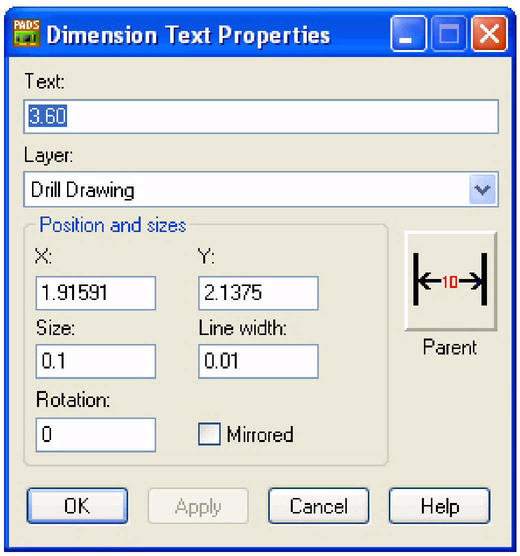


Table 45-141. Dimension Text Properties Dialog Box contents

Name	Description
Text	Displays the current content of the selected string. Type in the box to modify the text string. **Tip:** Modifying the content of the text string does not adjust the extension line to accommodate the new text length. Use Change Length to modify the position of the extension line and update the text string.
Layer list	Lists the current working layer. Select a new layer from the list.

Table 45-141. Dimension Text Properties Dialog Box contents (cont.)

Name	Description
X and Y boxes	Lists the X and Y coordinate locations of the text, calculated from the lower left corner of the text string. Type new values to change the location of the text.
Size	Specifies the size of the font. **Size (pts):** This is font size in points and appears for system fonts **Size (mils):** This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters. 
Line Width	Specifies the line width for stroke fonts only. 
Rotation	Lists the current rotation value. Positive values rotate in a counterclockwise direction. Negative values rotate in a clockwise direction. Type a new value to change the rotation.
Mirrored	Select to flip the text - text is considered readable from the bottom side of the board.
Parent Button	Opens the Dimension Properties dialog box for the dimension object with which the selected object is associated.

Related Topics

Dimensioning Process

Discarding Plane Data Dialog Box

Use the Discarding Plane Data dialog box to control which plane data is saved.

Accessing

The Discarding Plane Data dialog box appears automatically when you save a file in which you have plane data, and have set Prompt to discard plane data on the Options dialog box.

Figure 45-155. Discarding Plane Data Dialog Box

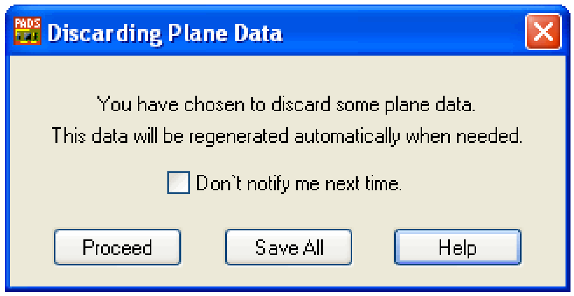


Table 45-142. Discarding Plane Data Dialog Box Contents

Name	Description
Don't notify me next time	Specifies to suppress this message for subsequent file saves.
Proceed	Saves only the plane polygons.
Save All	Saves all plane data and changes the option for future saves.

Disconnect Pin Dialog Box

Use the Disconnect Pin dialog box to confirm the disconnection of the pin from the net to which it's attached and to delete the route segment(s) that are attached to the pin.

Warning

If you choose not to delete any routes that are attached to the pin, the routes will still appear as attached in the design and the gerber output files. Without deleting the routes, the pin is only disconnected in the netlist.

Accessing

- select a pin > **ECO Toolbar** > **Delect Connection**

Figure 45-156. Disconnect Pin Dialog Box

Table 45-143. Disconnect Pin Dialog Box Contents

Name	Description
Delete Routes	Select to delete the route segment(s) that are connected to the pin. If you don't delete the routes, the pin will still appear as though it's attached. **Tip:** Only the routes of the pin pairs to/from the disconnected pin are deleted. All other segments belonging to the net are retained.

Related Topics

Deleting a Connection in ECO Mode

Display Colors Setup Dialog Box

Use the Display Colors Setup dialog box to Set display colors, save them, and restore them; Change the color palette; make objects visible; and make objects invisible.

Tip: Changes you make to the color configuration in the Display Colors Setup dialog box do not apply to disabled layers.

Accessing

- **Setup** menu > **Display Colors**

Figure 45-157. Display Colors Setup Dialog Box

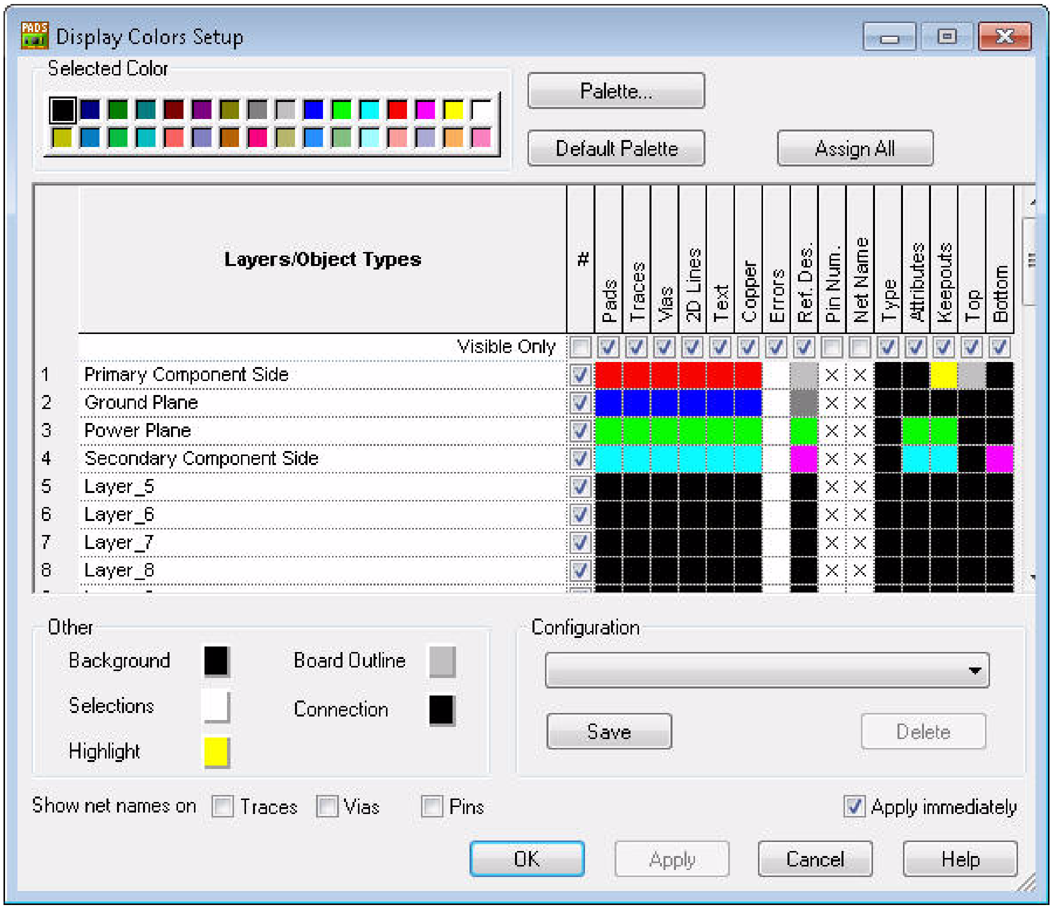


Table 45-144. Display Colors Setup Dialog Box Contents

Name	Description
Selected Color area	Select a color from the palette to assign to items on a layer. Once you select a color here, click the tile in the Color by Layer area of the item to which you want to assign the color. **See also:** To Change the Color Palette
Palette	Opens the Color dialog box where you can choose to use new colors or customize colors you want to use. **See also:** To Change the Color Palette
Default Palette	Reassigns all colors and settings to the default settings. **Tip:** You can change the default settings by saving a configuration and naming it default.
Assign All	Opens the Assign Color to All Layers dialog box.

Table 45-144. Display Colors Setup Dialog Box Contents (cont.)

<table>
<tr><th>Name</th><th>Description</th></tr>
<tr><td colspan="2">Layers/Object Types matrix—Use this area to assign various colors to different objects on different layers.
See also: To Assign Colors to Objects on Different Layers</td></tr>
<tr><td>Layers (rows)</td><td>The rows of layers lists the layers as you've named them in the Layers Setup dialog box.
Tips:
• You can clear the check box of a row to make the layer invisible.
• To make all items on a layer one color, click the number of the row to select all row objects and then click a color tile in the Selected Color area. You can also select multiple rows using Shift+click for a range or Ctrl+click for a selection.</td></tr>
<tr><td>Object Types (columns)</td><td>The columns list object types in the design.
Tips:
• You can clear the check box of a column to make the objects invisible on all layers.
• To make all of one object type the same color on each layer, click the object name in the column header to select the object on all layers and then click a color tile in the Selected Color area. You can also select multiple columns using Shift+click for a range or Ctrl+click for a selection.
• Pin Numbers and Net Names—sizing is controlled by the settings in the Display Options.
Restriction: Even if the Net Name column check box is selected and the tiles on layers are given a color, the display of net names is still restricted by the state of the Show net names on Traces, Vias, Pins check boxes.</td></tr>
<tr><td>Visible only</td><td>Lists only visible layers. A layer is visible if at least one tile is assigned a non-background color.</td></tr>
<tr><td colspan="2"></td></tr>
<tr><td>Other area</td><td>Globally assigns colors to other items. Click a color from the Selected Color area and click the tile of the item. You can globally control the colors for the following objects:
• Background—Setting other objects to this color makes them invisible.
• Selections—Objects that are selected for modifying.
• Highlight—Objects that are highlighted but not selected for edit operations.
• Board outline—Applies to the board outline and board cut outs.
• Connection—Unrouted pin pairs or the "ratsnest."</td></tr>
</table>

Table 45-144. Display Colors Setup Dialog Box Contents (cont.)

Name	Description
Configuration list	The list of saved configurations.
Save	Opens the Save Configuration dialog box.
Delete	Removes the selected configuration from the Configuration list.
Show net names on Traces, Vias, Pins	Select the check boxes to activate locations where net names should be visible. **Requirement:** To display net names on these objects, you must also select the check box for the Net Names column and give colors to color tiles in the column. **Tips:** • You can use the modeless commands NNT, NNV and NNP to toggle these check boxes. • The sizing and frequency of net name placement is controlled by the settings in the Display Options.
Apply immediately	Specifies that any change in color or visibility made in this dialog box is immediately applied to the design; clicking the Apply button is not needed.

Related Topics

Display Colors Setup Dialog Box in the Decal Editor

Setting Colors of Objects in the Display

To Save Color Assignments to a File

Display Colors Setup Dialog Box in the Decal Editor

Use the Display Colors Setup dialog box to Set display colors, save them, and restore them; change the color palette; make objects visible; and make objects invisible.

Restrictions

- Color changes are not permanently saved in the Decal Editor. To make permanent changes to the colors, see Creating a New Default Decal Editing Environment.
- Changes you make to the color configuration in the Display Colors Setup dialog box do not apply to disabled layers.

Accessing

- **Tools menu** > **PCB Decal Editor** > **Setup** menu > **Display Colors**

Figure 45-158. Display Colors Setup Dialog Box in the Decal Editor

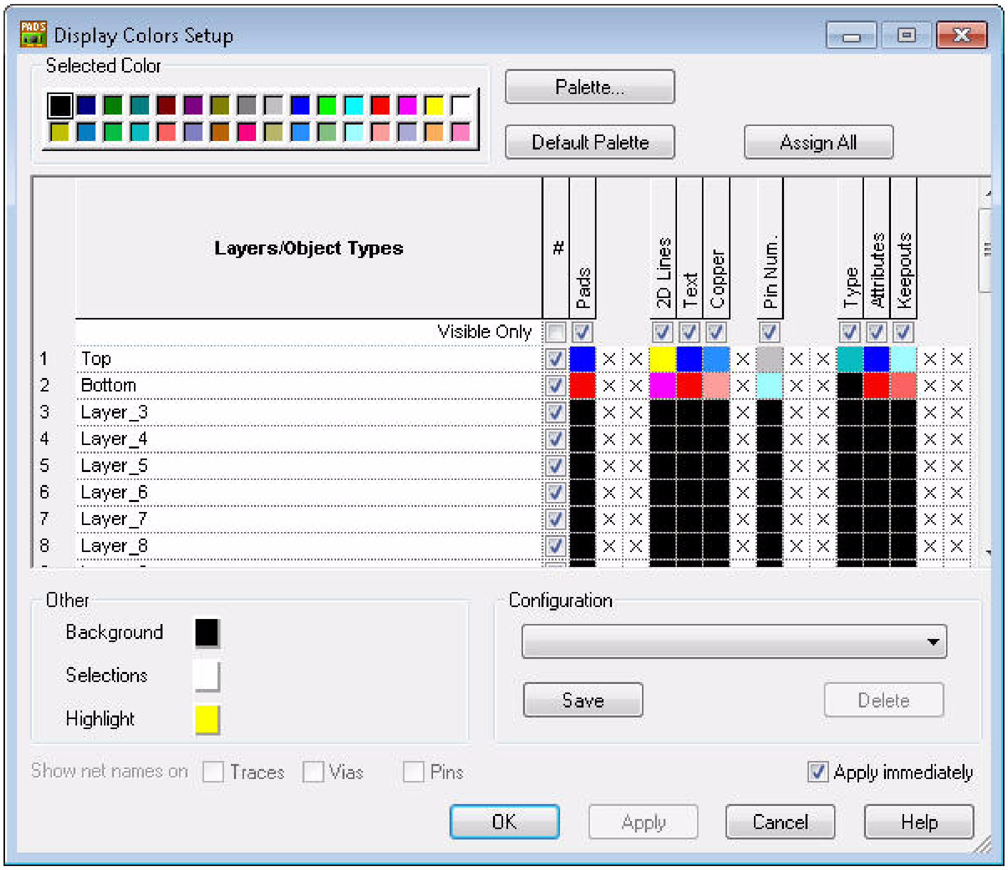


Table 45-145. Display Colors Setup Dialog Box in the Decal Editor Contents

Name	Description
Selected Color area	Select a color from the palette to assign to items on a layer. Once you select a color here, click the tile in the Color by Layer area of the item to which you want to assign the color. **See also:** To Change the Color Palette
Palette	Opens the Color dialog box where you can choose to use new colors or customize colors you want to use. **See also:** To Change the Color Palette
Default Palette	Reassigns all colors and settings to the default settings. **Tip:** You can change the default settings by saving a configuration and naming it default.
Assign All	Opens the Assign Color to All Layers dialog box.

Table 45-145. Display Colors Setup Dialog Box in the Decal Editor Contents

Name	Description
Layers/Object Types matrix—Use this area to assign various colors to different objects on different layers. **See also:** To Assign Colors to Objects on Different Layers	
Layers (rows)	The rows of layers lists the layers as you've named them in the Layers Setup dialog box. **Tips:** • You can clear the check box of a row to make the layer invisible. • To make all items on a layer one color, click the number of the row to select all row objects and then click a color tile in the Selected Color area.
Object Types (columns)	**The columns list object types in the design.** **Tips:** • Although the Ref. Des. column is not visible as in the Layout Editor, it is part of the Pin Num. column. • You can clear the check box of a column to make the objects invisible on all layers. • To make all of one object type the same color on each layer, click the object name in the column header to select the object on all layers and then click a color tile in the Selected Color area.
Visible only	Lists only visible layers. A layer is visible if at least one tile is assigned a non-background color.
Color by Layer matrix	Use this area to assign various colors to different objects on different layers. **Tip:** The layer names that you assigned in the Layers Setup dialog box appear along the left side of this area. **See also:** To Assign Colors to Objects on Different Layers
Other area	Globally assigns colors to other items. Click a color from the Selected Color area and click the tile of the item. You can globally control the colors for the following objects: • **Background**—Setting other objects to this color makes them invisible. • **Selections**—Objects that are selected for modifying. • **Highlight**—Objects that are highlighted but not selected for edit operations.
Configuration list	The list of saved configurations.
Save	Opens the Save Configuration dialog box.
Delete	Removes the selected configuration from the Configuration list.

Table 45-145. Display Colors Setup Dialog Box in the Decal Editor Contents

Name	Description
Apply immediately	Specifies that any change in color or visibility made in this dialog box is immediately applied to the design; clicking the Apply button is not needed.

Related Topics

Creating a New Default Decal Editing Environment

Setting Colors of Objects in the Display

To Save Color Assignments to a File

Drafting Corner Properties

Use the Drafting Corner Properties to view an object type, layer assignment, trace clearance, or assigned net. You can move a corner by coordinates, access net information, and select the parent shape.

Accessing

- **Select a drafting edge > Right-click > Properties**

Figure 45-159. Drafting Corner Properties Dialog Box

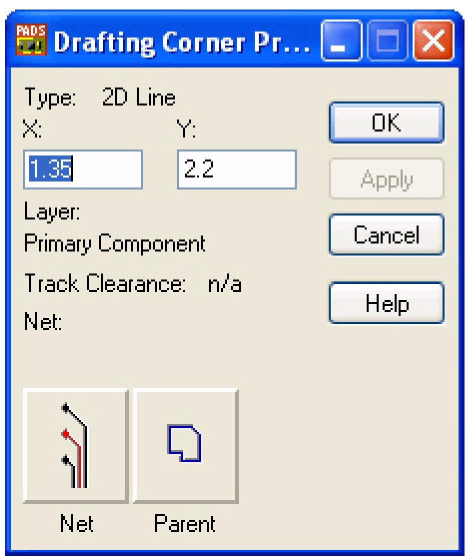


Table 45-146. Drafting Corner Properties Dialog Box contents

Name	Description
Type	Lists the object type.
X,Y	Lists the current X,Y location of the corners. Type new values in the boxes to move the corner.
Layer	Lists the layer on which the object is located.
Track Clearance	Specifies clearance values between the corner and objects around it.
Net	Lists the net associated with the corner.
Net button	Opens the Net Properties dialog box.
Parent button	Opens the Drafting Properties dialog box for the drafting object to which the corner belongs.

Related Topics

Modifying Drafting Corner Properties

Drafting Edge Properties Dialog Box

Use the Drafting Edge Properties dialog box to view an object type, layer assignment, trace clearance, or assigned net. You can move an edge by coordinates or an arc by radius or start/end angle, access net information, and select the parent shape.

Exceptions:

- For circles, you can change only Center Point and Radius.
- For arcs, use the X1,Y1 and X2,Y2 fields as the arc end points. To define an arc, you only need a subset of these fields. To redefine the arc, change only one field at a time. If you make a change that cannot be interpreted, the command is cancelled.

Accessing

- **Select a drafting edge** > **Right-click** > **Properties**

Figure 45-160. Drafting Edge Properties Dialog Box

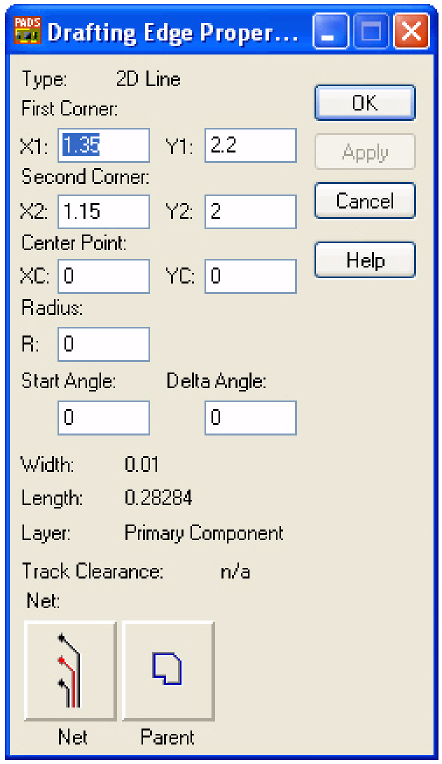


Table 45-147. Drafting Edge Properties Dialog Box contents

Name	Description
X1,Y1 / X2,Y2 boxes	List the current X,Y location of the first and second corners of the edge. Type new values in the boxes to move one or both corners.
XC, and YC boxes	The center coordinate of the arc or circle.
Radius box	The radius for circles or arcs. Type a new value for the radius.
Start Angle box	The starting angle of the arc. Zero (0) degrees is the positive X axis and positive angles are created counterclockwise.
Delta Angle box	The number of degrees in the angle. Positive angles are created counterclockwise. **Tips:** • For circles, you can only change Center Point and Radius. • For arcs, use the X1,Y1 and X2,Y2 fields as the arc end points. To define an arc, you only need a subset of these fields. To redefine the arc, only change one field at a time. If you make a change that can't be interpreted, the command is canceled.

Table 45-147. Drafting Edge Properties Dialog Box contents (cont.)

Name	Description
Width	Lists the width of the edge.
Length	Lists the length of the edge.
Layer	Lists the layer on which the object is located.
Track Clearance	Specifies clearance values between the corner and objects around it.
Net	Lists the net associated with the edge.
Net button	Opens the Net Properties dialog box.
Parent button	Opens the Drafting Properties dialog box for the drafting object to which the corner belongs.

Related Topics

Modifying Drafting Edge Properties

Drafting Properties Dialog Box

You can select and edit drafting shapes in pieces or as whole items. The properties you can modify depend on whether you select a corner of the drafting object, an edge of the drafting object, or the whole, or parent object of a 2D line, copper, or text.

Exception: Several of the options in this dialog box are unavailable if the object is part of a physical design reuse.

Accessing

- **Select a drafting shape > Right-click > Properties**

Figure 45-161. Drafting Properties Dialog Box

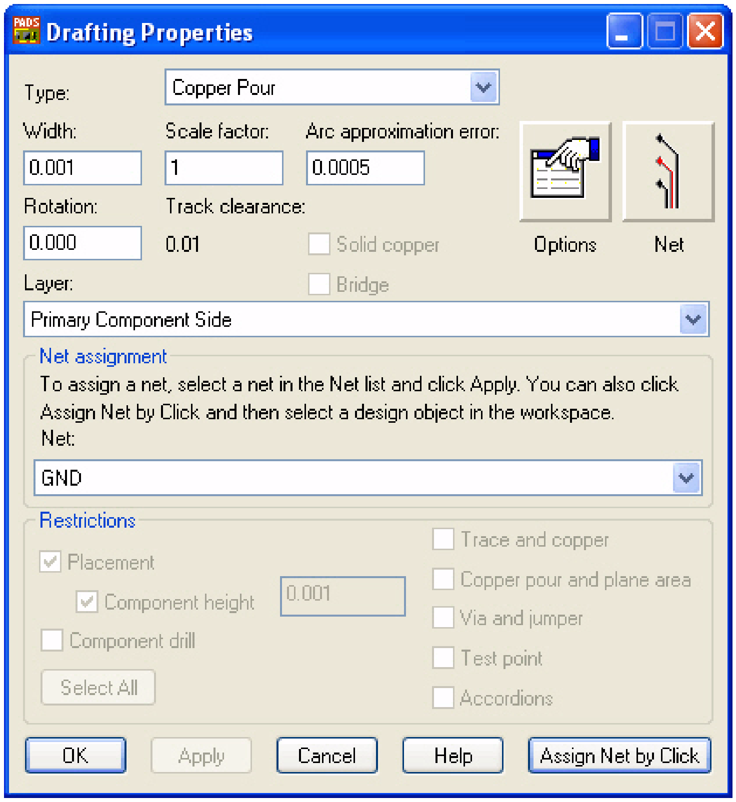


Figure 45-162. Showing the Nets to Bridge Button

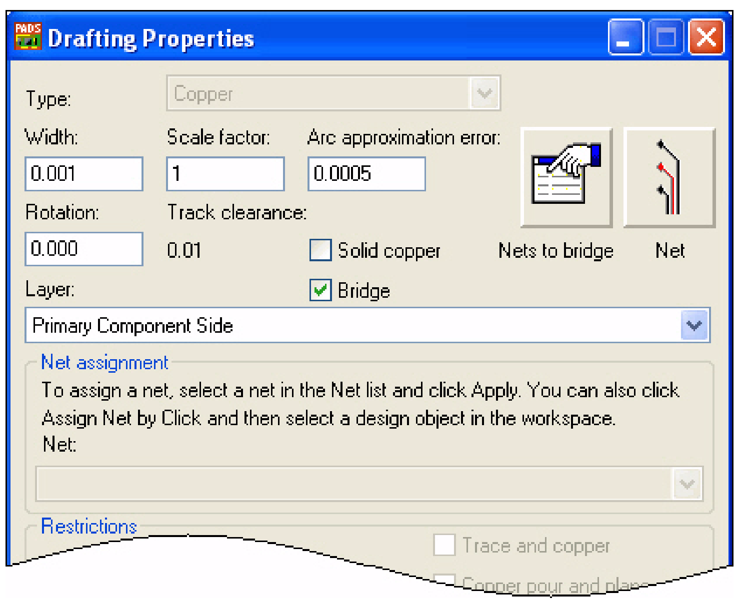


Table 45-148. Drafting Properties Dialog Box contents

Name	Description
Type	Converts the selected shape to a new one: 2D Line, Board cut out, Copper, Copper cut out, Keep out, Copper pour, Plane area, Plane area cut out. **Tip:** If your lines appear to be a closed polygon but you receive the error, “You cannot convert an open 2D Line into a close...”, with your shape selected, right-click and click the Close command. Unless there is a physical gap somewhere in your lines, your lines will be closed into a single polygon shape. **Restrictions:** • 2D lines that have been combined cannot be change into other shapes. First, explode the 2D line combination and then try to change to another object. • 2D lines that are continuous but are not a closed polygon can only be changed into a copper path.
Width	Specifies the width of the lines used for the outline and fill of the object if it applies.
Scale factor	Specifies the scaling of the object.
Arc approximation error	Specifies the allowable approximation error for scaling arcs. Arcs are converted to a set of straight-line segments that approximate the arc. The approximation error is the perpendicular distance from the approximation segment to the actual arc.
Rotation	Specifies the rotation of the object
Track clearance	Displays the clearance value between the drafting object and objects around it.
Solid copper	Specifies to fill the shape with solid copper despite the Width and Copper Hatch grid values which normally dictate the fill pattern of copper objects.

Table 45-148. Drafting Properties Dialog Box contents (cont.)

Name	Description
Bridge	Specifies to make the drafting object a copper to bridge two or more nets. Click the **Nets to bridge** button to associate the bridge nets to the copper. This ensures that with design rule checking (DRC) enabled, the bridge copper isn't flagged as a violation when it is physically bridging different nets. Only the bridge copper is not flagged by DRC; other objects accidentally connecting the nets will be flagged as a violation by DRC and/or caught by a Verify Design Clearance check. The Connectivity check also catches bridge copper that is not physically connected to its assigned nets. **Restriction:** This is only available to Copper drafting objects. **See also:** Bridging Nets with Copper
Options button	Opens the Flood and Hatch Options dialog box. **Restriction:** This button is available only for Copper Pour and Plane Area drafting objects.
Nets to bridge button	Opens the Net Association dialog box in order to select the nets to bridge using the copper. **See also:** Bridging Nets with Copper
Net button	Opens the Net Properties dialog box in order to view and change the properties of the net assigned to the drafting object. **Restriction:** The net doesn't function if there is no net assigned to the drafting object.
Layer list	Specifies the layer on which this object is located. **Restriction:** Copper, copper pours and plane areas cannot be placed on <All Layers>. If you need the object on all layers, you must copy the object to the other layers. **Tip:** When defining keepouts in the PCB Decal Editor, you can also assign keepouts to an <Opposite Side> layer. You can't do this in the Layout Editor.
Net assignment area	Specifies the net to assign to the drafting object. **Alternative:** You can also use the *Assign Net by Click* button also found in this dialog box. **Restrictions:** • Available for electrical drafting objects only. • Not available when the Bridge check box is selected. Multiple nets apply when using the copper as a bridge. Associate the nets using the Nets to bridge button instead.
Restrictions area This area is only available to Keepout drafting objects.	

Table 45-148. Drafting Properties Dialog Box contents (cont.)

Name	Description
Placement	**Prevents placement of components** within the keepout area when design rule checking is enabled**. When used in combination with the** *Component height* **check box, it prevents placement only of those components with heights greater than the Component height value.** **Restriction:** Unavailable when creating or editing a decal-level keepout.
Component height	Prevents placement of components with heights greater than the specified height within the keepout area when design rule checking is enabled. Type the maximum height value in the box to the right of Component Height. **See also:** Restricting Heights in Areas of Component Layers. **Restrictions:** • Unavailable unless you select the Placement check box. • Unavailable when creating or editing a decal-level keepout.
Component drill	Prevents placement of components that contain drilled through holes within the keepout area, when design rule checking is enabled. (Allows only surface mount placement.) **Restrictions:** • Unavailable unless you set the Layer list to <All Layers>. • Unavailable when creating or editing a decal-level keepout.
Select All button	Selects all check boxes in the Restrictions area except Component drill.
Trace and copper	Prevents the placement of traces within the keepout area, when design rule checking is enabled. **Restriction:** Copper areas are always placed with design rules disabled. Placement of copper areas will not be prevented but the Verify Design utility will report any copper located within the keepout area.
Copper pour and plane area	Prevents copper pours and split/mixed planes from flooding within the keepout area, when design rule checking is enabled.
Via and jumper	Prevents the placement of vias and jumpers within the keepout area, when design rule checking is enabled.
Test point	Restricts the placement of test points within the keepout area, when design rule checking is enabled.

Table 45-148. Drafting Properties Dialog Box contents (cont.)

Name	Description
Accordions	Restricts the placement of accordions within the keepout area, when design rule checking is enabled. **Tips:** • While you can set this option in PADS Layout, it applies to PADS Router only. • Accordion Keepouts should be used for batch and interactive routing only. Design Verification does not detect accordions that are placed on accordion keepouts.
Assign Net by Click	Specifies to assign nets to electrical drafting objects by clicking objects in the design. You can click objects such as a pin, via, trace, net, copper, or unroute to assign the netname of the object to the copper shape. **Alternative:** You can also select a net from the list in the Net assignment area of this dialog box. **Restriction:** Not available when the Bridge check box is selected. Multiple nets apply when using the copper as a bridge. Associate the nets using the Nets to bridge button instead.

Related Topics

Modifying Drafting Object Properties

Drill Drawing Options Dialog Box

Use the Drill Drawing Options dialog box to set drill drawing legend and marker parameters.

Accessing

- **File Menu > CAM > Add button > Select Drill Drawing from the Document Name List > Options button**

  or

- **File Menu > CAM > Select a document name > Edit button > Select Drill Drawing from the Document Name List > Options button**

Figure 45-163. Drill Drawing Options Dialog Box

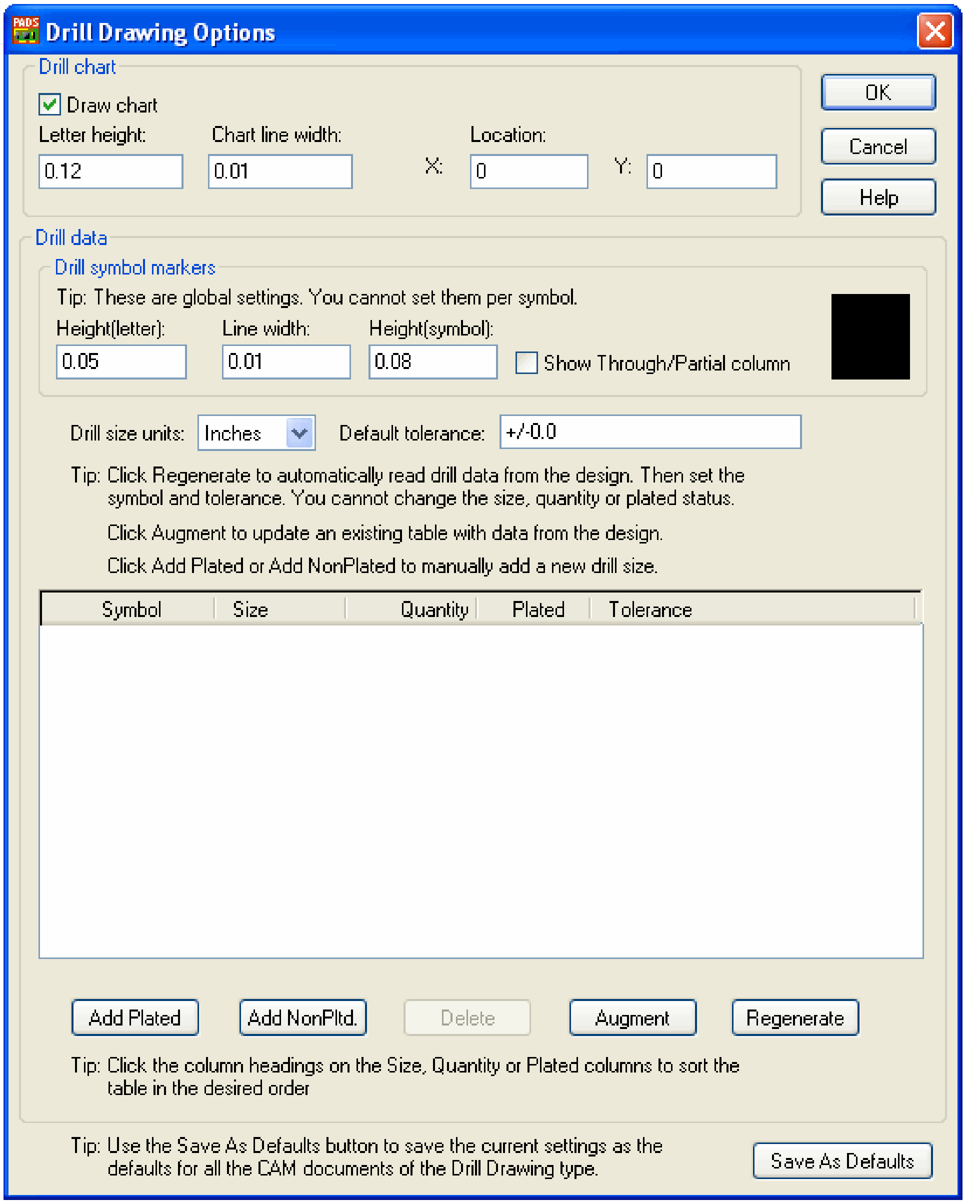


Table 45-149. Drill Drawing Options Dialog Box contents

Name	Description
Draw Chart	Specifies to includes the legend in the plot.
Letter Height	Specifies the height of letters in design units.
Chart Line Width	Specifies the width of the chart lines in design units.

Table 45-149. Drill Drawing Options Dialog Box contents (cont.)

Name	Description
X/Y Location	Specifies location of the drill chart in design units. **Tip:** The location is not saved when you click the Save As Defaults button.
Height(letter)	Specifies the letter height of the drill symbol marker.
Line Width	Specifies the line width for the drill marker symbol and for all text.
Height(Symbol)	Specifies the drill marker symbol height.
Show Through/Partial column	Specifies to show the Through/Partial column in the table.
Drill size units	Specifies the units to use for the drill size.
Default tolerance	Specifies the tolerance to use in the Tolerance column in the table.
Symbol and Size columns	Symbols are assigned in the order that the drill sizes are read into the table; for a manually created entry, the next available symbol is used. Symbol usage is exclusive to a drill size. The symbol assignment order for the 64 supported symbols is as follows: • Six of the 12 available graphical symbols are used first (+, X, Rectangle, Diamond, Hour Glass, and Bow Tie) • Letter symbols +A through +Z are assigned • Six more graphical symbols (Rectangle +, Rectangle X, Diamond +, Diamond X, Circle +, and Circle X) are assigned • Rectangle +A through Rectangle +Z are assigned
Quantity column	The Quantity column displays the count of each drill size in the design. It may contain a zero value for drill sizes loaded from the *cam.defaults* file, or for alternate drill sizes contained in the design database but not currently used in the design. See the Save As Defaults button for more information.
Plated column	The Plated column displays whether the plating type for each drill size is plated (Yes) or non-plated (No). Duplicate drill sizes can appear in the table if each has a unique plating type.

Table 45-149. Drill Drawing Options Dialog Box contents (cont.)

Name	Description
Through/Partial column	The check boxes in the Through/Partial column specify whether the drills listed in that row are to be output to the document (checked) or not output (unchecked). The boxes are checked by default. **Tips:** • This column is displayed only if the Show Through/Partial column check box in the Drill Data area is selected. • When this column is displayed, if both through and partial drills of a given drill size exist, they are counted separately and appear in separate rows. When this column is not displayed, through and partial drills of a given size are totaled together and appear in a single row.
Tolerance	Specifies the tolerance of the drill size.
Add Plated button	Adds a specific plated drill size to the drill table. The new entry is assigned the next-available drill symbol. **Tip:** To manually add a slotted hole to the drill table, use the format <*width*> X <*slot_length*> (for example, 0.020 X 1.000).
Add NonPltd button	Add a specific non-plated drill size to the drill table. The new entry is assigned the next-available drill symbol. **Tip:** To manually add a slotted hole to the drill table, use the format <*width*> X <*slot_length*> (for example, 0.020 X 1.000).
Delete button	Removes selected drill size entries from the table. To remove multiple entries, use Ctrl or Shift and Click to select them before you click delete.
Augment button	Automatically creates (appends) drill sizes for items in the current design file that are not already defined in the list. Your list could be prepopulated by a set of drill sizes that may or may not be in use in the current design. See the information for the Save As Defaults button for more information. Using Augment will retain any unused hole/drill size listings. **Tip:** The drill table reflects entries for slotted holes if they exist in the design database after you click Augment.
Regenerate button	Clears the drill table and replaces all existing data with the data from the design database. Your list could be prepopulated by a set of drill sizes that may or may not be in use in the current design. See the information for the Save As Defaults button for more information. Using Regenerate clears and creates a list of only the hole/drill sizes used in the design. **Tip:** The drill table reflects entries for slotted holes if they exist in the design database after you click Regenerate.

Table 45-149. Drill Drawing Options Dialog Box contents (cont.)

Name	Description
Save As Defaults button	If you manually build your own drill table, or you want to reuse the contents of the table as the defaults for all CAM drill drawings you can save the contents as the default. This creates a *cam.defaults* file located in the following location (default installation): C:\MentorGraphics\<version>PADS\SDD_HOME\Settings **Restriction:** The drill chart location is not saved when you click Save As Defaults.

Related Topics

Setting Drill Drawing Options

Creating CAM Outputs to Manufacture Your PCB

Adding or Editing CAM Documents

Drill Pairs Setup Dialog Box

Use the Drill Pairs Setup dialog box to define which layers to drill and plate together during manufacturing. Define these layers first to prevent defining or installing partial vias that cross layers that don't drill together.

Accessing

- **Setup** menu > **Drill Pairs**

Figure 45-164. Drill Pairs Setup Dialog Box

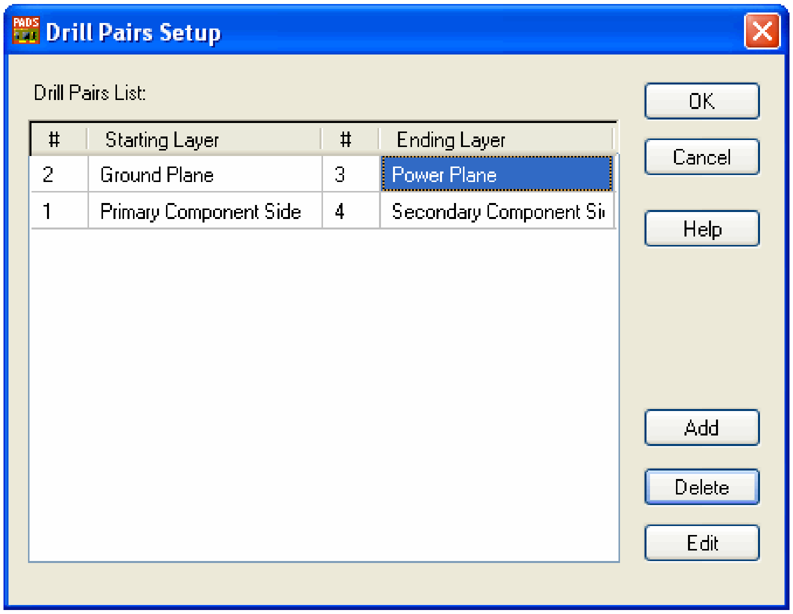


Table 45-150. Drill Pairs Setup Dialog Box Contents

Name	Description
#	Shows the number of the layer in the drill pair.
Starting Layer	Sets the starting layer of the drill pair.
#	Shows the number of the layer in the drill pair.
Ending Layer	Sets the ending layer of the drill pair.
Add	Adds a row to the table.
Delete	Removes the selected row.
Edit	Makes the selected cell available for editing.

Related Topics

Creating a Drill Pair

DxDesigner Link Dialog Box, Documents Tab

Use the Documents Tab of DxDesigner Link to connect a schematic in DxDesigner with a layout design in PADS Layout for bringing data forward and back and cross-probing.
See also: Working with DxDesigner.

Requirement: Ensure that cross-probing is turned on in DxDesigner; click Setup > Cross Probing. If there is a check mark to the left of Cross Probing, this option is already turned on.

Accessing

- **PADS Layout Tools** menu > **DxDesigner**

Figure 45-165. DxDesigner Link Dialog Box, Documents Tab

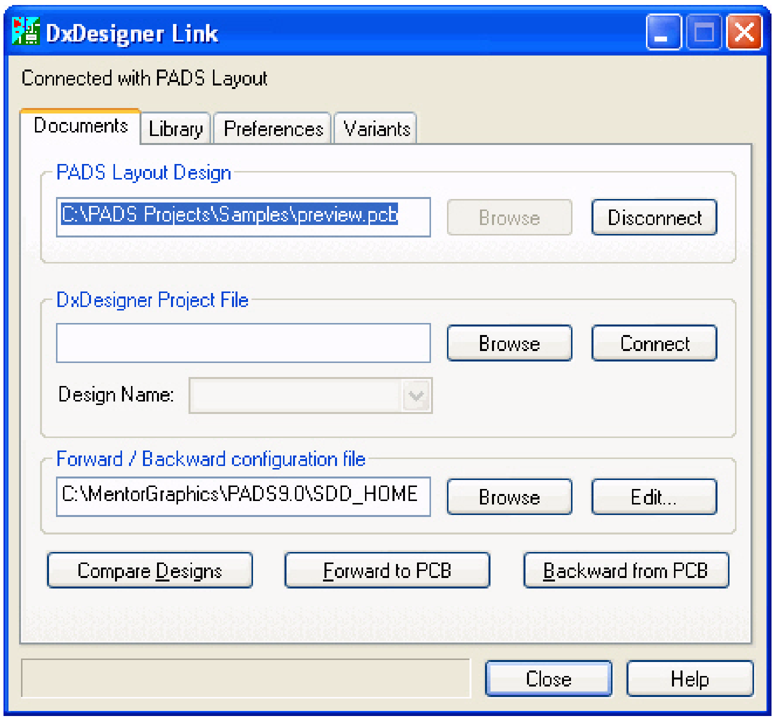


Table 45-151. Documents Tab contents

Name	Description
PADS Layout Design	The name of the PADS Layout design you want to link with DxDesigner. Click Browse to locate the file. **Tips:** • If you do not open a layout design, DxDesigner Link connects to the default layout design file (default.pcb), which you can save under a different name. • If the pathname lists a design, but you want to use the new default design opened in the design space, clear the pathname in the dialog box and then click Connect.
Disconnect/Connect button	This button changes names depending on if you are connected to PADS Layout or not. Click Connect to connect the file; Disconnect to disconnect it.

Table 45-151. Documents Tab contents (cont.)

Name	Description
DxDesigner Project File	The name of the DxDesigner project file you want to link with PADS Layout. Click Browse to locate the file. **Tip:** The DxDesigner Schematic field shows the file you last selected in DxDesigner Link.
Disconnect/Connect button	This button changes names depending on if you are connected to DxDesigner or not. Click Connect to connect the file; Disjoined to disconnect it.
Design Name	Lists the designs in the DxDesigner project file. Select the one you want.
Forward/Backward configuration file	The name of the configuration file you want to use, usually pads<*latest_release*>.cfg. Click Browse to locate the file. **Tip:** A default .cfg file is provided with DxDesigner. For PADS releases, it is located in the C:\MentorGraphics\<latest_release>PADS\SDD_HOME\standard folder. For PADS 9.x, the file name is pads90.cfg.
Edit button	Opens the configuration file in a text editor. See Schematic Design Help in DxDesigner for more information on editing configuration files. **See also:** "Configuration File Structure" topic in the Schematic Design Help in DxDesigner.
Compare Designs button	DxDesigner Link generates and displays three files: • A differences file (.dif) that reports the differences between the schematic and layout design files. • An error report (.err). • An Engineering Change Order file (.eco). The forward and backward annotation operations use this file to synchronize the layout design and the schematic files. **Tip:** Select the items you want to compare on the Preferences tab first. **See also:** Comparing Designs
Forward to PCB button	Opens the Forward Annotation dialog box.
Backward from PCB button	Opens the Backward Annotation dialog box.

Related Topics

Working with DxDesigner

DxDesigner Link Dialog Box, Library Tab

When you update a PADS Layout design with data from the schematic, you may also want to pass library parts or updates to library parts. Use the Library tab of the DxDesigner Link dialog box to specify how and to which PADS Layout library to save DxDesigner library information.

Accessing

- **Tools** menu > **DxDesigner** > **Library tab**

Figure 45-166. DxDesigner Link Dialog Box, Library Tab

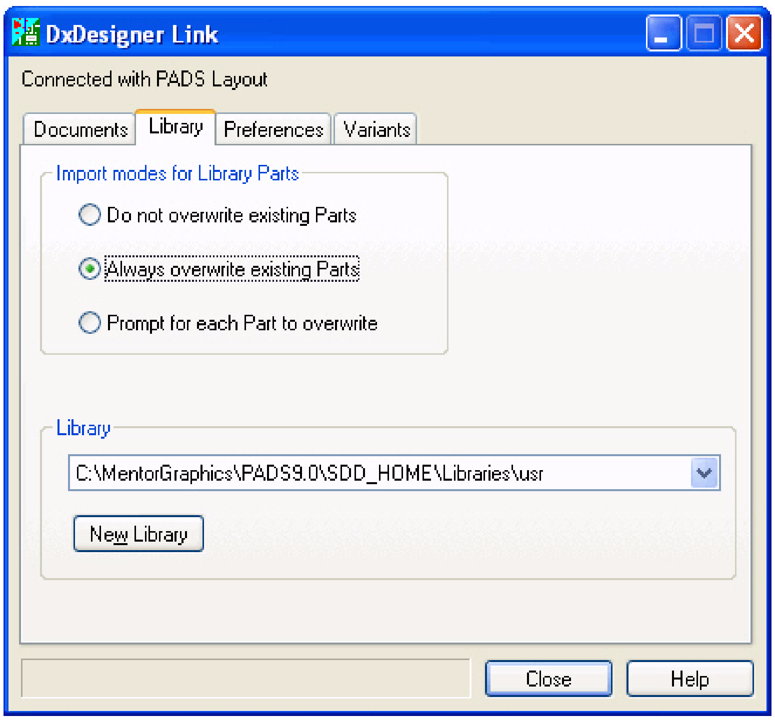


Table 45-152. Library Tab contents

Name	Description
Import modes for Library Parts area	**Specifies the mode you want for importing library parts:** • **Do not overwrite existing Parts**—Does not overwrite parts in the design library with parts from the schematic. • **Always overwrite existing Parts**—Overwrites parts in the design library with parts from the schematic. • **Prompt for each Part to overwrite**—Prompts you before overwriting parts in the design library with parts from the schematic.
Library list	Specifies the library into which to save DxDesigner parts.

Table 45-152. Library Tab contents (cont.)

Name	Description
New Library button	Click to create a new library for the imported parts from DxDesigner. The new library is added to the Library list. **Warning:** The library is added to the bottom of the search order.

Related Topics

Forward Annotating from DxDesigner to PADS Layout

DxDesigner Link Dialog Box, Placement Tab

When cross-probing, you may want to identify which parts are placed and which parts are unplaced. This allows you to easily select and place parts. Use the Placement tab of the DxDesigner Link dialog box to view the placement of parts in the schematic and design.

Tip: This tab is only available when DxDesigner Link is connected to both PADS Layout and DxDesigner.

Accessing

- **Tools** menu > **DxDesigner** > **Placement tab**

Figure 45-167. DxDesigner Link Dialog Box, Placement Tab

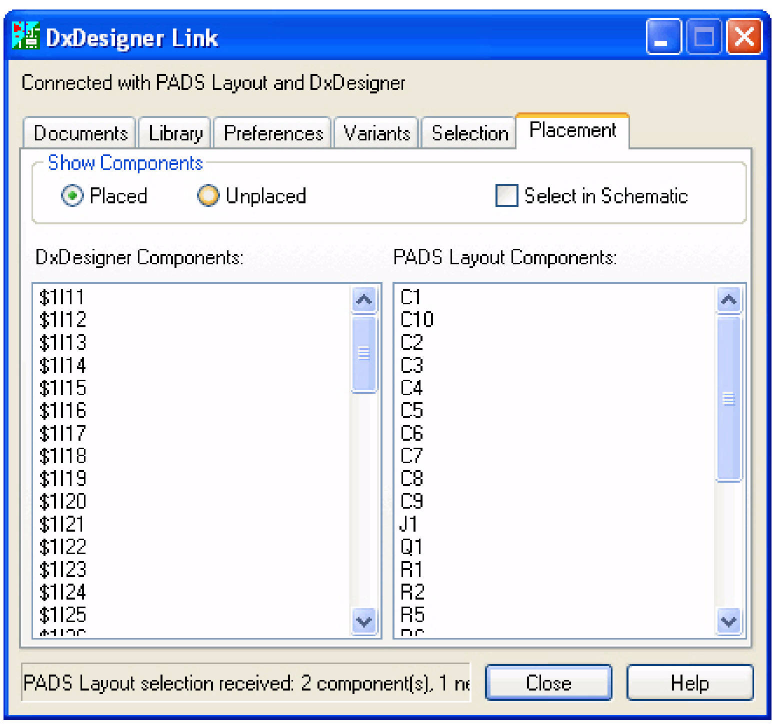


Table 45-153. Placement Tab contents

Name	Description
Show Components area	Specifies what type of components you want to show: Placed or Unplaced. Components are considered placed once they are located within the board outline.
Select in Schematic	Specifies to select all components in the schematic that are listed in the DxDesigner Components list.
DxDesigner Components list	Lists the components in DxDesigner that are placed if you selected placed, or unplaced if you selected unplaced.
PADS Layout Components list	Lists the components in PADS Layout that are placed if you selected placed, or unplaced if you selected unplaced.

Related Topics

Cross-Probing with DxDesigner

DxDesigner Link Dialog Box, Preferences Tab

In DxDesigner Link, both forward and backward annotation operations compare the schematic with the PADS Layout design. Forward annotation uses the results of comparison to update the PADS Layout design. Backward annotation uses the results of comparison to update the DxDesigner schematic.

In DxDesigner Link, use the Preferences tab to select the items for comparison.

Tips:

- Set preferences before you compare designs or forward/backward annotate.
- DxDesigner Link stores your preferences for future comparison and annotation operations. Set the preferences again only if you want to change them.

Accessing

- **Tools** menu > **DxDesigner** > **Preferences tab**

Figure 45-168. DxDesigner Link Dialog Box, Preferences Tab

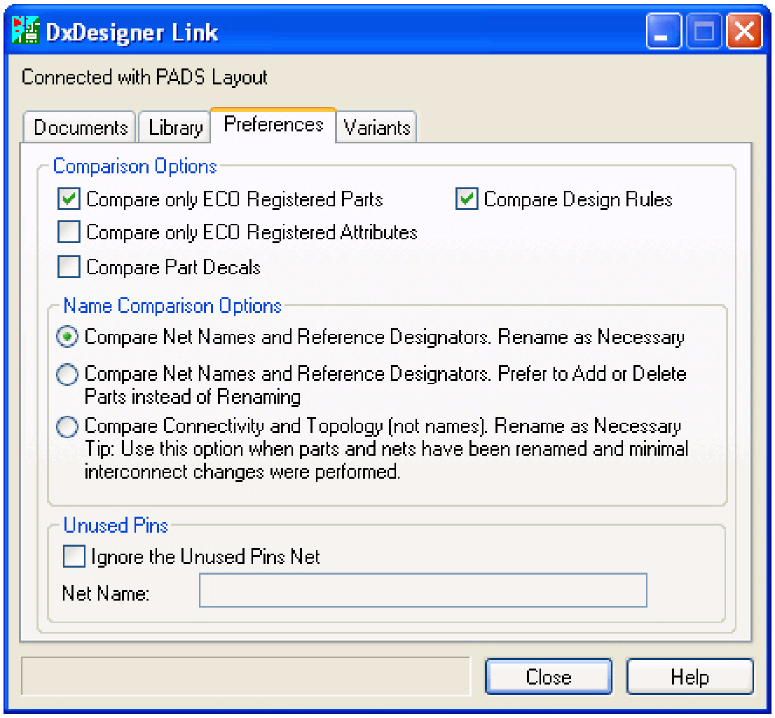


Table 45-154. Preferences Tab contents

Name	Description
Compare only ECO Registered Parts	Comparison excludes non-ECO-registered parts, such as mechanical or non-electrical parts present in the PCB design and not present in the schematic. To include all parts during comparison, clear the Compare Only ECO Registered Parts check box.
Compare only ECO Registered Attributes	Comparison excludes non-ECO-registered attributes. **Tip:** Via attributes are not ECO registered and cannot be added, deleted, or changed during the ECO process.
Compare Part Decals	Comparison includes part decals.
Compare Design Rules	Includes design rules in the comparison.

Table 45-154. Preferences Tab contents (cont.)

Name	Description
Name Comparison Options area	Specifies how you want to compare the names of design elements. • **Compare Net Names and Reference Designators. Rename as Necessary**—Compare differences using reference designators and net names. Best used to minimize changes to routed traces. Selecting this option may result in the positional swapping of parts. For example, if routed trace changes are minimized when R1 and R12 are swapped, simultaneously rename R12 to R1 and R1 to R12, and then reconnect R1 and R12 to the original nets. • **Compare Net Names and Reference Designators. Prefer to Add or Delete Parts Instead of Renaming**—Compare differences using reference designators and net names on the basis that few reference designators have been renamed and nets have not been renamed. Best used to minimize the positional swapping of parts, and the design disruption that may result. • **Compare Connectivity and Topology (not names). Rename as Necessary**—Compare differences without using reference designators or net names. Compare differences using pin names, part type names, and so on. Best used to compare designs when parts and nets have been renamed and minimal interconnect changes have been performed (for example, when only an auto renumber has been performed on the design).
Ignore the Unused Pins Net	Specifies that you want the design comparison to exclude the unused pins net in the PADS Layout design. Type the name of the unused pins net. An unused pins net contains pins that have no logical net association. Routing with SPECCTRA or other tools in the PCB design process can create unused pins nets. **Restriction:** You must use a net name of 47 characters or less. Use any alphanumeric characters except curly braces {}, asterisks *, or spaces. **Warning:** If you clear this option and you update the PCB design from a schematic or previous PCB layout, the unused pins net may be deleted.

Related Topics

Forward Annotating from DxDesigner to PADS Layout

Back Annotating from PADS Layout to DxDesigner

DxDesigner Link Dialog Box, Selection Tab

When cross-probing, you may want to set up how objects are selected in the two applications. Use the Selection tab of the DxDesigner Link dialog box to set up object selection between PADS Layout and DxDesigner.

Tip: This tab is available only when DxDesigner Link is connected to both PADS Layout and DxDesigner.

Accessing

- **Tools** menu > **DxDesigner** > **Selection tab**

Figure 45-169. DxDesigner Link Dialog Box, Selection Tab

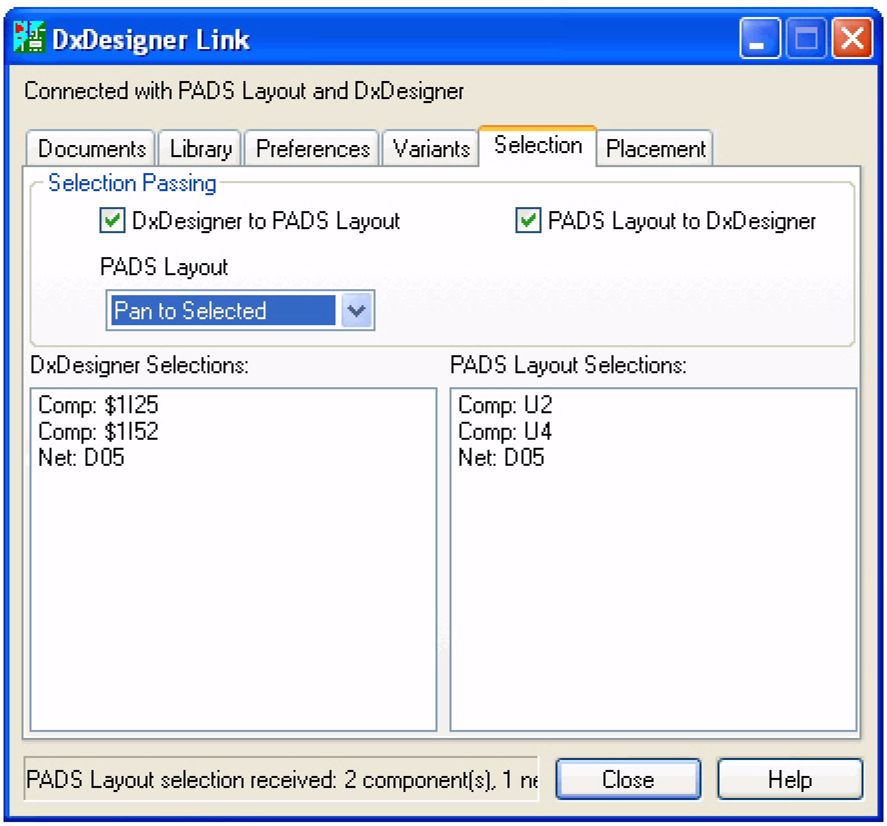


Table 45-155. Selection Tab contents

Name	Description
DxDesigner to PADS Layout	Specifies to allow selection of the PADS Layout object that corresponds to the selected DxDesigner object.

Table 45-155. Selection Tab contents (cont.)

Name	Description
PADS Layout to DxDesigner	Specifies to allow selection of the DxDesigner object that corresponds to the selected PADS Layout object.
PADS Layout	Specifies the zoom level you want to use in PADS Layout: • **None—**Does not change the view when selections are made in the other application. • **Zoom to Selected—**Zooms in on a component when it is selected in the other application. • **Pan to Selected—**Pans to a component when it is selected in the other application.
DxDesigner Selections list	Lists the objects that are selected in DxDesigner.
PADS Layout Selections list	Lists the objects that are selected in PADS Layout.

Related Topics

Cross-Probing with DxDesigner

DxDesigner Link Dialog Box, Variants Tab

The Variants tab of the DxDesigner Link dialog box is used to pass assembly variant information between DxDesigner's Variant Manager and PADS Layout's Assembly Variants feature.

Accessing

- On the Tools menu, click DxDesigner and in the DxDesigner Link dialog box, click the Variants tab.

  Restriction: The Variants Tab is available only once you connect to either the DxDesigner or PADS Layout design.

Figure 45-170. DxDesigner Dialog Box, Variants Tab

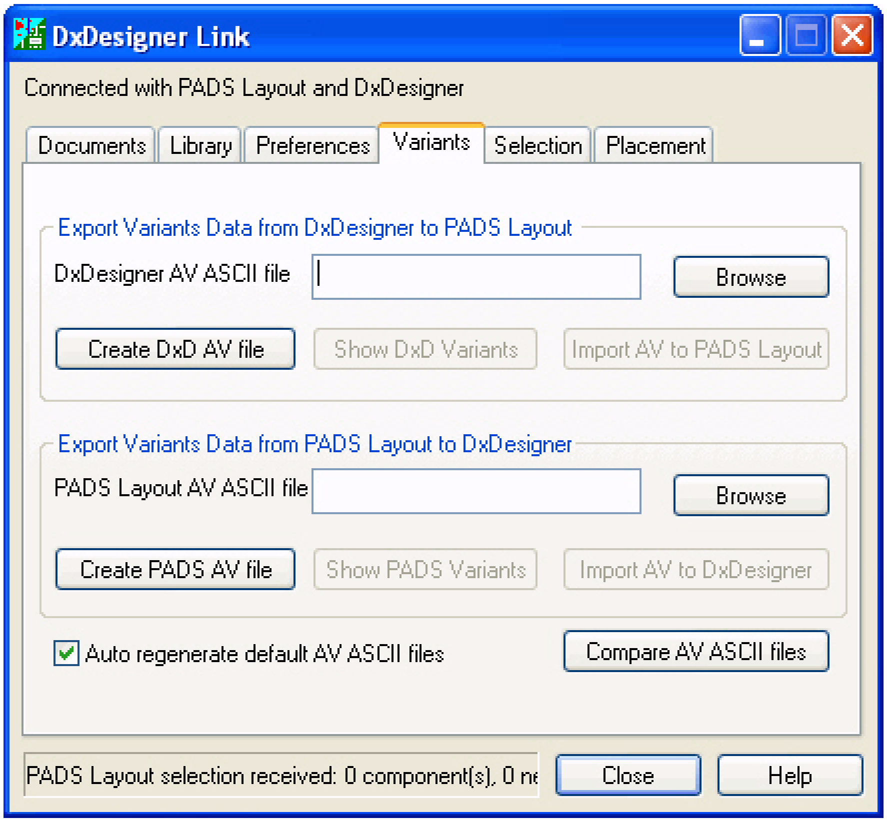


Table 45-156. Variants Tab Contents

Name	Description
DxDesigner AV ASCII file	Type the path to the assembly variant ASCII file created by DxDesigner or use the Browse button and browse to select the file. The file is located at the same location as the DxDesigner .prj file.
Create DxD AV file	Click to create an assembly variant ASCII file from DxDesigner’s Variant Manager. The generated .asc file is stored at the same location as the DxDesigner project .prj file. The path to the file is also added to the DxDesigner AV ASCII file field, but only if an entry is not already in the field. **Restriction:** This button is only available if you are connected to DxDesigner.
Show DxD Variants	Click to view the assembly variant .asc file listed in the DxDesigner AV ASCII file field.

Table 45-156. Variants Tab Contents

Name	Description
Import AV to PADS Layout	Click to import the assembly variant information from the DxDesigner assembly variant ASCII file into PADS Layout. **Restriction:** This button becomes available only when the DxDesigner Link is connected to DxDesigner and the path to the DxDesigner assembly variant ASCII file is added to the DxDesigner AV ASCII file field.
PADS Layout AV ASCII file	Type the path to the assembly variant ASCII file created by PADS Layout or use the Browse button and browse to select the file. The file is located at the same location as the PADS Layout .pcb file.
Create PADS AV file	Click to create an assembly variant ASCII file from the PADS Layout's Assembly Variant feature. The generated .asc file is stored at the same location as the PADS Layout .pcb file. The path to the file is also added to the PADS Layout AV ASCII file field, but only if an entry is not already in the field. **Restriction:** This button is only available if you are connected to PADS Layout.
Show PADS Variants	Click to view the assembly variant .asc file listed in the PADS Layout AV ASCII file field.
Import AV to DxDesigner	Click to import the assembly variant information from the PADS Layout assembly variant ASCII file into DxDesigner. **Restriction:** The button becomes available only when the DxDesigner Link is connected to PADS Layout, and the path to the PADS Layout variant ASCII file is added to the PADS Layout AV ASCII file field.
Auto regenerate default AV ASCII files	Select this check box to automatically regenerate the assembly variant file after importing variant information to either DxDesigner or PADS Layout.
Compare AV ASCII files	Click to compare the DxDesigner assembly variant against the PADS Layout assembly variant. A report file summarizes the differences. **Restriction: The button is functional when the path to both variants are loaded into the AV ASCII file fields for both DxDesigner and PADS Layout.**

Related Topics

DxDesigner's Variant Manager Users Manual

Importing Variant Data to PADS Layout

Importing Variant Data to PADS Layout Manually

Exporting Variant Data to DxDesigner

Comparing Variant Data Files

DXF Export Dialog Box

Use the DXF Export dialog box to export DXF formatted files to AutoCAD2004.

Tip: PADS DXF export attempts to preserve as much of the PADS information as possible. The translator maintains all your defined colors except the background color.

Accessing

- **File** menu > **Export** > **select DXF Files** > **Save**

Figure 45-171. DXF Export Dialog Box

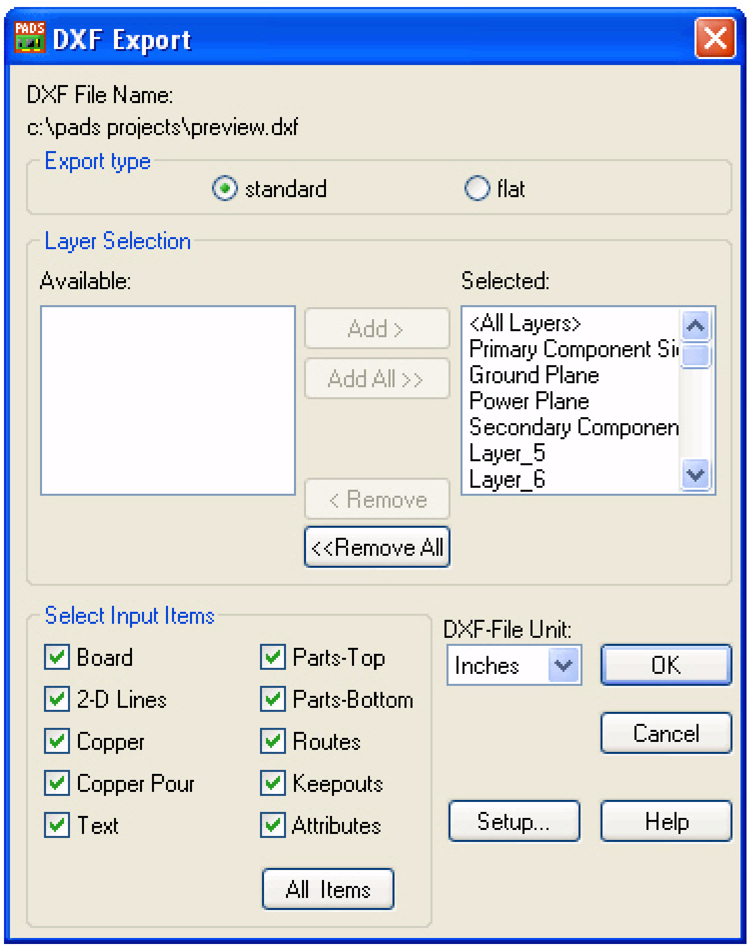


Table 45-157. DXF Export Dialog Box Contents

Name	Description
DXF File Name	The name of the DXF file you are importing.
Export Type	
Standard	Select this option to create an intelligent output that creates blocks, nested blocks, and an extensive layer setup. For example, the objects that make up a decal are made into a complete block. When you select a part of the decal, the whole decal is selected.
Flat	Select this option to create an output that creates a flat result with no blocks, and a basic layer setup. For example, the objects that make up a decal are not arranged in a block and each object can be moved independently. **Restriction:** When you select the flat output option, the Setup DXF Drill Size and Symbols dialog box is unavailable.
Layer Selection area—Select layers to export layer definition information and to permit the export of items on those Selected layers of your PADS Layout design. Items exported from the layer have the PADS Layout layer number appended to its AutoCAD layer name. For example, if you exported text from the top PCB layer, in AutoCAD it would be placed on layer TEXT_01. (Layer 0 or All Layers in PADS Layout is layer 00 in AutoCAD. In the PCB Decal Editor, the Opposite Layer maps to -1 in AutoCAD and the Inner Layers setting maps to -2 in AutoCAD.)	
Available	Lists the layers available for export.
Selected	Lists the layers selected for export.
Add >	Moves the selected layer from the Available list to the Selected list.
Add All >>	Moves all of the layers from the Available list to the Selected list.
< Remove	Moves the selected layer from the Selected list to the Available list.
<< Remove All	Moves all of the layers from the Selected list to the Available list.
Select Input Items Area—Use these check boxes to select the design items you want to export to the layers in the Selected list. These selections are global and apply to all the layers in the selected list.	
Board	Select to export the board outline and cutouts. Standard export outputs a polyline outline of real width to layer BOARD_OUTLINE_00. Creates a Block called BOARD_1 from the outline shapes of the board and any existing cut outs.

Table 45-157. DXF Export Dialog Box Contents (cont.)

Name	Description
2-D Lines	Select to export 2D lines. Standard export outputs polylines of real width to layers 2D_LINE_nn (where nn is the layer of origin in the PCB design).
Copper	Select to export copper objects. Standard export outputs polylines of real width to layers COPPER_nn (where nn is the layer of origin in the PCB design). **Tip:** Copper cut outs are exported as polylines of real width to layers COPPER_CUTOUT_nn (where nn is the layer of origin in the PCB design).
Copper Pour (includes Plane areas)	Select this check box to export copper pours from non-Split/Mixed layers and also Plane area pours from Split/Mixed layers. This exports either hatch outlines or pour outlines depending on the Display mode setting in the Drafting - Hatch and Flood options. • If the Display mode is set to **Pour outline**, selecting this under Standard export outputs copper pour objects as polylines of real width to layers POUR_HEADER_nn (where nn is the layer of origin in the PCB design). • If the Display mode is set to **Hatch outline**, selecting this under Standard export outputs hatch outline objects as polylines of real width to layers POUR_OUTLINE_nn (where nn is the layer of origin in the PCB design). Export includes the following: • Global Hatch and Flood Options: Minimum Hatch Area, Smoothing Radius, View, and Display Mode. • Individual custom Flood & Hatch Options settings if they exist as Copper Pour attributes: Hatch Grid, Smoothing Radius, Hatch Direction, and Flood over Vias. **Tip:** Flood data brought in from PADS can be seen by performing a Hatch using a solid fill.
	Copper Pour Cut Outs (includes Plane area cut outs) Either Pour outline cut outs or Hatch outline cut outs are exported depending on the Display mode setting in the Drafting - Hatch and Flood options. • If the Display mode is set to **Pour outline**, under Standard export, Pour outline cut outs are exported as polylines of real width to layers POUR_VOID_nn (where nn is the layer of origin in the PCB design). • If the Display mode is set to **Hatch outline**, under Standard export, Hatch outline cut outs are exported as polylines of real width to layers POUR_VOID_nn (where nn is the layer of origin in the PCB design).

Table 45-157. DXF Export Dialog Box Contents (cont.)

Name	Description
Text	Select to export text objects. Standard export outputs text to layers TEXT_nn (where nn is the layer of origin in the PCB design). Right-reading text strings are supported. **See also:** Working With Labels
Parts-Top	Select to export parts mounted on the top layer of the PCB. Standard export outputs blocks with the name PART_TOP_x (where x is a consecutive number for each additional part) to layer PART_TOP. The translator exports height information if the part has either of the following properties: • Geometry.Height general attribute • Text string on layer 30 of form $height or $height1 height2 (where height and height1 indicate the component height and height2 indicates the component mounting offset) For details of how keepouts are constructed in AutoCAD, see the Keepouts section below.
	• **Component names**—are exported, under Standard export, as text on layer PART_NAME_TOP. • **Decal names**—are exported, under Standard export, to the SYM_NAME layer. • **Drill symbols**—are exported, under Standard export, to the DRIL_SYMBOL layer. • **Jumper silkscreen**—is exported, under Standard export, as a polyline on layer JUMPER_BOX. • **Outlines/Text**—is exported, under Standard export, as sub blocks on layers BODY_nn (where nn is the layer of origin in the PCB design). • **Pad Stacks**—are exported, under Standard export, as polyline outlines of real width on layers PADS_TOP, PADS_INNER, PADS_INNER_ANTI, PADS_BOT, DRIL_PLTE_THRU_nn, DRIL_NPLTE_THRU_nn, DRIL_PLTE_PRTL_nn, DRIL_NPLTE_PRTL_nn. • **Part 2D lines**—are exported, under Standard export, to layers PART_TOP_2DLINE_nn (where nn is the layer of origin in the PCB design). • **Part types**—are exported, under Standard export, as blocks on layer PART_INFO • **Part type names**—are exported, under Standard export, as text on layer PART_TYPE_TOP • **Thermal reliefs**—are exported, under Standard export, as polyline segments of real width on layers POUR_PADTHERM_nn

Table 45-157. DXF Export Dialog Box Contents (cont.)

Name	Description
Parts-Bottom	Select to export parts mounted on the bottom layer of the PCB. Standard export outputs blocks with the name PART_BOTTOM_x to layers PART_BOTTOM. The translator exports height information if the part has either of the following properties: • Geometry.Height general attribute • Text string on layer 30 of form $height or $height1 height2 (where height and height1 indicate the component height and height2 indicates the component mounting offset) For details of how keepouts are constructed in AutoCAD, see the Keepouts section below.
	• **Component names**—are exported, under Standard export, as text on layer PART_NAME_BOT • **Decal names**—are exported, under Standard export, to the SYM_NAME layer • **Drill symbols**—are exported, under Standard export, to the DRIL_SYMBOL layer • **Jumper silkscreen**—is exported, under Standard export, as a polyline on level JUMPER_BOX • **Outlines/Text**—is exported, under Standard export, as sub blocks on layers BODY_nn (where nn is the layer of origin in the PCB design). • **Pad Stacks**—are exported, under Standard export, as polyline outlines of real width on layers PADS_TOP, PADS_INNER, PADS_INNER_ANTI, PADS_BOT, DRIL_PLTE_THRU_nn, DRIL_NPLTE_THRU_nn, DRIL_PLTE_PRTL_nn, DRIL_NPLTE_PRTL_nn. • **Part 2D lines**—are exported, under Standard export, to layers PART_BOT_2DLINE_nn (where nn is the layer of origin in the PCB design). • **Part types**—are exported, under Standard export, as blocks on layer PART_INFO • **Part type names**—are exported, under Standard export, as text on layer PART_TYPE_nn (where nn is the layer of origin in the PCB design). • **Thermal reliefs**—are exported, under Standard export, as polyline segments of real width on layers POUR_PADTHERM_nn (where nn is the layer of origin in the PCB design).

Table 45-157. DXF Export Dialog Box Contents (cont.)

Name	Description
Routes	Select to export Traces and Vias. Standard export outputs traces as polylines of real width(s) to layers TRACE_nn (where nn is the layer of origin in the PCB design), and vias as blocks to layer VIA.
	• **Connections**—are exported, under Standard export, as lines on layer LINK_nn (where nn is the layer of origin in the PCB design). • **Drill symbols**—are exported, under Standard export, to the DRIL_SYMBOL layer • **Signals**—are exported, under Standard export, as linetype with name and information • **Teardrops**—are exported, under Standard export, as blocks to layers TEAR_nn (where nn is the layer of origin in the PCB design). • **Thermal reliefs**—are exported, under Standard export, as polyline segments of real width on layers POUR_VIATHERM_nn (where nn is the layer of origin in the PCB design).
Keepout	Select to export board keepouts. (Decal keepout are exported with Parts.) Standard export outputs keepouts as closed polylines to layer KEEPOUT_nn (where nn is the layer of origin in the PCB design). Each keepout is also made into a Block with the name KEEPOUT_x (where the x is a consecutive number for each additional keepout). Keepouts are also given a Linetype of KEEPOUT_ABCDEFGHx to store its Restriction settings (where A=Placement, B=Component height, C=Component drill (this can only be used if you assign the keepout to All Layers), D=Trace and copper, E=Copper pour and plane area, F=Via and jumper, G=Test point, H=Accordions, and x=the value for the component height when used in conjunction with A & B. The value is set in database units. For example 1 mil =3810000 database units. Values for ABCDEFGH are either 0 or 1. A value of 0 means a cleared check box and a 1 means a selected check box. The translator uses the Reverse for Keepout Hatch Direction. The translator uses the Keepout hatch grid setting.

Table 45-157. DXF Export Dialog Box Contents (cont.)

Name	Description
Attributes	Select to export the attribute dictionary, individual attributes and value assignments, attribute labels, and attribute status (read only, system, ECO Registered, or hidden). The attribute hierarchy is not exported. Attribute names are exported, under Standard export, to the LABEL_INFO layer. Attribute values are exported, under Standard export, to the LABEL_ATTRIBUTE layer. **Tips:** • DXF supports the increase in reference designator length to 15 characters. • Import is different - Reference designators, part types, attribute labels, and attribute status (read only, system, ECO Registered, or hidden). Jumper names are treated as reference designator labels. **See also:** Working With Labels, Setting Up Jumpers
All Items	Toggles all the check boxes of Input Items on or off.
DXF-File Unit	The unit that will be used in the DXF file.
Setup	Opens the Setup DXF Drill Size and Symbols Dialog Box.

Mapping of General Objects

- **Dimension objects**—are exported to layers DIMENSIONx_nn (where x is the object number, and nn is the layer of origin in the PCB design).
- **General PCB information**—is exported as text to layer PCB_PARAMS
- **Layer information**—is exported as text to layer DBLAYERS (display colors, type of layer, thickness, etc.)

Related Topics

Exporting DXF Files

DXF Format in the *Concepts Guide*

Importing and Exporting Files in the *Concepts Guide*

DXF Import Dialog Box

You can import specialized shapes of DXF format into your decal or into the design using the AutoCAD 2004 DXF format.

Recommendation: There are two interfaces to import DXF files. This interface is optimized for importing RF shapes and only converts into 2D Lines and Copper objects. It allows you to map DXF layers to PADS Layout layers.
If you require more advanced import of any and all design objects, use the other DXF Import dialog box. It imports the design objects from specially named block and layers names in AutoCAD. But its mapping is hard-coded and can't be changed.

Restriction: DXF import only supports the following geometries: POINT, LINE, ARC, CIRCLE, ELLIPS, TRACE, SOLID, 3DFACE, POLYLINE, LWPOLYLINE (AutoCAD R14), and BLOCKS with hierarchy.

Accessing

- **Drafting Toolbar** button > **Import DXF File** button **> Open a .dxf file**

Figure 45-172. DXF Import Dialog Box

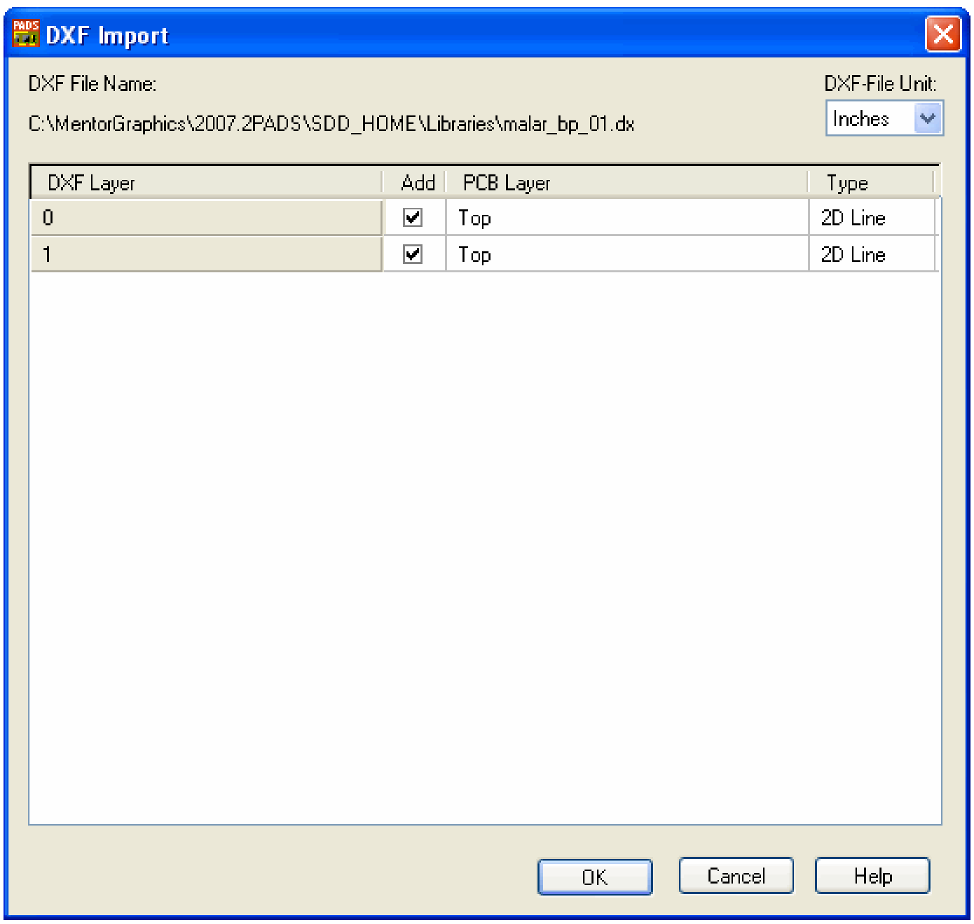


Table 45-158. DXF Import Dialog Box Contents

Name	Description
DXF File Name	The name of the .dxf file you opened.
DXF-File Unit	The units used in this .dxf file: Mils, Metric, Inches.
DXF Layer	Lists the DXF layers available in this .dxf file.
Add	Specifies whether to import this layer.
PCB Layer	Specifies the PCB Layer to which you want to import the DXF items. **Restriction:** A PCB layer set to <All Layers> cannot be imported as copper. You cannot have copper items on <All Layers> in a PCB decal.
Type	Specifies the type of item on the layer: 2D line or copper.

Related Topics

Importing RF Shapes in DXF Format

DXF Format in the *Concepts Guide*

DXF Import Dialog Box

Use the DXF Import dialog box to import files of the AutoCAD 2004 DXF format into the design.

Restriction: Assigning unique colors in AutoCAD may result in translation back to PADS as black if the color is not one of the PADS colors.

Recommendation: There are two interfaces to import DXF files. This interface is designed for advanced import of any and all design objects. It imports the design objects from specially named block and layers names in AutoCAD. This mapping is hard-coded and can't be changed. This allows a whole design, exported from PADS Layout, to be reimported from AutoCAD. If you need to simply import RF shapes into the decal or into the design, use the other DXF Import dialog box. It allows you to map the DXF layers to PADS Layout layers.

Accessing

- **File** menu > **Import** > set file type to **DXF Files (*.dxf)** > select a **DXF File** > **Open**

Figure 45-173. DXF Import Dialog Box

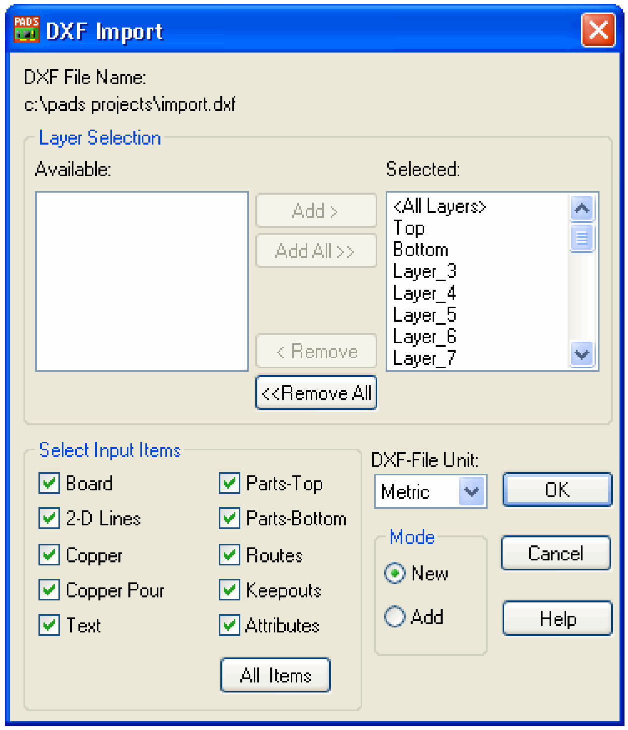


Table 45-159. DXF Import Dialog Box Contents

Name	Description
DXF File Name	The name of the DXF file you are importing.
Layer Selection area—Select layers to accept imported items. Items exported to the layer have the PADS Layout layer number appended to its AutoCAD layer name. For example, if you import text to the top PCB layer, in AutoCAD it is placed on layer TEXT_01. (Layer 0 or All Layers in PADS Layout is layer 00 in AutoCAD. In the PCB Decal Editor, the Opposite Layer maps to -1 in AutoCAD and the Inner Layers setting maps to -2 in AutoCAD.)	
Available	Lists the layers available for import.
Selected	Lists the layers selected for import.
Add >	Moves the selected layer from the Available list to the Selected list.
Add All >>	Moves all of the layers from the Available list to the Selected list.
< Remove	Moves the selected layer from the Selected list to the Available list.
<< Remove All	Moves all of the layers from the Selected list to the Available list.

Table 45-159. DXF Import Dialog Box Contents (cont.)

Name	Description
Select Input Items Area—Use these check boxes to select the design items you want to import to the layers in the Selected list. These selections are global and apply to all the layers in the selected list.	
Board	Select to import the board outline and cutouts. Imports a polyline outline of real width from layer BOARD_OUTLINE_00. The outline shapes of the board and any existing cut outs must be in a Block called BOARD_1. See also: Importing a Board Outline and Cut Out from AutoCAD
2-D Lines	Select to import 2D lines. Imports polylines of real width from layers 2D_LINE_nn (where nn is the layer of origin in the PCB design).
Copper	Select to import copper objects. Imports polylines of real width from layers COPPER_nn (where nn is the layer of origin in the PCB design). **Tip:** Copper cut outs are imported as polylines of real width from layers COPPER_CUTOUT_nn (where nn is the layer of origin in the PCB design).
Copper Pour (includes Plane areas)	Select this check box to import copper pours from non-Split/Mixed layers and also Plane area pours from Split/Mixed layers. • Imports pour outline objects from polylines of real width on layers POUR_HEADER_nn (where nn is the layer of origin in the PCB design). • Imports hatch outline objects from polylines of real width on layers POUR_OUTLINE_nn (where nn is the layer of origin in the PCB design). Import includes the following: • Global Hatch and Flood Options: Minimum Hatch Area, Smoothing Radius, View, and Display Mode. • Individual custom Flood & Hatch Options settings if they exist as Copper Pour attributes: Hatch Grid, Smoothing Radius, Hatch Direction, and Flood over Vias.
	Copper Pour Cut Outs (includes Plane area cut outs) Either Pour outline cut outs or Hatch outline cut outs are imported. • Pour outline cut outs are imported from polylines of real width on layers POUR_VOID_nn (where nn is the layer of origin in the PCB design). • Hatch outline cut outs are imported from polylines of real width on layers POUR_VOID_nn (where nn is the layer of origin in the PCB design).

Table 45-159. DXF Import Dialog Box Contents (cont.)

Name	Description
Text	Select to import text objects. Imports text from layers TEXT_nn (where nn is the layer of origin in the PCB design). Right-reading text strings are supported. **See also:** Working With Labels
Parts-Top	Select to import parts mounted on the top layer of the PCB. Imports blocks with the name PART_TOP_x (where x is a consecutive number for each additional part) to layer PART_TOP. For details of how keepouts are constructed in AutoCAD, see the Keepouts section below.
	• **Component names**—are imported from text on layer PART_NAME_TOP. • **Decal names**—are imported from the SYM_NAME layer. • **Drill symbols**—are imported from the DRIL_SYMBOL layer. • **Jumper silkscreen**—is imported from a polyline on layer JUMPER_BOX. • **Outlines/Text**—is imported from sub blocks on layers BODY_nn (where nn is the layer of origin in the PCB design). • **Pad Stacks**—are imported from polyline outlines of real width on layers PADS_TOP, PADS_INNER, PADS_INNER_ANTI, PADS_BOT, DRIL_PLTE_THRU_nn, DRIL_NPLTE_THRU_nn, DRIL_PLTE_PRTL_nn, DRIL_NPLTE_PRTL_nn. • **Part 2D lines**—are imported from layers PART_TOP_2DLINE_nn (where nn is the layer of origin in the PCB design). • **Part types**—are imported from blocks on layer PART_INFO • **Part type names**—are imported from text on layer PART_TYPE_TOP • **Thermal reliefs**—are imported from polyline segments of real width on layers POUR_PADTHERM_nn

Table 45-159. DXF Import Dialog Box Contents (cont.)

Name	Description
Parts-Bottom	Select to import parts mounted on the bottom layer of the PCB. Imports blocks with the name PART_BOTTOM_x from layers PART_BOTTOM. For details of how keepouts are constructed in AutoCAD, see the Keepouts section below.
	• **Component names**—are imported from text on layer PART_NAME_BOT • **Decal names**—are imported from the SYM_NAME layer • **Drill symbols**—are imported from the DRIL_SYMBOL layer • **Jumper silkscreen**—is imported from a polyline on level JUMPER_BOX • **Outlines/Text**—is imported from sub blocks on layers BODY_nn (where nn is the layer of origin in the PCB design). • **Pad Stacks**—are imported from polyline outlines of real width on layers PADS_TOP, PADS_INNER, PADS_INNER_ANTI, PADS_BOT, DRIL_PLTE_THRU_nn, DRIL_NPLTE_THRU_nn, DRIL_PLTE_PRTL_nn, DRIL_NPLTE_PRTL_nn. • **Part 2D lines**—are imported from layers PART_BOT_2DLINE_nn (where nn is the layer of origin in the PCB design). • **Part types**—are imported from blocks on layer PART_INFO • **Part type names**—are imported from text on layer PART_TYPE_nn (where nn is the layer of origin in the PCB design) • **Thermal reliefs**—are imported from polyline segments of real width on layers POUR_PADTHERM_nn (where nn is the layer of origin in the PCB design)
Routes	Select to import Traces and Vias. Imports traces from polylines of real width(s) on layers TRACE_nn (where nn is the layer of origin in the PCB design), and vias as blocks to layer VIA.
	• **Connections**—are imported from lines on layer LINK_nn (where nn is the layer of origin in the PCB design). • **Drill symbols**—are imported from the DRIL_SYMBOL layer • **Signals**—are imported from linetypes with name and information • **Teardrops**—are imported from blocks to layers TEAR_nn (where nn is the layer of origin in the PCB design). • **Thermal reliefs**—are imported from polyline segments of real width on layers POUR_VIATHERM_nn (where nn is the layer of origin in the PCB design).

Table 45-159. DXF Import Dialog Box Contents (cont.)

Name	Description
Keepout	Select to import board keepouts. (Decal keepout are imported with Parts.) Imports keepouts from closed polylines on layer KEEPOUT_nn (where nn is the layer of origin in the PCB design). Each keepout must also be made into a Block with the name KEEPOUT_x (where the x is a consecutive number for each additional keepout). Keepouts are also given a Linetype of KEEPOUT_ABCDEFGHx to store its Restriction settings (where A=Placement, B=Component height, C=Component drill (this can only be used if you assign the keepout to All Layers), D=Trace and copper, E=Copper pour and plane area, F=Via and jumper, G=Test point, H=Accordions, and x=the value for the component height when used in conjunction with A & B. The value is set in database units. For example 1 mil =3810000 database units. Values for ABCDEFGH are either 0 or 1. A value of 0 means a cleared check box and a 1 means a selected check box. The translator uses the Keepout hatch grid setting.
Attributes	Select to import reference designators, part types, attribute labels, and attribute status (read only, system, ECO Registered, or hidden). Jumper names are treated as reference designator labels. Attribute names are imported from the LABEL_INFO layer. Attribute values are imported from the LABEL_ATTRIBUTE layer. **Tips:** • All attributes are imported with ECO registration off. If the attribute already exists in the attribute dictionary, it is not changed. • DXF supports the increase in reference designator length to 15 characters. • Export is different - Attribute dictionary, individual attributes and value assignments, attribute labels, and attribute status (read only, system, ECO Registered, or hidden). The attribute hierarchy is not exported. **See also:** Working With Labels, Setting Up Jumpers
All Items	Toggles all the check boxes of Input Items on or off.
DXF-File Unit	The unit that will be used in the DXF file. When it is not necessary to set the Unit, this list is unavailable.
Mode area	
New	Specifies to import a DXF file that was exported from PADS Layout. Reads PADS Layout layer and via information in the file.

Table 45-159. DXF Import Dialog Box Contents (cont.)

Name	Description
Add	Specifies to import a native AutoCAD DXF file. Reads 2D line, text, keepout, and copper items in the file. You can also use this option to ignore layer and via information in a DXF file exported from PADS Layout.

Related Topics

Importing DXF Files

DXF Format in the *Concepts Guide*

Importing and Exporting Files in the *Concepts Guide*

ECO Options Dialog Box

Use the ECO Options dialog box to specify whether and how an ECO file will be written when you make engineering (netlist) changes to a design using the ECO toolbar.

Accessing

- **Tools menu** > **ECO Options**

  or

- **ECO Toolbar** button (when you select it the first time in a session)

  or

- **ECO Toolbar** button > **ECO Options** button

Figure 45-174. ECO Options Dialog Box

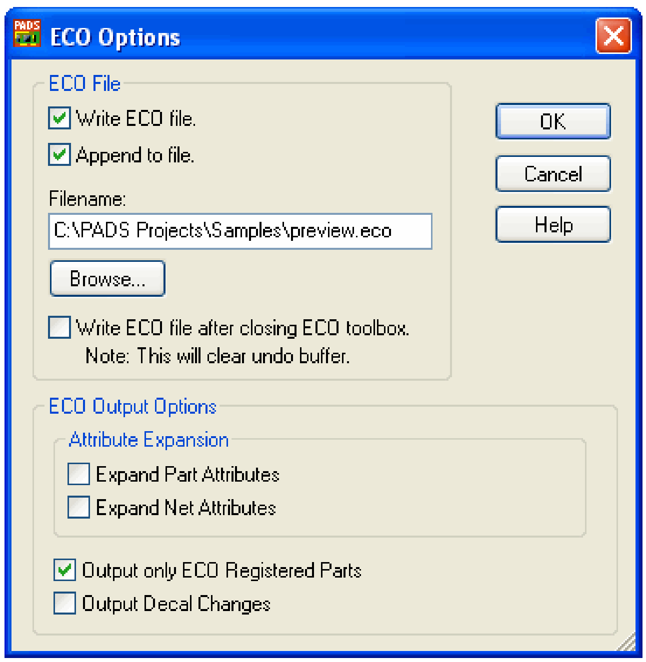


Table 45-160. ECO Options Dialog Box Contents

Name	Description
Write ECO File	Select to record all ECO operations to a file. You can use this file to back-annotate your schematic. **See also:** ECO Options - Check Box Combinations

Table 45-160. ECO Options Dialog Box Contents

Name	Description
Append To File	Select to append ECO operations to the end of an existing .eco file named in the Filename box. Clear to overwrite an existing .eco file named in the Filename box. If the named file does not exist, a new file is created. **See also:** ECO Options - Check Box Combinations
ECO Filename	Type a pathname or click the Browse button to select an existing file.
Write ECO File After Closing ECO toolbar	Select to update the .eco file when you close the ECO toolbar to leave ECO mode. This allows you to review the file immediately, but clears the undo buffer, so undo capabilities are lost. Clear to continue to have undo capabilities after you close the ECO toolbar - until you save the design, when the .eco file is written and the undo buffer is cleared. **See also:** ECO Options - Check Box Combinations
Expand Part Attributes	Records attributes from higher levels in the design hierarchy, such as from the Part Type. It also moves the attributes to the highest level readable by a schematic capture program.
Expand Net Attributes	Records attributes from higher levels in the design hierarchy, such as from the board in the .eco file. It also moves the attributes to the highest level readable by a schematic capture program.
Output Only ECO Registered Parts	Select to specify that objects must be ECO-registered to be included in the ECO file. **See also:** ECO Registration in the *Concepts Guide*

Table 45-160. ECO Options Dialog Box Contents

Name	Description
Output Decal Changes	Select to record decal changes to the ECO file. If this option is selected and if PADS Layout is not in ECO mode: • When you try to right-click and click Edit Decal, you get the error message “*You can modify decals only in ECO mode or if ECO option Output Decal Change is off*.” You must simply open the ECO toolbar to edit the decal. • When you try to change the Pad stack of a decal through the Pin Properties, you get the prompt message “You can modify decal pad stacks only in ECO mode. Continue?” You can edit the pad stack without needing to open the ECO toolbar. **Caution: Decals are switched to alternates using the** Component Properties dialog box **and can be changed outside of ECO mode. An .eco file created by the ECO toolbar will not contain decal changes to alternates. Use the Compare/ECO dialog box to create an .eco file that lists changes to alternate decals.**

Table 45-161. ECO Options - Check Box Combinations

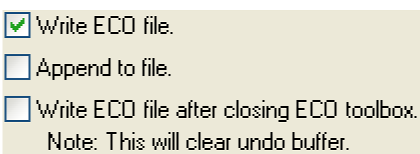
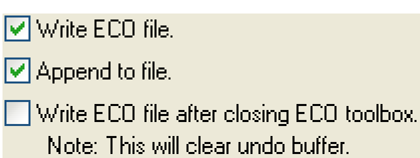
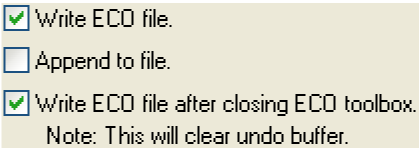
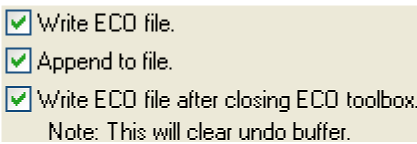
Check Box Combination	Result
	Creates a new .eco file according to the pathname in the Filename box. Overwrites any existing .eco file with the same name at that location. Appends additional ECO changes to the .eco file, despite opening or closing the ECO toolbar, until the changes are saved to the .eco file when: • (the ECO toolbar is closed) and you Save the .pcb file or • you exit the software or open a new design.
	Appends to the .eco file listed in the Filename box. Creates a new .eco file if none exists. Appends additional ECO changes to the .eco file, despite opening or closing the ECO toolbar, until the changes are saved to the .eco file when: • (the ECO toolbar is closed) and you Save the .pcb file or • you exit the software or open a new design.

Table 45-161. ECO Options - Check Box Combinations

Check Box Combination	Result
	Creates a new .eco file according to the pathname in the Filename box. Overwrites any existing .eco file with the same name at that location. Writes changes to the .eco file only when you: • close the ECO toolbar or • exit the software or open a new design.
	Appends to the .eco file listed in the Filename box. Creates a new .eco file if none exists. Writes changes to the .eco file only when you: • close the ECO toolbar or • exit the software or open a new design.

Related Topics

Comparing Designs

Recording ECO Changes

EDC Parameters Dialog Box

Use the EDC Parameters dialog box to define global rules like layer thickness and copper thickness. You can also specify how detailed a design verification report you want.

Accessing

- **Tools menu** > **Verify Design** > **High Speed check** > **Setup button** > **Parameters button**

Figure 45-175. EDC Parameters Dialog Box

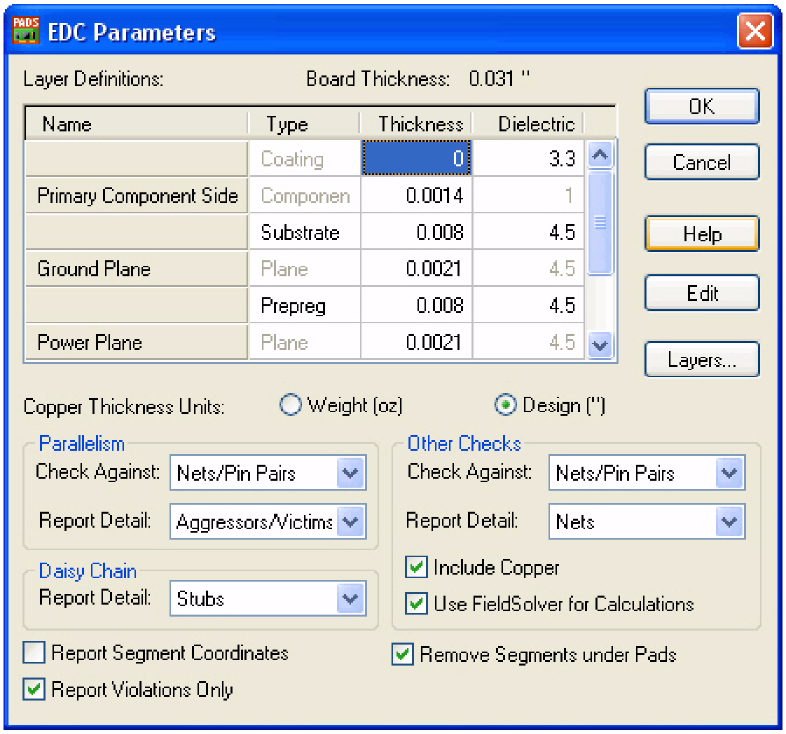


Table 45-162. EDC Parameters Dialog Box contents

Name	Description
Layer Definitions table	• **Name—The name of the layer.** • **Type—The type of layer: Prepeg or Substrate.** • **Thickness—Specifies the required coating.** **Exception:** If no coating is required, set thickness to zero. • **Dielectric—Specifies** the dielectric constant value.
Board Thickness	Displays the total value of material and layer thicknesses in the current design units.
Edit button	Allows you to edit a selected table cell.
Layers button	Opens the Layers Setup dialog box.
Copper Thickness Units	**Specifies the copper thickness unit you want to use:** • **Weight (oz)**—Weight of copper in ounces, per square foot. • **Design Units**—Same unit of measure as the current database unit of measure

Table 45-162. EDC Parameters Dialog Box contents (cont.)

Name	Description
Parallelism Check Against	Specifies the extent of checking. • **Nets/Pin Pairs**—Checks the parallelism and tandem rules against the entire net or pin pair. • **Segments**—Checks the parallelism and tandem rules against only individual segments.
Parallelism Report Detail	Specifies the extent of reporting. • **Net Names Only**—Displays only net names and violations. • **Aggressors/Victims**—Displays specific aggressor and victim nets. • **Segments**—Displays segment coordinates and layers in addition to aggressor and victim nets.
Daisy Chain Report Detail	Specifies the extent of reporting. • **Net Names Only**—Include the number of T points and whether the net is daisy chained. • **Stubs**—Include the group of pins within each stub, the total stub length for each group, the number of T points, and whether or not the net is daisy chained. • **Pin Pairs**—Include the pin to pin length of all pin pairs, the total pin pair length added together to form stubs, the number of T points, and whether the net is daisy chained. • **Segments**—Include the coordinates and layer of all track corners, the pin to pin length of all pin pairs, the total pin pair length added together to form stubs, the number of T points, and the nets being daisy chained.
Other Checks Check Against	Specifies the extent of checking of Length and Delay rules. • **Nets/Pin Pairs**—Check the Length and Delay rules against the entire net or pin pair. • **Segments**—Check the Length and Delay rules against individual segments.
Other Checks Report Detail	Specifies the extent of reporting for capacitance, impedance, delay, and length. • **Nets**—Include the starting and ending pins of nets and net values for capacitance, impedance, delay, and length. • **Pin Pairs**—Include pin-to-pin points, pin pair values, and net values for capacitance, impedance, delay, and length. • **Segments**—Include individual segment coordinates and segment values for capacitance, impedance, delay, and length.
Include Copper	Specifies to include copper polygons with signal names in the capacitance calculations.

Table 45-162. EDC Parameters Dialog Box contents (cont.)

Name	Description
Use FieldSolver for Calculations	Specify to calculate electric parameters of transmission lines such as: impedance, delay (per unit length), and capacitance (per unit length). **See also:** the BoardSim User's Guide.
Report Segment Coordinates	Specifies to include segment coordinates in reports where Segments has been selected in the Report Detail in any one of Parallelism, Daisy Chain, or Other Checks sections.
Report Violations Only	Specifies to list only items that contain violations in the high-speed report.
Remove Segments Under Pads	Specifies to exclude trace segments under pads from calculations. When routing, traces are routed into the middle of pads. This final segment is excluded.

Related Topics

Setting Up EDC Parameters

Edit CAM Document Dialog Box

See: Add/Edit CAM Document Dialog Box

Edit Button Image Dialog Box

Use the Edit Button Image dialog box to create or edit button icons.

Accessing

- **Tools** menu > **Customize** > **Commands tab** > **New button > Select User-Defined Image > New or Edit**

Figure 45-176. Edit Button Image Dialog Box

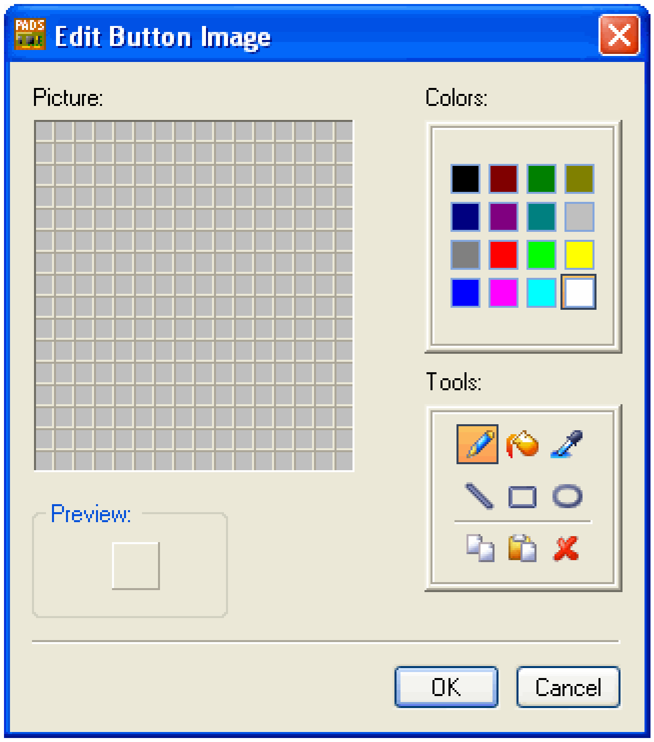


Table 45-163. Edit Button Image Dialog Box Contents

Field	Description
Colors area	Select a color to use with the tools
Tools area	Select a tool to draw/edit the picture or icon of the button

Edit Die Size Dialog Box

Use the Edit Die Size dialog box to change the size of a die.

Restriction: This information applies only to the BGA toolkit.

Accessing

- **BGA Toolbar** button > **Wire Bond Editor button** > **click the BGA** > **Right-click** > **Edit Die Size**

Figure 45-177. Edit Die Size Dialog Box

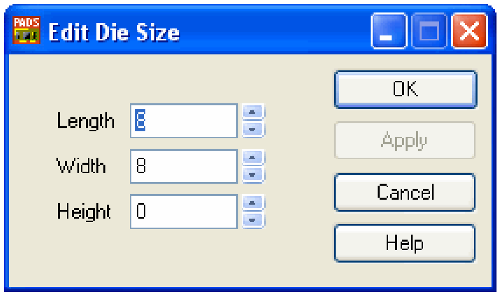


Table 45-164. Edit Die Size Dialog Box contents

Name	Description
Length	Specifies the new length of the die.
Width	Specifies the new width of the die.
Height	Specifies the physical height of the die, for programs that use this value.

Related Topics

To Edit the Die Size

Electrodynamic Check Dialog Box

Use the Electrodynamic Check dialog box to enable high-speed checks for individual nets and classes or for the whole design.

Requirement: You must have specified your plane layers in the Layers Setup dialog box before you run an electrodynamic check. For a two-layer board, temporarily identify one of the layers as a plane layer.

Accessing

- **Tools** menu > **Verify Design** > **High Speed** check > **Setup button**

Figure 45-178. Electrodynamic Check Dialog Box

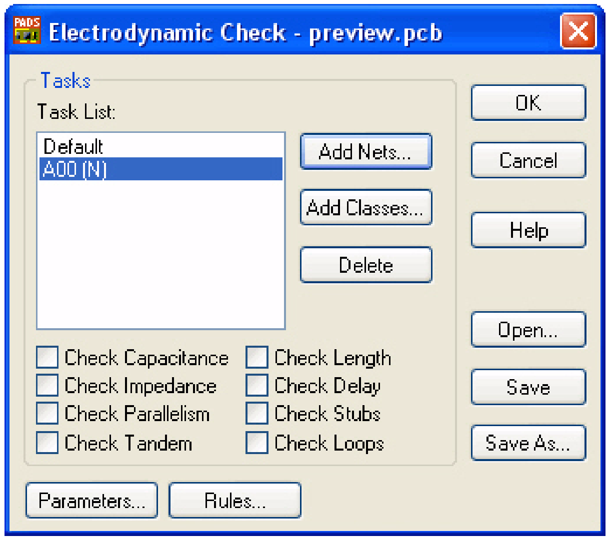


Table 45-165. Electrodynamic Check Dialog Box contents

Name	Description
Task List	Lists the specific checks of certain nets or classes you want to check. You can enable different checks for each items in the Task list. **Tip:** (N) is added to Task list items that are nets and (C) is added to items that are classes.
Add Nets button	Opens the Add Net Tasks dialog box.
Add Classes button	Opens the Add Class Tasks dialog box.
Delete button	Removes the task from the Task List.
Check Capacitance	A function of line length. Capacitance values change when the routed length of a track changes. Use the HiSpeed Rules dialog box to specify minimum and maximum capacitance values.
Check Impedance	Calculated based on track width and copper thickness. Modifying the track width or changing the copper thickness changes the Impedance values. Use the HiSpeed Rules dialog box to specify minimum and maximum impedance values.
Check Parallelism	You can search for tracks on the same layer that may run parallel closely enough or long enough to cause cross talk problems. Use the HiSpeed Rules dialog box to define the maximum permissible parallel lengths, the minimum gap, and if a net is an aggressor or a victim.

Table 45-165. Electrodynamic Check Dialog Box contents (cont.)

Name	Description
Check Tandem	You can search for tracks on different layers that run parallel closely enough or long enough to cause crosstalk problems. Use the HiSpeed Rules or Conditional Rule Setup dialog box to define the maximum permissible tandem lengths, the minimum gap, and if a net is an aggressor or a victim.
Check Length	You can determine mismatched signal lengths. Use the HiSpeed Rules dialog box to specify minimum and maximum lengths.
Check Delay	Calculated based on track length. The delay value changes when the track length changes. Use the HiSpeed Rules dialog box to specify minimum and maximum delay values.
Check Stubs	Stub length is important to proper line termination. Use the HiSpeed Rules dialog box to specify a maximum stub length.
Check Loops	A daisy-chained net has no loops or T-junctions.
Parameters button	Opens the EDC Parameters dialog box.
Rules button	Opens the Rules dialog box.
Open button	Opens the File Open dialog box where you can retrieve settings from an .edp file.
Save button	Stores your settings to an .edp file in the \PADS Projects folder using the same name as your design.
Save As button	Opens the File Save As dialog box where you can store your settings to an .edp file using the location you want and the file name you want.

Related Topics

Setting Up High Speed (Electrodynamic) Checking

Enable/Disable Layers Dialog Box

Use the Enable/Disable Layers dialog box to specify which nonelectrical layers to enable or disable. This shows or hides layers in layer lists - for example, the Layer list on the standard toolbar, or the Layers Setup and Display Colors dialog boxes.

Tip: Switching to increased layer mode does not affect the enabled or disabled status of a layer.

Accessing

- **Setup** menu > **Layer Definition** > **Enable/Disable** button

Figure 45-179. Enable/Disable Layers Dialog Box

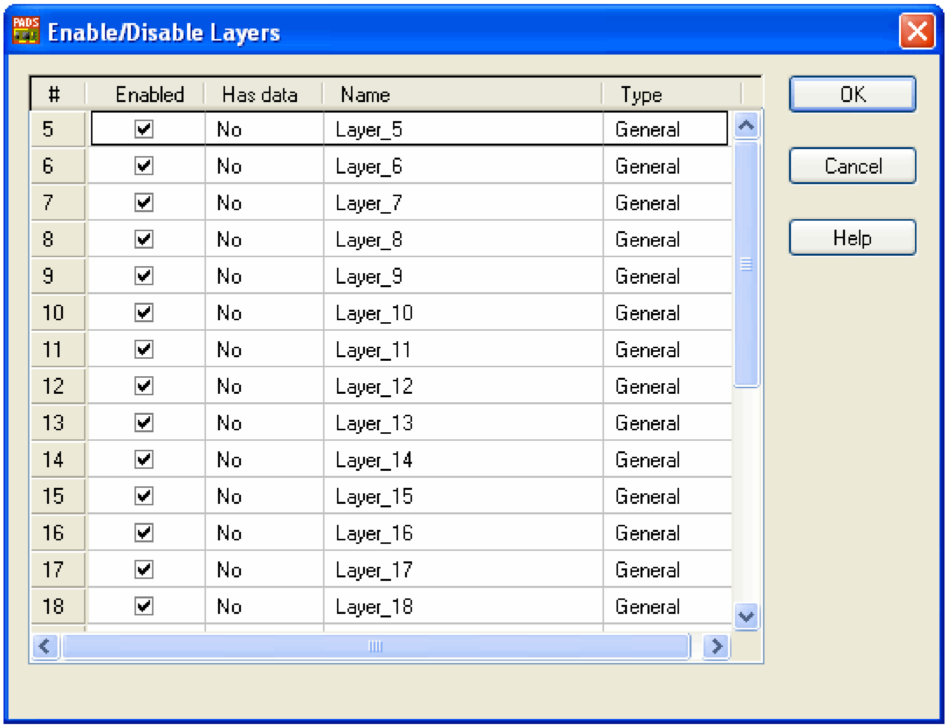


Table 45-166. Enable/Disable Layers Dialog Box Contents

Name	Description
Enabled	Indicates whether the specified layer is enabled.
Has data	Indicates whether the specified layer contains data.
Name	Indicates the name of the specified layer.
Type	Indicates the type of the specified layer.

Related Topics

Enabling or Disabling Non-electrical Layers

Extension Properties Dialog Box

The Extension Properties dialog box displays coordinate information for the selected extension line and provides several areas for modifications.

Click Apply to apply your modifications or Cancel to cancel the changes.

The Extension Properties dialog box remains open until you click OK or Cancel. Selecting another extension line while the dialog box is open updates the information for the selected object.

Accessing

- Select an extension > right-click > **Properties**

Figure 45-180. Extension Properties Dialog Box

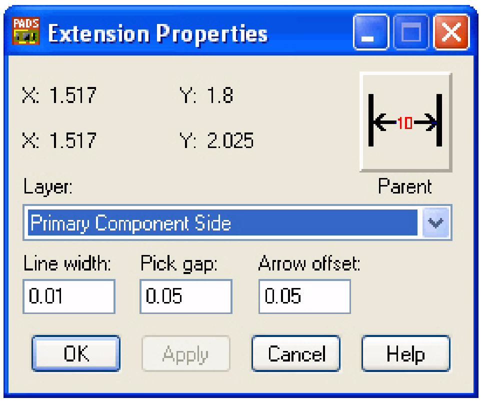


Table 45-167. Extension Properties Dialog Box contents

Name	Description
X and Y	Lists the X and Y coordinate locations of the selected object.
Layer list	Lists the current working layer. Select a new layer from the list.
Line Width	Lists the current line width used for the dimension object. Type a new value to change the line width.
Pick Gap	Lists the current gap between the selection point and the end of the extension line. Type a new value to change the gap.
Arrow Offset	Lists the current length that the extension line extends beyond the arrow. Type a new value to change the length.
Parent button	Opens the Dimension Properties dialog box for the dimension object with which the selected object is associated.

Related Topics

Dimensioning Process

Fabrication Checking Setup Dialog Box

Use the Fabrication Checking Setup dialog box to enable fabrication checks, or to load DFF errors from a preexisting CAM350 database and annotate into the design.

Requirement: You need the CAM350 Link license option to use this. On the Help menu, click Installed Options and on the Options tab, check if the option is available in your license.

Accessing

- **Tools** menu > **Verify Design** > **Fabrication** check > **Setup button**

Figure 45-181. Fabrication Checking Setup Dialog Box

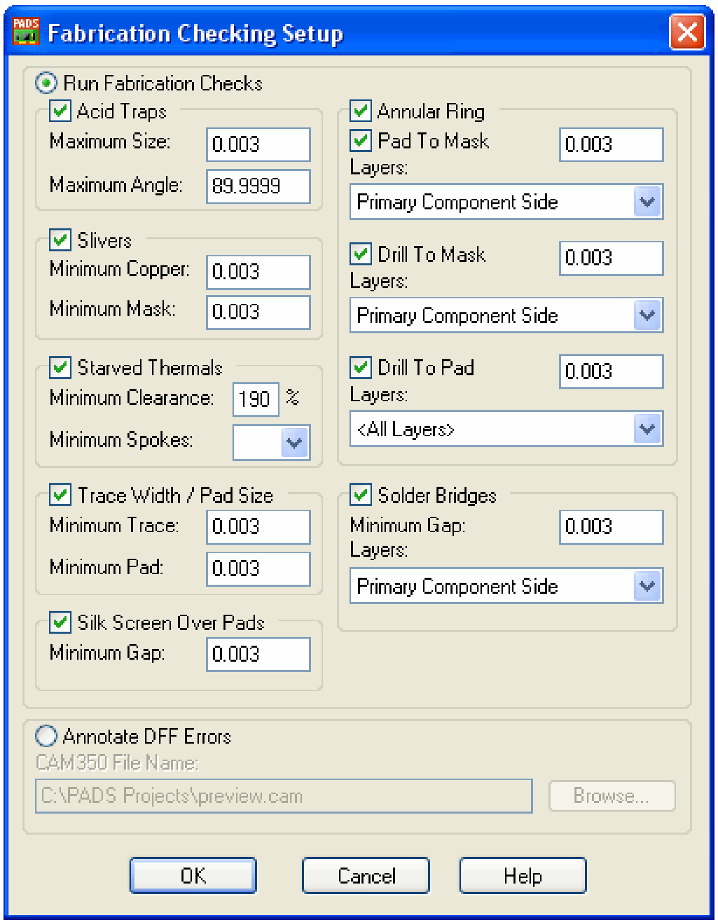


Table 45-168. Fabrication Checking Setup Dialog Box contents

Name	Description
Fabrication Checking Setup	• **Run Fabrication Checks**—Specifies to run the fabrication checks in PADS Layout. • **Annotate DFF Errors**—Specifies to load DFF errors from a CAM350 file. Select this if you use CAM350 for checking your fabrication errors.
Acid Traps	Specifies to flag small areas where acid will pool up. The check is run on all visible electrical layers as defined in the CAM documents.
Maximum Size	Specifies the maximum value of the acid traps to detect. The areas of the “pools” that are flagged will be less than this value.
Maximum Angle	Specifies the maximum angle for traces, pads, or any other data that exists on the layer. Any items that form an angle smaller than this will be flagged as an acid trap.
Slivers	Specifies to flag copper sliver and solder mask sliver areas. This compares the top solder mask layer against the top electrical layer and the bottom solder mask layer against the bottom electrical layer as defined in the CAM documents.
Minimum Copper	Specifies the minimum value for copper slivers. This flags slivers with less area than this value.
Minimum Mask	Specifies the minimum value for solder mask slivers. This flags the slivers with a width less than this value, checking the top and bottom solder mask layers if they are visible.
Starved Thermals	Specifies to flag invalid thermals where adjacent data overlaps the thermal spokes. **Restriction:** Starved Thermals are only checked on (negative) CAM planes.
Minimum Clearance	Specifies the percentage of the thermal's spoke that can be unblocked by another object. Any less of an opening will be considered “starved.”
Minimum Spokes	Specifies the minimum allowable number of the thermal's spokes that cannot be blocked by another object. Any less will be considered “starved.”
Trace Width/Pad Size	Specifies to flag traces and pads that are too small. Checks all electrical layers as defined in the CAM documents.
Minimum Trace	Specifies the minimum trace width value. This flags traces with a width less than this value. This check runs on all visible electrical layers.

Table 45-168. Fabrication Checking Setup Dialog Box contents (cont.)

Name	Description
Minimum Pad	Specifies the minimum pad size. This flags pads with a diameter of less than this value. This check runs on all visible electrical layers.
Silkscreen Over Pads	Specifies to flag silkscreen over pads on top and bottom layers as defined in the CAM documents.
Minimum Gap	Specifies the minimum allowable distance between silkscreen features and a region exposed by solder mask.
Annular Ring	Specifies to flag minimum annular rings on top and bottom layers, comparing electrical, drill, and mask layers.
Pad to Mask	Specifies to flag minimum clearance distances between a pad and its solder mask opening. The offset and annular ring is checked against the specified clearance value. This compares the top electrical layer against the top solder mask layer or the bottom electrical layer against the bottom solder mask layer.
Layers list	Specifies the layer to use for checking.
Drill to Mask	Specifies to flag minimum clearance distances between a drill and its solder mask opening. The offset and annular ring is checked against the specified clearance value. This compares the top drill layer against the top solder mask layer or the bottom drill layer against the bottom solder mask layer.
Layers list	Specifies the layer to use for checking.
Drill to Pad	Specifies to flag minimum clearance distances between a drill and its associated pad. The offset and annular ring is checked against the specified clearance value. This check is run on each specified layer.
Layers list	Specifies the layer to use for checking.
Solder Bridges	Specifies to flag solder mask bridging. Solder can bridge and cause a connection to an adjacent object within the same mask opening. If the adjacent object is farther from the pad than this distance, even if it is exposed by the mask layer, it will not be identified as a bridge. This compares the top solder mask layer against the top electrical layer or the bottom solder mask layer against the bottom electrical layer as defined in the CAM documents.
Minimum Gap	Specifies the minimum clearance value.
Layers list	Specifies the layer to use for checking.

Table 45-168. Fabrication Checking Setup Dialog Box contents (cont.)

Name	Description
CAM350 File Name	Specifies the .cam path and file name of a file to back-annotate DFF errors into PADS Layout for design verification. **Tip:** Click Browse to navigate to the location.

Related Topics

Setting Up Fabrication Checking

Fanout Rules Dialog Box

Use the Fanout Rules dialog box to specify fanout rules. Use fanouts to make routing easier and to ensure that connections are made. PADS Router optimally places fanout vias and routes from vias to corresponding pins for ordered components or separate pins. Fanouts are most useful for some complex SMDs, like BGAs.

Restrictions

- While you can define these rules in either PADS Layout or PADS Router, the rules are used only in PADS Router.
- In PADS Layout, you can define fanout rules for only SOIC/QUAD type components. Use PADS Router to also define fanout rules for BGA and staggered BGA fanout patterns.

Accessing

- **Setup** menu > **Design Rules** > **Default** button > **Fanout** button
-

Figure 45-182. Fanout Rules Dialog Box

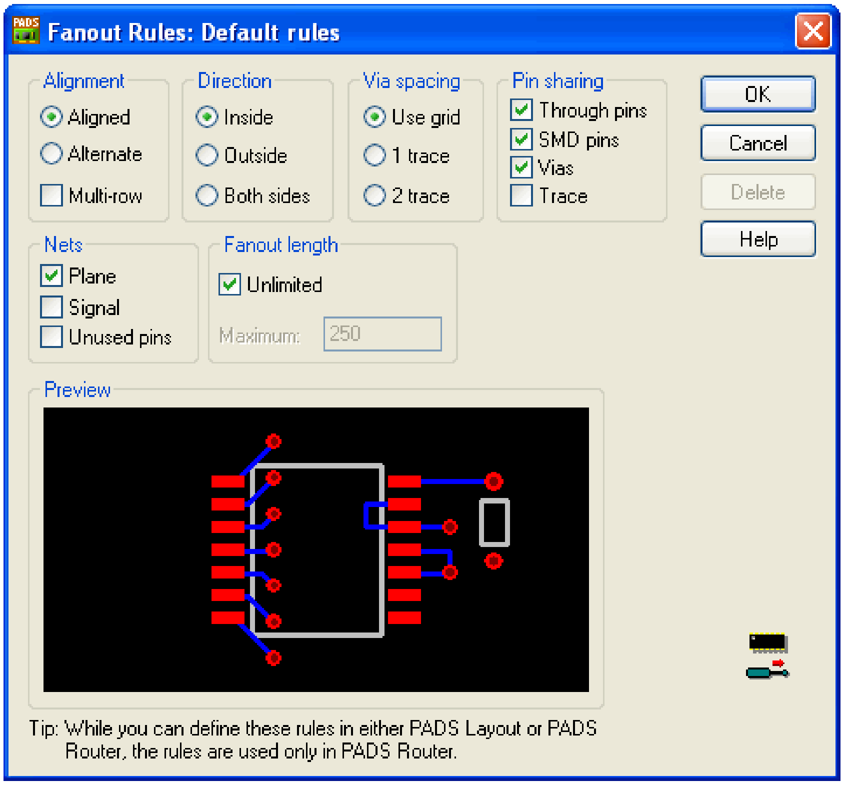


Table 45-169. Fanout Rules Dialog Box

Area	Description
Alignment	**Specify the via placement:** • **Aligned**—Aligns fanout vias on the PADS Router fanout grid. • **Alternate**—Staggers the direction of fanout vias. For example, the first pin fans out to the left and the second pin fans out to the right, and the pattern repeats. • **Multi-row** check box—Specifies to create two rows of vias on each side of the component that has pins.
Direction	Specify where you want to locate the fanout vias, relative to the component outline: • **Inside**—All fanout vias are located inside the component outline. • **Outside**—All fanout vias are located outside of the component outline. • **Both Sides**—Fanout vias are located both inside and outside of the component outline.

Table 45-169. Fanout Rules Dialog Box (cont.)

Area	Description
Via Spacing	Specify the fanout via spacing you want: • **Use Grid**—Place fanout vias on the fanout grid. • **1 Trace**—Width of one trace. This sets a via grid that enables PADS Router to route one wire between adjacent vias. • **2 Trace**—Width of two traces. This sets a via grid that enables PADS Router to route two wires between adjacent vias.
Pin Sharing	Specify all the connections on the same net that PADS Router can use while routing from the pin to the escape via: • **Through pins**—Connect to a through pin if the cost is lower than the cost to use a via. • **SMD pins**—Connect to an SMD pin if the cost is lower than the cost to use a via. If this is disabled, PADS Router routes each SMD pad directly to a pin or via. • **Vias**—Connect to a shared via. If this is disabled, PADS Router creates unique vias for each surface-mount pad. • **Trace**—Connect to a shared trace, which creates a T-Junction.
Nets	Specify the types of nets where fanouts can be created: • **Plane**—Create fanouts for pins belonging to plane nets. • **Signal**—Create fanouts for pins belonging to signal nets. • **Unused pins**—Create fanouts for pins that do not belong to signal nets or plane nets.
Fanout length	**Specify whether or not to restrict fanout lengths:** • **Unlimited**—Specifies that you do not want to restrict fanout lengths. • **Maximum**—Specifies the restriction to the fanout length if you clear the Unlimited check box.
Preview	Shows the fanout layout based on your selections.
Delete button	Removes non-default fanout rules at the current level of the rules hierarchy. **Restriction:** You cannot delete the Default fanout rules.

Tip: Fanning out an entire design is not generally recommended because it creates many vias. However on component-dense PCBs requiring many layers, it may be necessary to fan out all SMD components.

Related Topics

Design Rule Hierarchy in the *Concepts Guide*

Find Dialog Box

Use Find to find and select single or multiple objects by reference designator, part type, line width, or other attributes.

Find works two ways, depending on your selection mode:

- **Select mode** — Find ignores the Selection Filter settings and selects whatever you ask it to.
- **Verb mode** — Find only looks for items that are logical for verb mode.

Accessing

- **Edit** menu > **Find**

Figure 45-183. Find Dialog Box

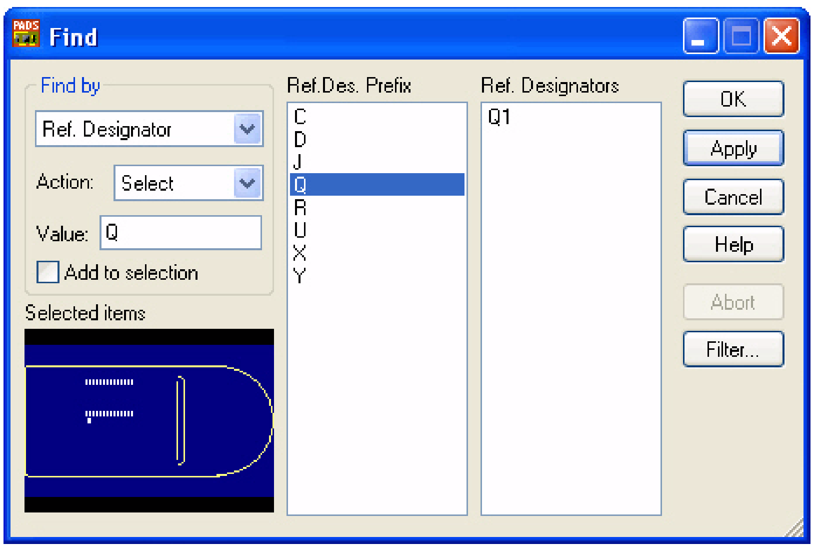


Table 45-170. Find Dialog Box Contents

Name	Description
Object Type list	Specifies how you want to search. You can find by: • reference designators • nets • line widths • part types • pin pairs • via types • decals • net classes • groups • physical design reuses • clusters • unions • pour areas • isolated pours • pad sizes • thermal attributes (custom pad stack thermals) • jumper vias • keepouts • attributes • label fonts • text fonts • nets with bridges • test point types
Action	Specifies the action to perform on items you find. • Select • Highlight • Unhighlight • Rotate 90 • Flip Side • Move Sequential **Exception:** You cannot use Rotate 90, Flip Side, or Move Sequential on test points. **Tip:** When you select objects with the Find dialog box, the shortcut menus change to the relevant commands for modifying the items.
Value	Narrows the search by the value you type. You can use wildcards or expressions.

Table 45-170. Find Dialog Box Contents (cont.)

Name	Description
Add to Selection	Selects the items that you find. Each selected item appears in the Selected Items preview area. **Tip:** When you select objects with the Find dialog box, the shortcut menus change to the commands relevant for modifying the items.
Selected Items Preview area	Displays the selected items.
Find lists	The content and title of these lists change depending on what you select from the Find By list. For example, if you are finding by reference designator, the first list displays reference designator prefixes. When you select a reference designator prefix to search by, for example D, the second list displays all reference designators with the D prefix. You can further limit the search by choosing a specific D reference designator prefix to search by, such as D2.
Abort	Specifies that you want to cancel the find process if it is taking a long time.
Filter	Opens the Selection Filter.

Related Topics

Finding Objects

Using the Find Dialog Box During Placement

Find in Vault Dialog Box

Use the Find in Vault dialog box to search the vault. You can search for projects, archives, folders, or all three.

Tip: Click any item in the Find results list to select it in the vault view tree.

Searches

- Searches are not word-based; a search for the string “ic” will find Iceland, picnicking, and alphanumeric.
- Wild cards are not supported.
- Searches are not case-sensitive.
- A null string in a field returns items with any value.
- Items must match all fields to satisfy the search criteria.
- Both the Name and ID attributes of items are searched for the search string specified in the Name field.

Accessing

- **Vault view** > **Find in Vault** button

Figure 45-184. Find in Vault Dialog Box

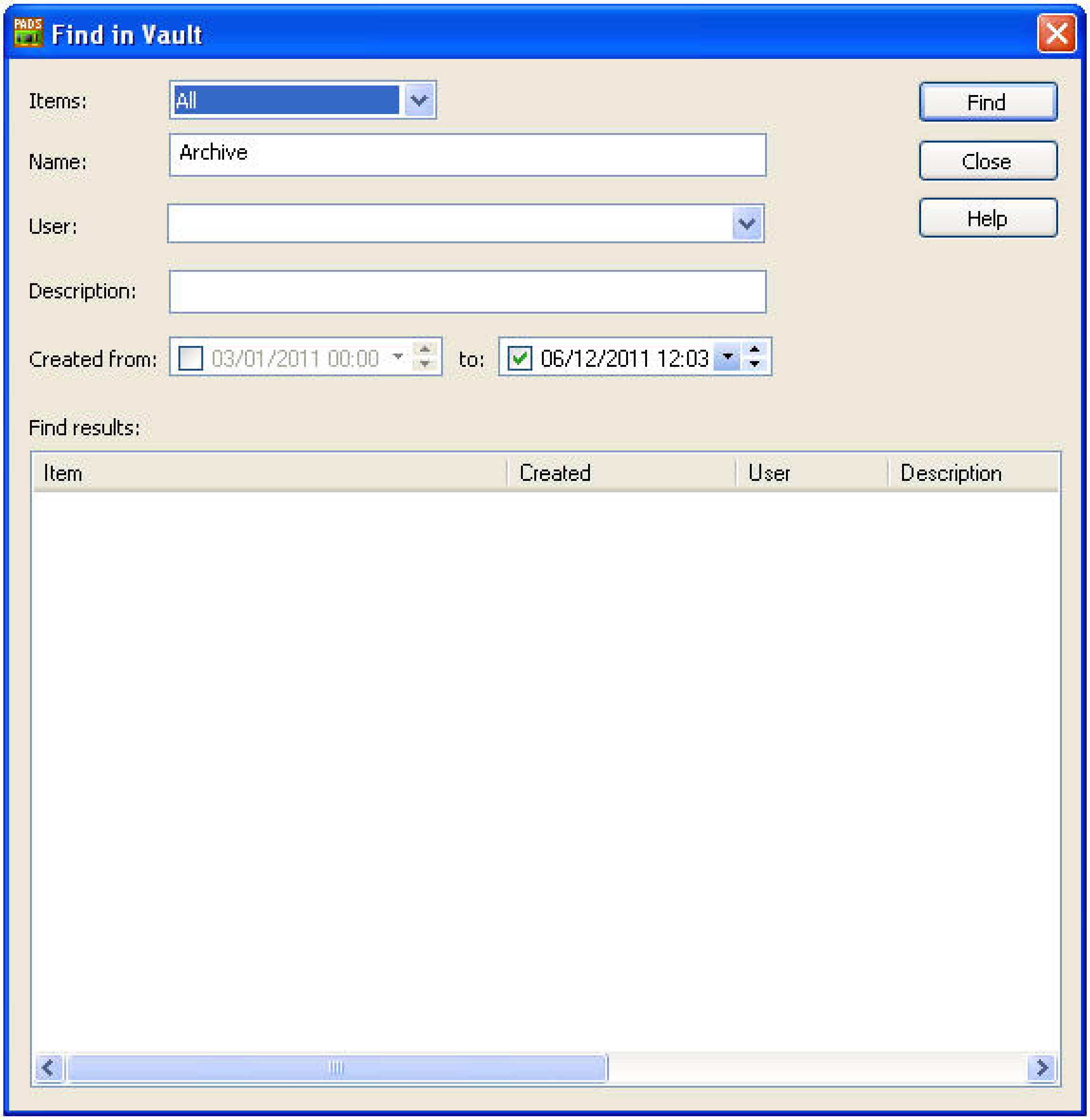


Table 45-171. Find In Vault Dialog Box Contents

Name	Description
Items	Specifies the type of items to find—Projects, Archives, Folders, or All
Name	Specifies to find items whose Name and/or ID contains the entered search string. A null string returns items with any name.

Table 45-171. Find In Vault Dialog Box Contents

Name	Description
User	Select a user to find items whose User contains that user name. If no user is selected, items with any user are returned.
Description	Specifies to find items whose Description contains the entered search string. A null string returns items with any description or none.
Created from ... to	Specifies to find items by creation date/time, as follows: • **Created from**—Finds items created after the specified date/time. • **to**—Finds items created before the specified date/time. If both are specified, finds items created between the two dates/times.
Find button	Initiates the find operation.
Find results	Lists the search results. Double-click an item to close the dialog box and select the item in the vault view tree.

Related Topics

Finding Projects or Archives in the Vault

Flood and Hatch Options Dialog Box

Use the Flood and Hatch Options dialog box to establish unique display settings for copper pour or split/mixed plane areas.

Accessing

Restriction: You must display the pour or plane area's outline before you assign unique display settings. For copper pour areas, select the area, use the modeless command PO. For split/mixed plane areas, use the modeless command SPO.

- Select edge of pour or plane area > right-click > **Select Shape** > right-click > **Properties** > **Options** button

Figure 45-185. Flood & Hatch Options Dialog Box

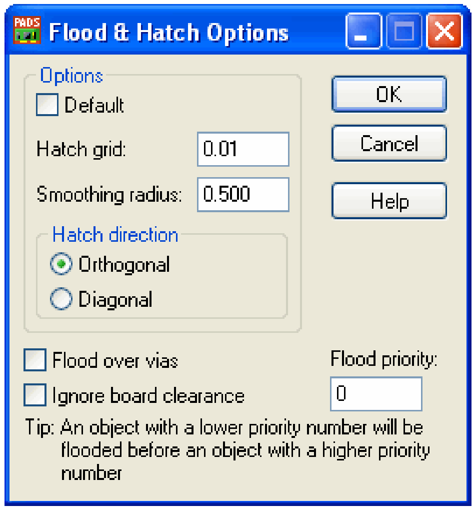


Table 45-172. Flood & Hatch Options Dialog Box contents

Name	Description
Options area	**Default check box**—Specifies to use the default settings as found on the Grids and Drafting / Hatch and Flood pages of the Options dialog box. Clear this check box to access the hatch settings. **Hatch grid**—Specifies the distance between hatch lines. **Smoothing radius**—Specifies the resolution of arced corners. A high smoothing radius value produces smoother, rounder corners. **Hatch direction**—Specifies the hatch line orientation.
Flood over vias	Specifies to flood over any vias within the copper pour or plane area that are part of the same net. **Tips:** • When this setting is changed and applied to the selected object, the changed setting becomes the default for all newly-created plane areas and copper pours. Existing objects (except the selected one) are not affected. • When you flood over vias, a Thermal Relief Error report, therm.err, is generated and automatically appears in the default text editor. The report lists the vias that use different flood over settings than those established in the Thermals tab.
Ignore board clearance	Specifies to ignore the board-to-copper clearance setting and have the copper pour flood outside the board area.

Table 45-172. Flood & Hatch Options Dialog Box contents (cont.)

Name	Description
Flood priority	Specifies to add a flood priority to the selected object. An object with a lower priority number will be flooded before an object with a higher one. Copper pour or plane areas on different layers are processed independently. **See also:** Setting Flooding Order of Overlapping Copper Pour and Plane Areas, Copper Pour and Plane Area Flood Priorities in the *Concepts Guide* **Requirements:** • Type a value from 0 to 250. • If you create an embedded plane, the smaller inner plane must have a lower flood priority than the larger outer plane or the smaller plane area will be flooded over by the outer plane. **See also:** To Create an Embedded Plane

Related Topics

Creating a Copper Pour Area

Creating a Plane Area

Flooding Over Vias in a Copper Pour or Plane Area

Font Replacement Dialog Box

Use the Font Replacement dialog box to manage how missing fonts are replaced in your design.

Accessing

When you open a design created with fonts that are not installed on your system, the Font Replacement dialog box opens automatically.

Tip: If the design uses fonts or character sets that are not installed on your system, empty boxes will appear where you expect to find text or symbols. Once the font replacement process completes, the symbols display properly.

Figure 45-186. Font Replacement Dialog Box

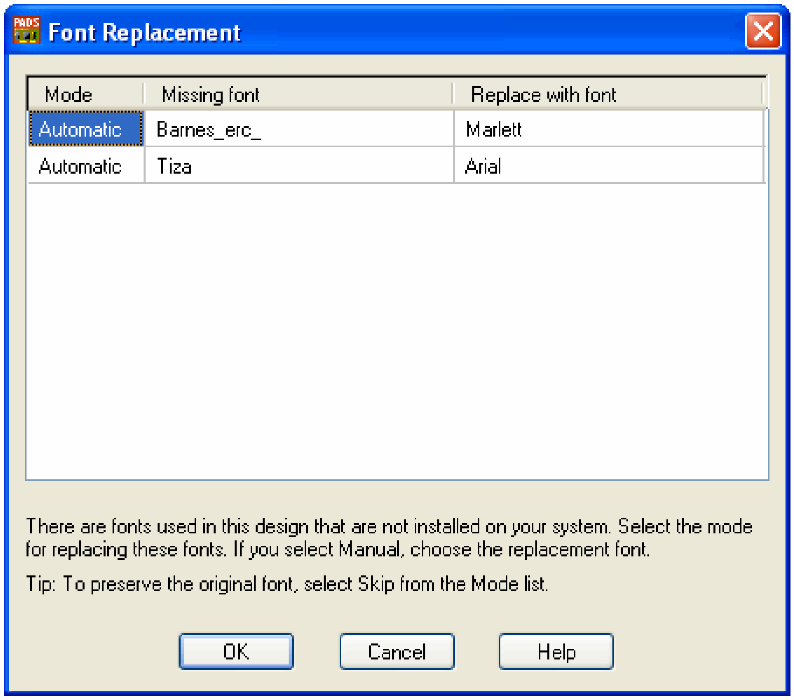


Table 45-173. Font Replacement Dialog Box Contents

Name	Description
Mode	Specifies the mode to replace this font. • **Automatic**—Specifies to replace the font automatically with the one selected by PADS Layout. • **Manual**—Specifies to replace the font with one you select from the Replace with font list. • **Skip**—Specifies to preserve the original font.
Missing font	The name of the font in this design that is missing from your system.
Replace with font	If you chose Manual, lists the fonts available for you to replace the missing font. If you chose Automatic, lists the font PADS Layout chose to replace the missing font.

Tips:

- You can select some fonts for automatic replacement, select others for manual replacement, and choose that other font replacements be skipped entirely.
- You can have a combination of stroke font and system fonts within the same design.

- You must set up fonts for each text string and/or label you create in your design. Once you set up fonts for a text string or label, you can then use the Properties dialog to apply a font and font characteristics to all objects that you select for modification with the Properties dialog box.

Restrictions:

- If the design uses fonts or character sets that are not installed on your system, a font substitution process begins automatically when the file is loaded. During this process, you are asked to choose fonts to substitute for those that are missing from your system.
- System font text is supported in RS274X Gerber format when Fill mode is on. System font text is output to Gerber format as a set of filled polygons.
- System fonts are not supported in the RS-274D CAM output format. If you attempt to use this format with system fonts, the program displays a warning message. If you proceed, system fonts will not be output. Instead, you should use the 274X format with system fonts.
- Type 1 fonts are not supported.

Related Topics

Replacing Fonts

Finding Fonts

Forward Annotation Dialog Box

In DxDesigner Link, forward annotation sends data from a DxDesigner schematic to a PADS Layout design file and updates the PADS Layout design to match the schematic.

Use the Forward Annotation dialog box to filter the schematic data used in forward annotation.

Accessing

- **Tools** menu > **DxDesigner > Forward to PCB button**

Figure 45-187. Forward Annotation Dialog Box

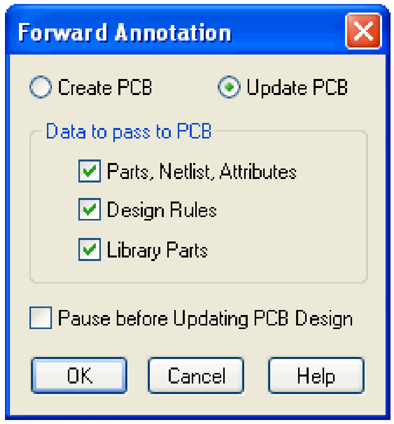


Table 45-174. Forward Annotation Dialog Box contents

Name	Description
Forward Annotation	• **Create PCB**—Specifies that the PCB design in PADS Layout is new. • **Update PCB**—Specifies that you want to update the existing PADS Layout design.
Data to pass to PCB area	DxDesigner Link includes the selected data in an ASCII file (.asc) and uses it in the forward annotation operation. You can send: • Parts, netlist, and attributes names and values • Design rules • Library parts information **Tip:** To include design rules in the forward annotation operation, select the Design Rules option, even if you already selected **Compare Design Rules** on the Preferences tab.
Pause before Updating PCB Design	Specifies that you want to review the ECO file containing changes that the update will make to the PADS Layout design.

Related Topics

Forward Annotating from DxDesigner to PADS Layout

From SPECCTRA Dialog Box

Use the From SPECCTRA dialog box to translate design data modified by SPECCTRA back into a .pcb design file.

Accessing

- Use Windows Explorer to navigate to your ...\SDD_HOME\Programs directory, and double-click **pads2sp.exe > From SPECCTRA button**

Figure 45-188. From SPECCTRA Dialog Box

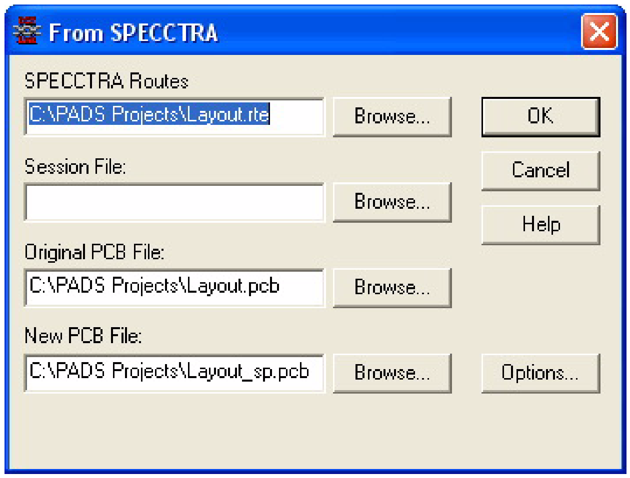


Table 45-175. From SPECCTRA Dialog Box contents

Name	Description
SPECCTRA Routes	Specifies the routing information file that is returned by SPECCTRA after processing. Include the command to write this file after autorouting at the end of the .do file. **Tip:** Click Browse to locate the file.
Session File	Specifies the placement and routing information file. You do not need to supply this file name if you did not use any of the SPECCTRA placement capabilities. **Tip:** Click Browse to locate the file.
Original PCB File	Specifies the original (source) .pcb file. **Tip:** Click Browse to locate the file.
New PCB File	Specifies the file to be created from the SPECCTRA file. **Tip:** Click Browse to locate the file.
Options button	Opens the Options dialog box.

Related Topics

Translating Design Data from SPECCTRA to PADS Layout

Generate Drafting Shape Dialog Box

You can use the outline of a terminal as the basis to create new drafting shapes.

Accessing

- **Tools** menu > **PCB Decal Editor** > Select terminal > Right-click and click **Generate Drafting Shape**

Figure 45-189. Generate Drafting Shape Dialog Box

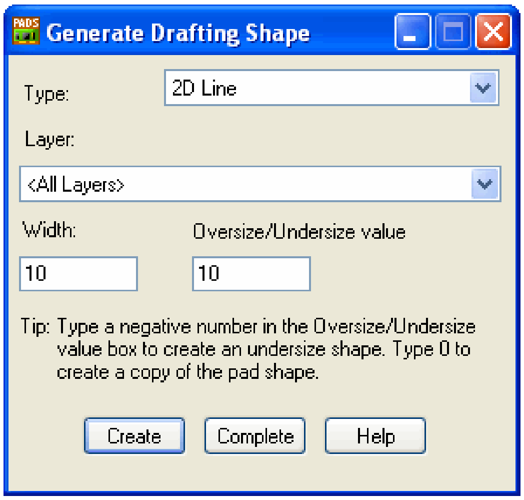


Table 45-176. Generate Drafting Shape Dialog Box Contents

Name	Description
Type	Lists the type of drafting shapes available to generate.
Layer	Lists the layers available for placement of the shape.
Width	sets the line width of the new shape.
Oversize/Undersize value	Sets the size of the new shape: • To create a new drafting shape larger than the terminal outline by the typed value, type a positive number. • To create a new drafting shape equal in size to the terminal, type 0 (zero). • To create a new drafting shape smaller than the terminal outline by the typed value, type a negative number.
Create	Completes this new shape and keeps the dialog box open for the creation of new shapes.
Complete	Completes this new shape and closes the dialog box.

Related Topics

Generating Drafting Shapes from Terminals

Get Drafting Item from Library Dialog Box

Use the Get Drafting Item from Library dialog box to add a drafting item into your design from the library. You can also save a drafting item to a library.

Accessing

- **Drafting Toolbar** button > **From Library** button

Figure 45-190. Get Drafting Item from Library Dialog Box

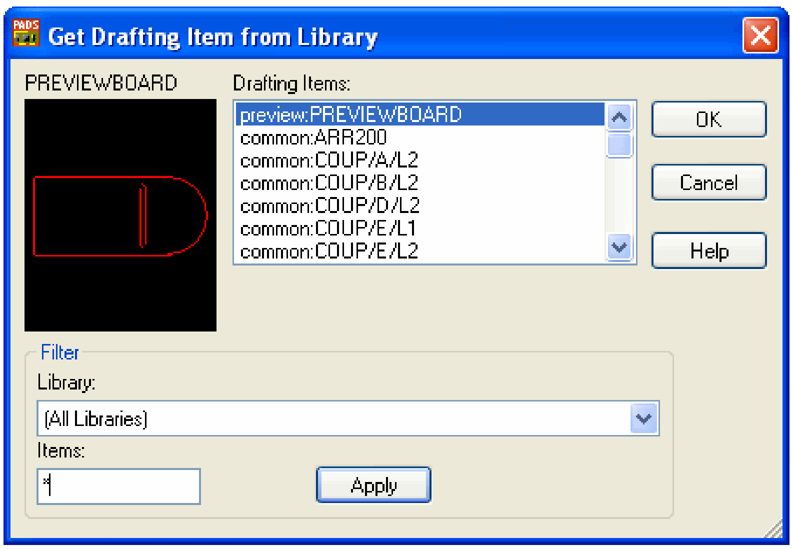


Table 45-177. Get Drafting Item from Library Dialog Box contents

Name	Description
Preview area	Shows the selected drafting item.
Drafting Items	Lists the drafting items in the selected library. The number of objects that appear depends on the filter settings.
Library list	Specifies the library you want to use.
Items	Narrows the search. You can use wildcards or expressions. An asterisk (*) displays all parts in the list.
Apply button	Searches the library for the specified item.

Related Topics

Adding Drafting Items from a Library

Get Part Type from Library Dialog Box

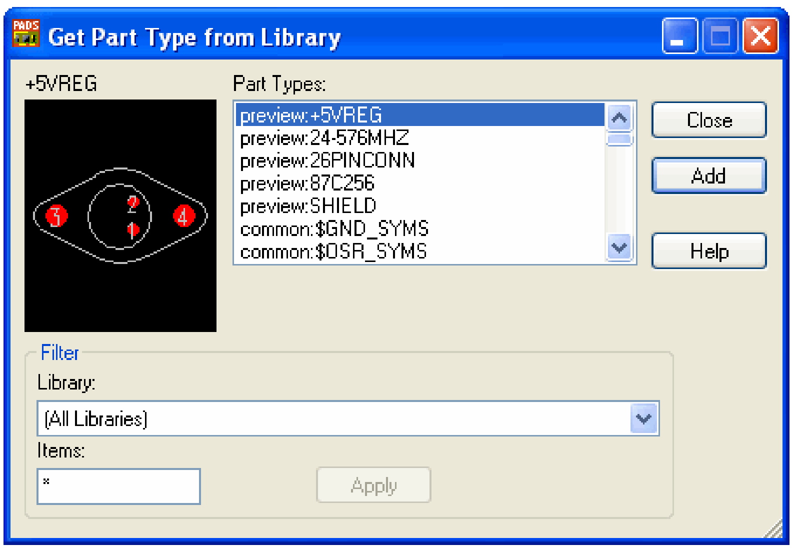
Use the Get Part Type from Library dialog box to Retrieve parts from a library when you add or update parts.

The dialog box is slightly different depending on if you are adding or updating parts.

Accessing

- **ECO Toolbar** button > **Add Component** button

Figure 45-191. Get Part Type from Library Dialog Box



or

- **ECO Toolbar** button > Select a part > **Change Component** button

Figure 45-192. Get Part Type from Library Dialog Box - Change Component

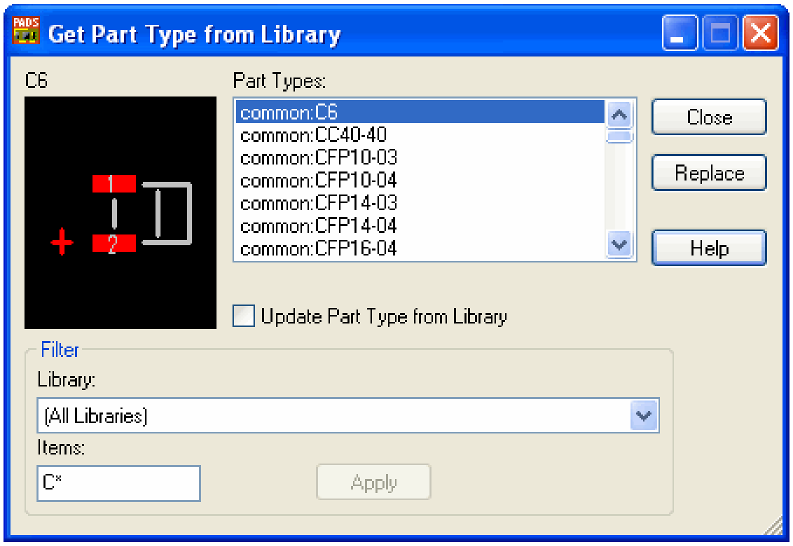


Table 45-178. Get Part Type from Library Dialog Box contents

Name	Description
Preview area	Shows the selected part type.
Part Types	Part search results based on the Filter Settings appear in this list where you can select the part you want to use. The viewer to the left of this window displays the decal of each part as you select them.
Add button	Adds the selected party type to the design. **Restriction:** Available for the Add Part command only.
Replace button	Replaces the part type selected in the design with the part type selected in the Part Types list. **Restriction:** Available for the Change Part command only.
Update Part Type from Library	Updates the part type data for the selected parts based on the part types in the library. **Restriction:** Available for the Change Part command only.
Library list	Specifies the library you want to use.
Items	Narrows the search. You can use wildcards or expressions. An asterisk (*) displays all parts in the list.
Apply button	Searches the library for the specified item.

Related Topics

Adding a Component in ECO Mode

Changing a Component in ECO Mode

Get PCB Decal From Library Dialog Box

Use the Get PCB Decal from Library dialog box to open the decal you want to edit.

Accessing

- **Tools** menu > PCB **Decal Editor > Open button**

Figure 45-193. Get PCB Decal from Library Dialog Box

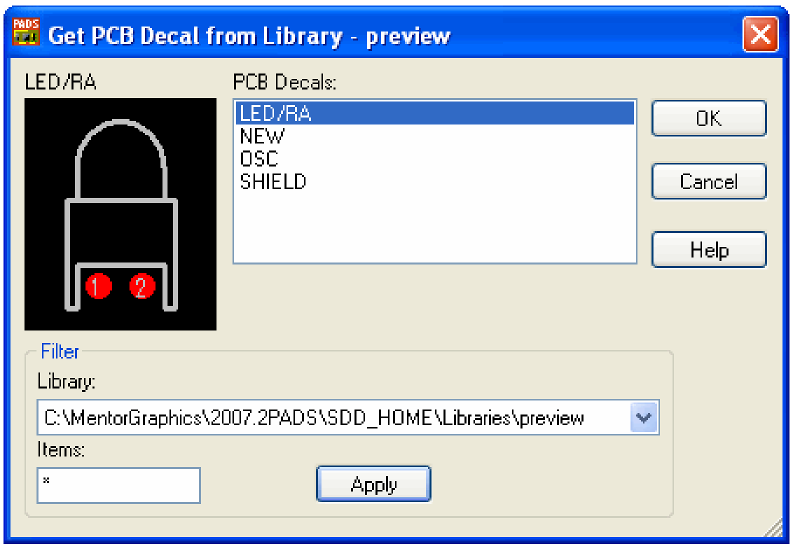


Table 45-179. Get PCB Decal from Library Dialog Box Contents

Name	**Description**
Preview area	Shows the item selected in the PCB Decals list.
PCB Decals list	Lists the decals available to you in the selected library.
Library list	Lists all libraries available to you.
Items	Narrows down your PCB Decals list. **Tip:** You can use wildcards in this box.
Apply	Executes the filter arguments.

Related Topics

Editing a Library Decal

Grid/Width Dialog Box

Use the Grid/Width dialog box to quickly change grid and width settings.

Accessing

- In the PCB Decal Editor, click **Setup** menu, then click **Grid and Width**.

Figure 45-194. Grid/Width Dialog Box

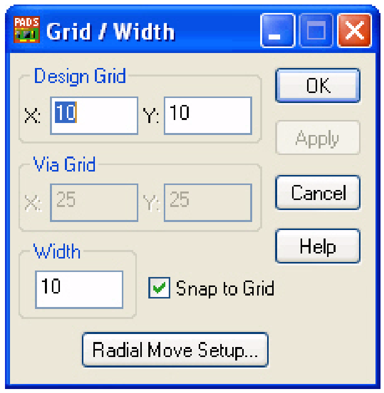


Table 45-180. Grid/Width Dialog Box Contents

Name	Description
Design Grid	Sets the spacing of the design grid, which controls the general placement of parts. The design grid uses positions relative to the origin. Type X and Y values for the grid in current design units.
Via Grid	Sets the spacing of the via grid which controls the placement of vias. Type X and Y values for the grid in current design units. **Restriction:** This section is unavailable in the PCB Decal Editor.
Width	Sets the current drafting width.
Snap to Grid	Snaps parts from grid point to grid point instead of freely and smoothly through the design space as you move or place the part. With this check box on, you cannot place a part off the current grid.
Radial Move Setup	Opens the Radial Move Setup dialog box where you can set up the polar grid.

Group Rules Dialog Box

Use the Group Rules dialog box to add and manage groups of pin pairs, and to define design rules that apply to them.

Accessing

- **Setup** menu > **Design Rules** > **Group** button

Figure 45-195. Group Rules Dialog Box

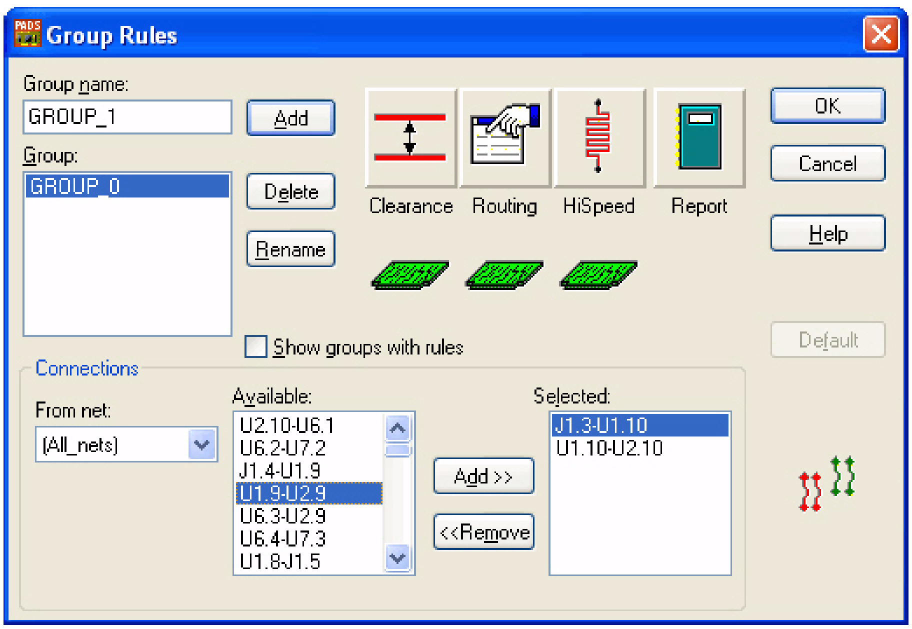


Table 45-181. Group Rules Dialog Box

Name	Description
Group name	Specifies the name of the group.
Group list	Lists all group names.
Add	Adds the group name to the Group list.
Delete	Removes the selected group from the Group list.
Rename	Renames the group selected in the Group list with the text in the Group Name box.
Clearance	Opens the Clearance Rules Dialog Box.
Routing	Opens the Routing Rules Dialog Box.

Table 45-181. Group Rules Dialog Box (cont.)

Name	Description
HiSpeed	Opens the HiSpeed Rules Dialog Box.
Report	Opens the Rules Report Dialog Box.
Picture below rule button	The picture below each type of rule button identifies which rules hierarchy level is used for that rule type. The meaning of the picture corresponds to the button in the Hierarchy area of the Rules dialog box. For example, if you select a class in the Class list and a green polygon appears below the Clearance button, then the default values apply to the class.
Show groups with rules	Specifies to show only groups that have rules.
Default	Removes non-default rules from the selected classes, so that only default rules apply.
From Net	Lists all available nets.
Connections Available list	Lists the connections available for this group. **Tip:** Connections cannot exist in more than one group. The Available list displays only connections that have not been assigned to a group.
Connections Selected list	Lists the connections selected for this group.
Add >>	Moves the connection from the Available list to the Selected list.
<< Remove	Moves the connection from the Selected list to the Available list.

Related Topics

Creating Group Design Rules

Deleting a Design Rule Group

Adding Pin Pairs to an Existing Design Rule Group

Removing Pin Pairs from a Design Rule Group

Modifying Group Design Rules

Renaming a Design Rule Group

Resetting Group Rules to Default Rules

Displaying the Pin Pairs of a Design Rule Group

HiSpeed Rules Dialog Box

Use the HiSpeed Rules dialog box to set up high-speed rules.

Requirement

The Electrodynamic Checking (EDC) licensed option is required to check high-speed rules.

Accessing

- **Setup** menu > **Design Rules** > choose a hierarchy level > **High Speed** button

Figure 45-196. HiSpeed Rules Dialog Box

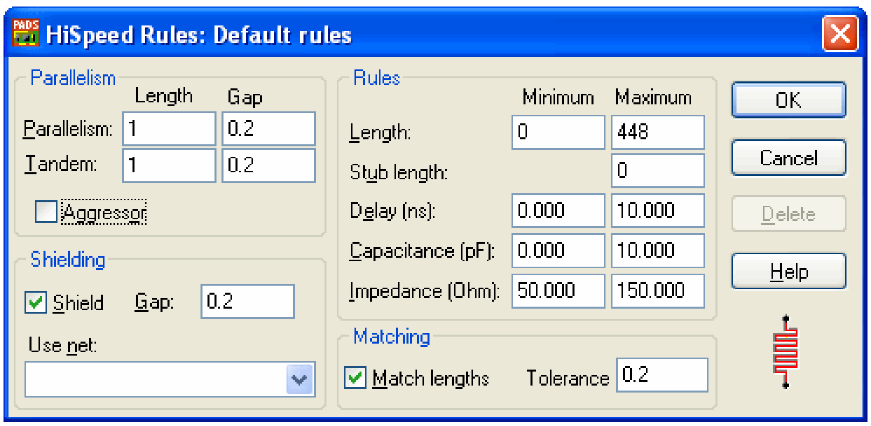


Table 45-182. HiSpeed Rules Dialog Box

Area	Description
Parallelism	• **Parallelism**—Specifies Length and Gap values for parallelism. **Tip:** Parallelism rules specified in the Conditional Rule Setup dialog box override rules specified in this dialog box. • **Tandem**—Specifies Length and Gap values for tandem. **Tip:** Tandem rules specified in the Conditional Rule Setup dialog box override rules specified in this dialog box. • **Aggressor**—Specifies that objects in this set act as aggressors.

Table 45-182. HiSpeed Rules Dialog Box (cont.)

Area	Description
Shielding	• **Shield** check box—Specifies to automatically route traces connected to a plane layer in order to shield selected nets from electromagnetic interference. **Exception:** Shielding rules do not apply when shielding with vias. **Restrictions:** • Options in the Shielding area are unavailable if the design has no plane layers. • PADS Layout does not automatically route shielding nets and it does not check shielding rules. • **Gap**—Specifies the clearance between the shielding and shielded nets. • **Use net** list—Lists all nets associated with the plane layer you want to implement the shielding traces with.
Rules area	
Length	Specifies the minimum and maximum length allowed.
Stub length	Specifies the maximum distance from the T-Junction to the end of the trace.
Delay (ns)	Specifies the minimum and maximum delay allowed. The baseline delay calculations use the trace parameters, the board material values you set in the Layer Thickness Dialog Box, and require at least one adjacent plane. A maximum of two adjacent planes are used in calculations. If the plane layer is a split/mixed plane layer, you must have a plane shape on the layer. The location and outline of the shape and cutouts of the plane area are all ignored in the calculation. Delay values can be viewed in the Pin Pair Properties and Net Properties dialog boxes. **Restrictions**: • This rule is only verified when you run Tools > Verify Design after you set up a High Speed check to check Delay. It is not verified by online DRC.
Capacitance (pF)	Specifies the minimum and maximum capacitance allowed. The baseline capacitance calculations use the trace parameters, the board material values you set in the Layer Thickness Dialog Box, and require at least one adjacent plane. A maximum of two adjacent planes are used in calculations. If the plane layer is a split/mixed plane layer, you must have a plane shape on the layer. The location and outline of the shape and cutouts of the plane area are all ignored in the calculation. Capacitance values can be viewed in the Net Properties dialog box. **Restrictions**: • Capacitance can’t be set for Pin Pairs. • This rule is only verified when you run Tools > Verify Design after you set up a High Speed check to check Capacitance. It is not verified by online DRC.

Table 45-182. HiSpeed Rules Dialog Box (cont.)

Area	Description
Impedance (Ohm)	Specifies the minimum and maximum impedance allowed. The baseline impedance calculations use the trace parameters, the board material values you set in the Layer Thickness Dialog Box, and require at least one adjacent plane. A maximum of two adjacent planes are used in calculations. If the plane layer is a split/mixed plane layer, you must have a plane shape on the layer. The location and outline of the shape and cutouts of the plane area are all ignored in the calculation. Impedance values can be viewed in the Pin Pair Properties and Net Properties dialog boxes. **Restriction—**This rule is only verified when you run Tools > Verify Design after you set up a High Speed check to check Impedance. It is not verified by online DRC.
Matching	Some routers can automatically route traces with matched lengths. For example, traces for differential pairs have matched lengths in order to avoid signal timing skew between the signals. • **Match lengths check box—**Specifies to automatically route traces with matched lengths. • **Tolerance—**Specifies the maximum permissible difference between the shortest and longest lengths between traces in the matched-length group. **Restriction:** PADS Layout does not check length-matching rules.
Delete button	Removes non-default hispeed rules at the current level of the rules hierarchy. **Restriction:** You cannot delete the Default High Speed rules.

Tip: Parallelism and tandem rules specified in the Conditional Rule Setup dialog box override rules specified in this dialog box.

Related Topics

Design Rule Hierarchy

HiSpeed Rules Dialog Box, Associated Nets

Use the HiSpeed Rules dialog box to assign min/max length rules and create matched length groups for associated nets.

Requirement

The Electrodynamic Checking (EDC) licensed option is required to check high-speed rules.

Accessing

- **Associated Net Rules dialog box > HiSpeed button**

Figure 45-197. HiSpeed Rules Dialog Box, Associated Nets

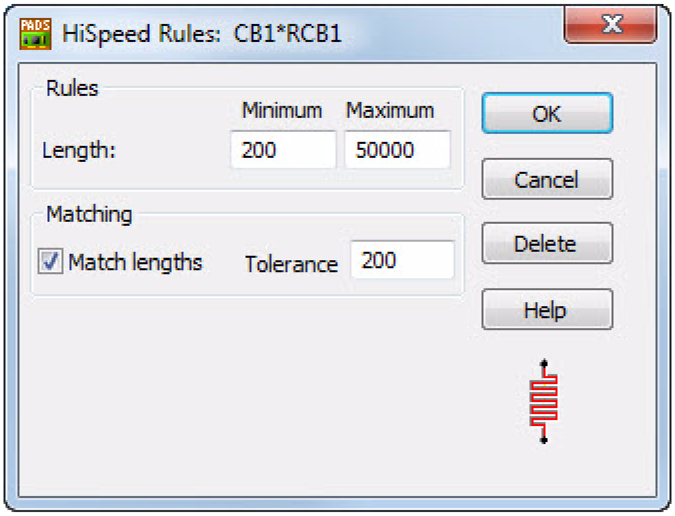


Table 45-183. HiSpeed Rules Dialog Box, Associated Nets

Name	Description
Length	Set the minimum and maximum length for the associated nets you selected in the Associated Net Rules Dialog Box
Match lengths	Select this checkbox to make the associated nets you selected in the Associated Net Rules Dialog Box members of a matched length group. Clear the checkbox to remove the selected associated nets from an existing matched length group.
Tolerance	Set the tolerance value for length-matching.

Related Topics

Design Rule Hierarchy

HYP Export Dialog Box

Use the HYP Export dialog box to export the design in the form of a HYP file. For information about the format of the HYP file and creating a “BoardSim-friendly” design, see the HyperLynx online Help. PADS Layout passes the Value, Tolerance, Voltage, HyperLynx, and

PowerGround attributes to the HYP file. BoardSim uses these attributes to obtain values for resistors and capacitors, and to transfer information about fixed voltage nets.

You would use this export functionality only if you don't have HyperLynx BoardSim installed, or if you need to send the file to another computer. If you have it installed, you could use the more direct method - use the Tools menu > Analysis > Signal/Power Integrity to export the file and open it in HyperLynx BoardSim. See BoardSim Dialog Box.

Restriction

Only HyperLynx v8.0 or newer can open the file that is exported by this dialog box. The .hyp file that is created is a v2.34 format file.

Accessing

- **File menu** > **Export** > select **HYP Files** > **Save**

Figure 45-198. HYP Export Dialog Box

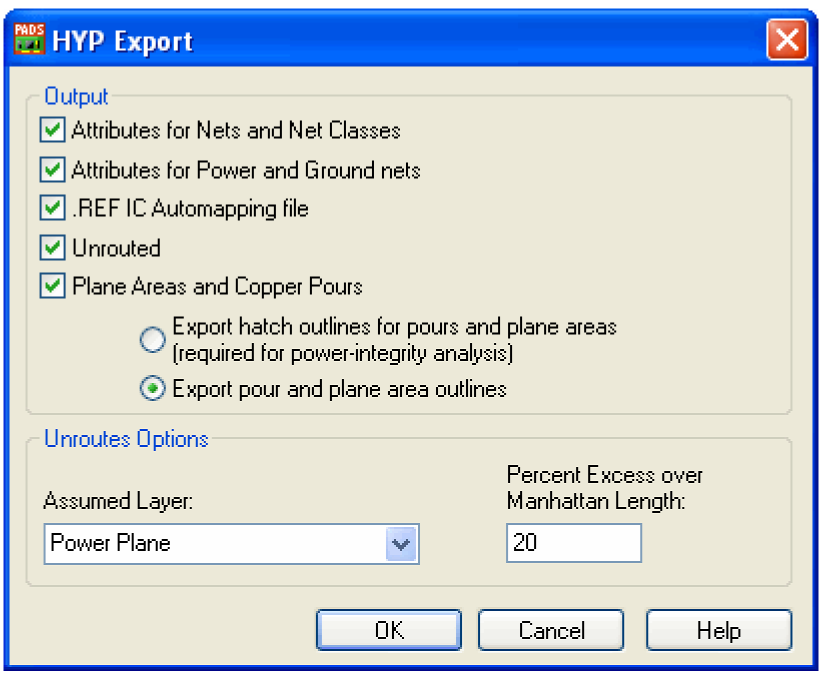


Table 45-184. HYP Export Dialog Box Contents

Name	Description
Output area	
Attributes for Nets and Net Classes	Specifies to export attributes for nets and net classes.

Table 45-184. HYP Export Dialog Box Contents (cont.)

Name	Description
Attributes for Power and Ground nets	Specifies to export attributes for power and ground nets.
.REF IC Automapping file	Specifies to create a HyperLynx .ref file, which maps IC reference designators in the design to the BoardSim models that represent the ICs. BoardSim uses the mappings to automatically load IC models when you select a net for simulation. **Restriction:** This check box is only available if a component has the HyperLynx.Model attribute with a value.
Unrouted	Specifies to export unrouted nets.
Plane Areas and Copper Pours	Specifies to export plane areas and copper pours. • **Export hatch outlines for pours and plane areas**—Specifies to export hatch outlines for pours and plane areas. **Requirement:** This setting creates extra constructs in the .hyp file and is required for power-integrity analysis. • **Export pour and plane area outlines**—Specifies to export pour and plane area outlines.
Unroutes Options area **Tip:** Available only if Unrouted is selected in the Output area.	
Assumed Layer	Specifies the layer on which to implement the unrouted nets.
Percent Excess over Manhattan Length	Specifies the value to estimate the routing lengths. **Tip**: This value adds a percentage of the Manhattan length to the route length, to account for indirect routing paths. Net lengths are based on the Manhattan distance between pin pairs, which is Delta X plus Delta Y.

Related Topics

BoardSim Dialog Box

Creating HyperLynx BoardSim - HYP Files

IDF Export Dialog Box

U the IDF Export dialog box to exchange design data between PADS Layout and a mechanical design system. You can export IDF files to export board outlines, keepouts, components, and holes to a mechanical design system.

Tip: Set any IDF-specific part height information, drilled hole information, or part outline information for a more accurate IDF export.

Accessing

- **File** menu > **Export** > **Select IDF File** > **Save**

Figure 45-199. IDF Export Dialog Box

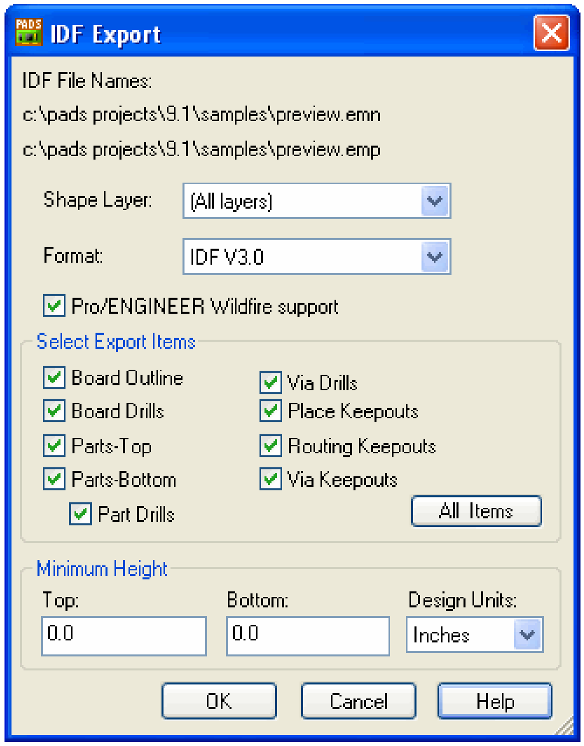


Table 45-185. IDF Export Dialog Box Contents

Name	Description
IDF File Names	The names of the files you are exporting. The .emn (board and placement) and .emp (part library) files are created.
Shape Layer	Specifies the layer containing the outline information for the decals in your design that you want to send to the mechanical design system.
Format	Specifies the IDF version to use.
Pro/ENGINEER Wildfire support	Select this check box to convert characters to underscores (_) when they are illegal characters to Pro/ENGINEER Wildfire. Clear this check box to allow all characters to export in the IDF files.

Table 45-185. IDF Export Dialog Box Contents (cont.)

Name	Description

Select Export Items area	Use these check boxes to select the items you want to export to the layer in the Shape Layer list.
	Tip: Click All Items to select everything in this area.
	• **Board Outline**—Board outline, cutouts, and holes.
	• **Board Drills**—Drilled holes associated with mounting holes, including drill diameter and plated status.
	• **Parts-Top**—Components mounted on the top of the board, including their locations.
	• **Parts-Bottom**—Components mounted on the bottom of the board, including their locations.
	• **Part Drills**—Drilled holes associated with a part pin, including drill diameter and plated status.
	Tip: This option is unavailable for IDF 2.0 or if the Parts-Top and Parts-Bottom check boxes are cleared.
	• **Via Drills**—Drilled holes associated with a via, including drill diameter and plated status.
	Tip: This option is unavailable for IDF 2.0 and during import.
	• **Place Keepouts**—Placement keepouts, including keepouts with height restrictions, and their locations. IDF files can contain the following information:
	  • Board-level placement keepouts
	  • Top and bottom component height restrictions defined for the whole board in the Options Dialog Box, Drafting / Text and Lines Page.
	• **Routing Keepouts**—Trace keepouts and their locations.
	Only trace keepouts on top, bottom, both, inner (IDF 3.0 only), and All layers are supported. Trace keepouts on a single inner layer are not supported.
	Restriction: This option is unavailable for IDF 2.0.
	• **Via Keepouts**—Via keepouts and their locations.
	IDF supports only via keepouts that apply to all layers. Imported via keepouts are always set on all layers.
	Restriction: This option is unavailable for IDF 2.0.

Top/Bottom	Specifies the minimum height of components you want to export. **Tip:** Components less than these heights are not exported. If a part is not exported because of a minimum height value, a message is written to the status log file.
Design Units	Specifies the design units for this file.

Related Topics

Exporting IDF Files

IDF Import Dialog Box

Use the IDF Import dialog box to import board outlines, keepouts, components, and holes from a mechanical design system. PADS Layout cannot import the information in the .emp library file.

Accessing

- **File** menu > **Import** > **Select IDF File** > **Open**

Figure 45-200. IDF Import Dialog Box

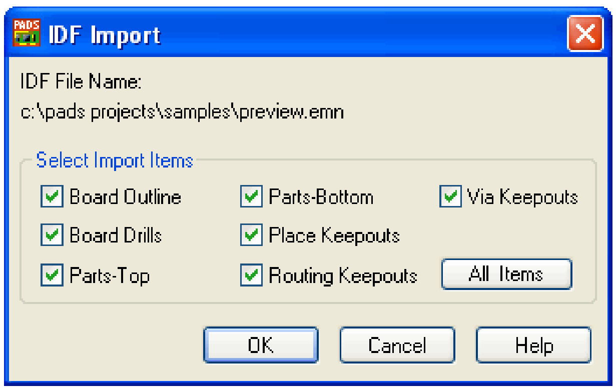


Table 45-186. IDF Import Dialog Box Contents

Name	Description
IDF File Name	The name of the file you are importing.

Table 45-186. IDF Import Dialog Box Contents (cont.)

Name	Description

Select Import Items area	Use these check boxes to select the items you want to import.
	Tip: Click All Items to select everything in this area.
	• **Board Outline**—Board outline, cutouts, and holes.
	• **Board Drills**—Drilled holes associated with mounting holes, including drill diameter and plated status.
	• **Parts-Top**—Components mounted on the top of the board, including their locations.
	• **Parts-Bottom**—Components mounted on the bottom of the board, including their locations.
	• **Part Drills**—Drilled holes associated with a part pin, including drill diameter and plated status.
	Tip: This option is unavailable for IDF 2.0 or if the Parts-Top and Parts-Bottom check boxes are cleared.
	• **Via Drills**—Drilled holes associated with a via, including drill diameter and plated status.
	Tip: This option is unavailable for IDF 2.0 and during import.
	• **Place Keepouts**—Placement keepouts, including keepouts with height restrictions, and their locations. IDF files can contain the following information:
	  • Board-level placement keepouts
	  • Top and bottom component height restrictions defined for the whole board in the Options Dialog Box, Drafting / Text and Lines Page
	• **Routing Keepouts**—Trace keepouts and their locations.
	Only trace keepouts on top, bottom, both, inner (IDF 3.0 only), and All layers are supported. Trace keepouts on a single inner layer are not supported.
	Restriction: This option is unavailable for IDF 2.0.
	• **Via Keepouts**—Via keepouts and their locations. IDF supports only via keepouts that apply to all layers. Imported via keepouts are always set on all layers.
	Restriction: This option is unavailable for IDF 2.0.

Related Topics

Importing IDF Files

Increase Maximum Layer Number Dialog Box

Use the Increase Maximum Layer Number dialog box to increase the number of available electrical and documentation layers from 30 to 250.

Accessing

- **Setup** menu > **Layer Definition** > **Max Layers** button

Figure 45-201. Increase Maximum Layer Number Dialog Box

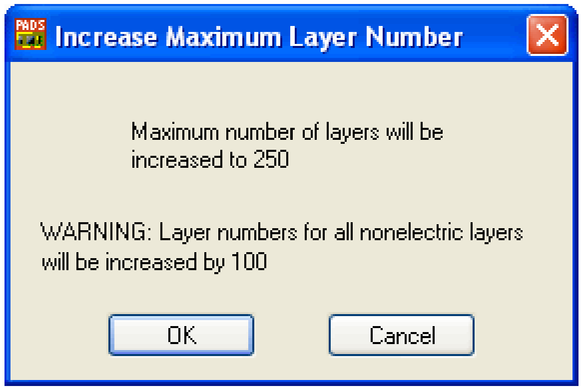


Table 45-187. Increase Maximum Layer Number Dialog Box Contents

Name	Description
Maximum number of layers will be increased to 250	The default layer setup contains a total of 30 layers. There are 20 electrical and 10 non-electrical (documentation) layers. You can increase this layer setup to 250 layers. The increase would allow up to 64 electrical layers and 186 non-electrical layers.
Warning: Layer numbers for all nonelectrical layers will be increased by 100	In the default layer setup, the documentation layers start at level 21 - the Solder Mask Top layer. In the increased layer setup, this layer becomes layer 121.

Related Topics

Increasing the Maximum Number of Available Layers

Layer Modes in the *Concepts Guide*

Installed Options Dialog Box, License File Tab

If you are using node-locked licensing, you can view the contents of a license file. If you are using floating licenses, you cannot view the actual license file, but you can view the status of the features associated with a server license.

Accessing

- **Help** menu > **Installed Options** > **License File** tab

Figure 45-202. Installed Options Dialog Box, License File tab

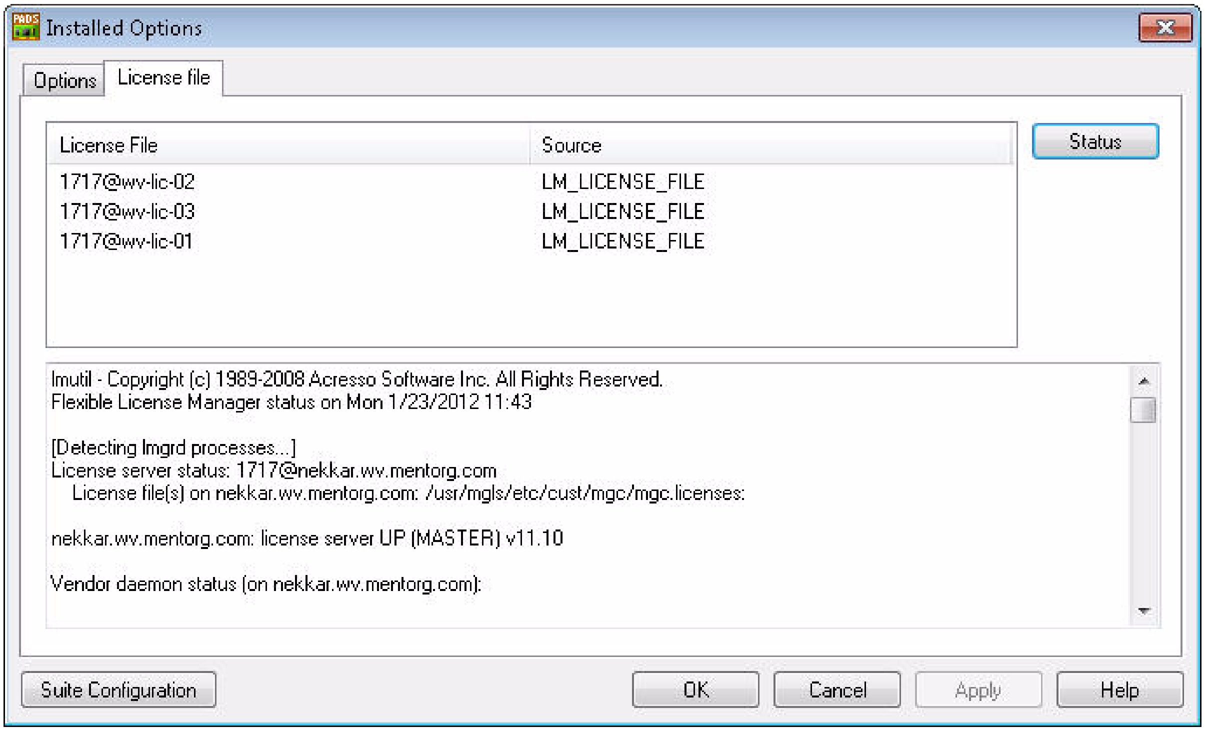


Table 45-188. License File tab contents

Name	Description
License File column	Displays the location of the license file(s) found on your computer.
Source column	Displays the source of the license.
View button	Click to display the contents of the license file in the License Information box. **Restriction:** Node-locked only.
Status button	Click to display the status of this license in the License Information box. **Restriction:** Floating License only.
License Information box	Displays the contents (Node-locked) or the status (Floating) of the selected license.
Suite Configuration button	Opens the PADS Suite Configuration dialog box. **Restriction:** Available only with floating/server-based licenses, a mix of different PADS Suites, or a mix of unbundled licenses and suites.

Related Topics

Viewing a License File or License Status

Installed Options Dialog Box, Options Tab

Use the Options tab to configure your licensing information. You generally need to check individual licensing options in or out if you are using floating licenses. The configuration is stored in the powerpcb.ini file.

Tip: For node-locked licensing, all of the licensed options are available and checked out. For floating licensing, the first person logged in controls the available options.

Accessing

- **Help** menu > **Installed Options** > **Options** tab

Figure 45-203. Installed Options Dialog Box, Options tab

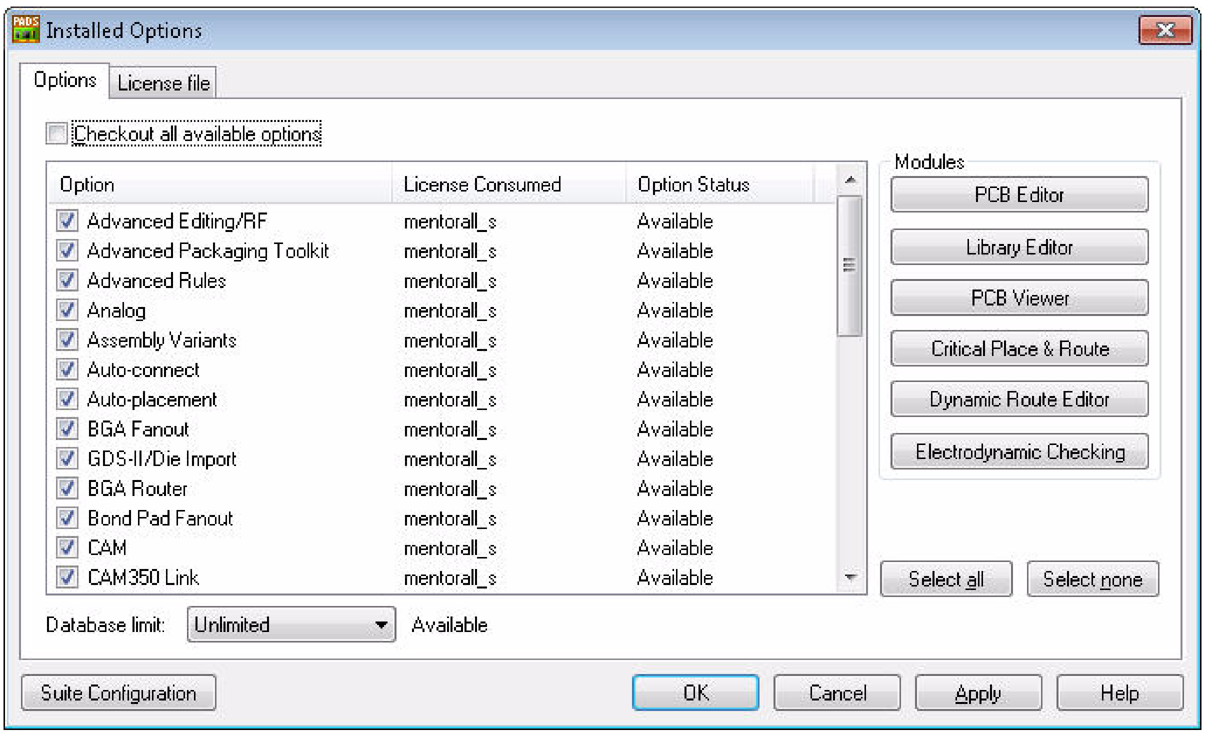


Table 45-189. Options tab contents

Name	Description
Check Out All Available Options	Specifies to check all available licensing options in and out. Click to clear if you want to select individual options associated with individual PADS Layout modules and optionally select a database limit.
Option column	Lists the available options. A check mark indicates you want to use this option.
License Consumed column	Displays the licensing feature associated with the option listed.
Status column	Displays whether the option is available to you.
Modules area	Lists the predefined set of options associated with one or more PADS Layout modules. **Tip:** If more than one module sets a particular option, turning off one of the modules does not turn off that shared option. The option is only turned off when all of the modules that use it are turned off. Also, some individual options control other options. For example, clearing the Advanced Packaging Toolkit option clears BGA options.
Database Limit	Specifies the database limit that is appropriate for your licensing scheme in the Database Limit area. The database limit is associated with the number of connections (pin pairs) you can have: 1500 for the Standard Database and 6250 for the Expanded Database. Select Standard, Expanded, or Unlimited for the database limit and verify the availability of the selected limit, per your license.
Suite Configuration button	Opens the PADS Suite Configuration dialog box. **Restriction:** Available only with floating/server-based licenses, a mix of different PADS Suites, or a mix of unbundled licenses and suites.

Related Topics

Checking Licensing Options In and Out

IPC Export Dialog Box

Use this dialog box to choose between IPC-D-356 netlist formats.

Accessing

- **File** menu > **Export** > **Select IPC356 Files** > **Save**

Figure 45-204. IPC Export Dialog Box

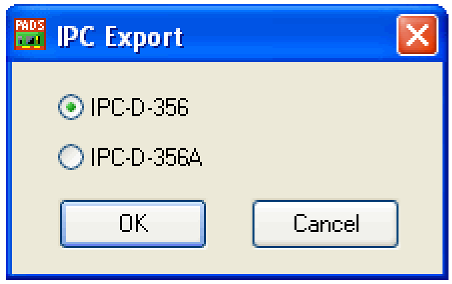


Table 45-190. IPC Export Dialog Box Contents

Name	Description
IPC-D-356	Select this option to create a bare board test-information netlist in the basic 356 format.
IPC-D-356A	Select this option to create a bare board test-information netlist in the more advanced 356 revision A format.

Related Topic

The IPC-D-356 Netlist

Exporting an IPC-D-356 Netlist

JEDEC Array Pinning Dialog Box

Use the JEDEC Array Pinning dialog box to assign an alphanumeric string to each pin in an array following the JEDEC standard.

Pin rows are lettered from top to bottom starting with A. The letters I, O, Q, S, X, and Z are not used.

Pin columns are numbered starting with 1. For component type, column numbering is left to right and for substrate type, right to left. For arrays with more than 20 rows, row 21 is designated AA and subsequent rows are designated AB, AC, and so on.

Accessing

- **Tools** menu > **PCB Decal Editor** > **Tools** menu > **Assign JEDEC Array Pinning**

Figure 45-205. JEDEC Array Pinning Dialog Box

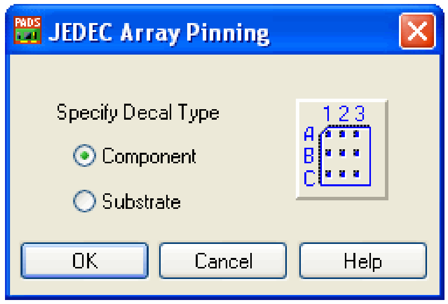


Table 45-191. JEDEC Array Pinning Dialog Box Contents

Name	Description
Specify Decal Type	Specifies whether the decal is a component or substrate type.
Preview area	Displays the alphanumeric assignment orientation for Component or Substrate decal type.

Related Topics

To Assign JEDEC Pinning

Jumper Name Properties Dialog Box

Use the Jumper Name Properties dialog box to modify the jumper name and its attributes.

Accessing

- **Select a jumper name > Right-click > Properties**

  Or

- Select a jumper > Right-click > **Properties** > **Label** button

Figure 45-206. Jumper Name Properties Dialog Box

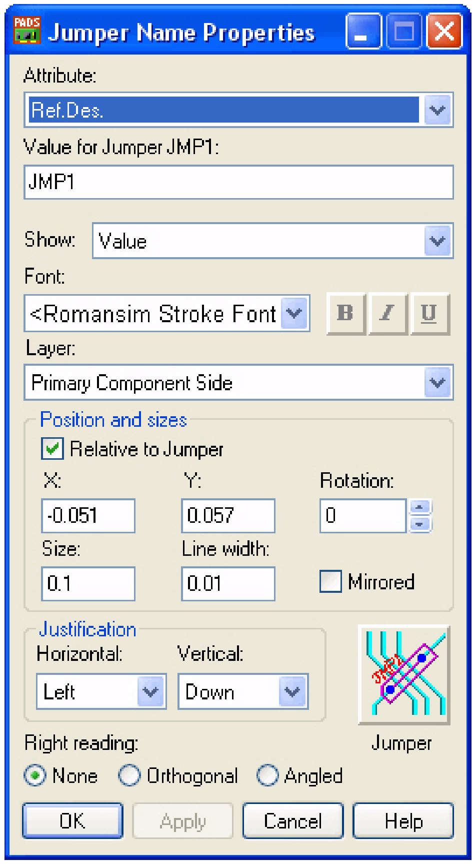


Table 45-192. Jumper Name Properties Dialog Box contents

Name	Description
Attribute	The attributes available to you. If you are creating labels for jumpers, Reference Designator is the only available attribute. **Tip:** Hidden attributes do not appear in the Attribute list unless the attribute was selected for the label before it was set as a hidden attribute.

Table 45-192. Jumper Name Properties Dialog Box contents (cont.)

Name	Description
Value for	The value of the selected attribute. **Tips:** • Unavailable if you selected Reference Designator or Part Type from the Attribute list, if the attribute is read-only, or if you open the Properties dialog box for labels of different attribute types. However, if the labels you select belong to attributes of the same type, you can edit this box. • If the attributes have different values, the box is blank. You can type a new value in the box to apply it to all of the selected attribute labels and their parent objects. • Value is also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.
Show	Controls the visibility of the label. • **None**—Turns visibility off. • **Value**— Displays only the label value. • **Name and Value**—Displays the name and value. • **Full Name and Value**—When labeling a structured attribute, displays the full structured name and value. **Tip:** Labels are invisible regardless of this setting unless you use the Display Colors Setup dialog box to change the color of labels to a color different from that of the background.
Font	The fonts available to you. **Tips:** • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.
Layer	The layers available to you.
Relative to	Places the label at the X and Y location relative to the component or the jumper. If you clear this check box, the label is placed at the X and Y location relative to the design origin.
X,Y	Places the decal label in a specified location.
Rotation	Specifies the rotation angle of the label.

Table 45-192. Jumper Name Properties Dialog Box contents (cont.)

Name	Description
Size	Specifies the size of the font. **Size (pts):** This is font size in points and appears for system fonts **Size (mils):** This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters. 
Line Width	Specifies the line width for stroke fonts only. 
Mirrored	Flips the label - text is considered readable from the bottom side of the board.
Horizontal, Vertical	Sets the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change. **Tips:** • For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**. • Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.
Right reading	Controls whether labels are readable (from left to right, or from bottom to top if the label is rotated). Click the **None**, **Orthogonal**, or **Angled** button to indicate the direction of reading you want.
Jumper button	Opens the Jumper Properties dialog box where you can modify the jumper.

Related Topics

Modifying Jumper Name Properties

Modifying Jumper Pin Properties

Modifying Jumper Properties

Using Jumpers

Jumper Pin Properties Dialog Box

Jumper Properties Dialog Box

Jumper Pin Properties Dialog Box

Use the Jumper Pin Properties dialog box to display the net to which the jumper pin belongs, the reference designator, pin number, connection to which the jumper pin is attached, and the coordinates of the selected jumper pin.

Accessing

- **Select a jumper pin > Right-click > Properties**

Figure 45-207. Jumper Pin Properties Dialog Box

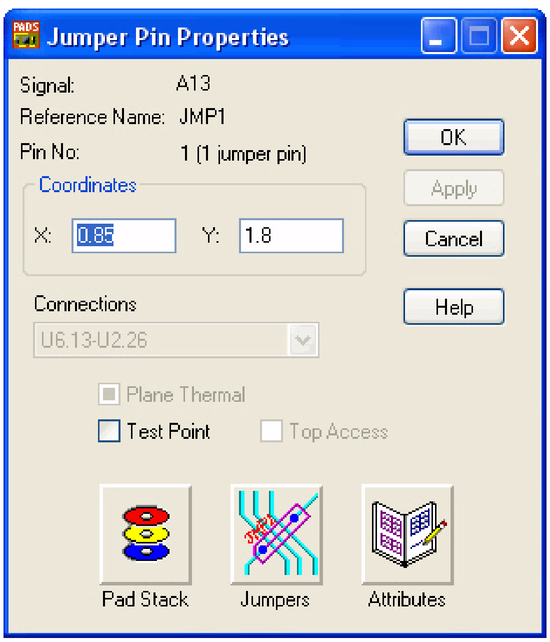


Table 45-193. Jumper Pin Properties Dialog Box contents

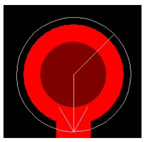
Name	Description
Signal	The designation of the net to which the jumper belongs.
Reference Name	The reference designator of the jumper.
Pin Number	The number of the pin in the jumper. Pin one is the first pin you entered in the jumper.
X/Y	The X and Y location of the selected pin. Type in these fields to change the pin location.
Connections	The connection to which the jumper is attached.
Plane Thermal	Determines whether the pin or via is eligible to receive a thermal. The status of a thermal is set individually by pin and via, and the thermal indicators will appear on plane layers only. Click to clear this check box if you do not want the via or pin to connect to any plane. Once a pin or via is eligible, it is not automatically assigned a thermal attribute.
Test Point	Makes the via or pin a test point. **See also:** Performing a Test Point Audit This is a three-state check box that depends on the state of the selected objects. If all of the selected vias or pins are a test point, then it is on. If none of the selected vias or pins are a test point, then it is off. If some of the selected vias or pins are a test point and some are not, then it is undefined. You can make all selected vias or pins test points by turning Test Point on and choosing Apply. You can remove the test point from all selected vias or pins by turning Test Point off and choosing Apply. When you click Apply the pad stack is automatically checked to see if the via or pin can be a test point; for example, you cannot make buried vias test points because a probe cannot access a buried via. **Tip:** When the via or pin is flagged as a test point, and Show Test Points is checked on the Routing / General page of the Options dialog box, an arrow is drawn on it in the design: 

Table 45-193. Jumper Pin Properties Dialog Box contents (cont.)

Name	Description
Top Access	Attempts to probe the test point from the top and bottom in DFT Audit. The default is bottom; so with Top Access off, DFT Audit automatically tries to probe the test point from the bottom. **See also:** Performing a Test Point Audit When you click Apply, the pad stack is automatically checked to see if top access is valid; for example, you must assign Top Access to partial vias with only top access if you want to use the vias as test points. You can only set the Top Access option if the via or pin is a test point (Test Point is on).
Pad Stack button	Opens the jumper pin in the Jumper Parameters Properties Dialog Box. You can change the pad stacks for individual pins in the jumpers. **See also:** Editing a Pad Stack in the PCB Decal Editor
Jumpers button	Opens the Jumper Properties dialog box where you can edit the settings for the entire jumper.
Attributes button	Opens the Object Attributes dialog box and displays attribute information for the selected objects. You can view and modify nail diameter and nail number pin attributes for component pins, vias, and jumper pins, including test point attributes.

Related Topics

Modifying Jumper Name Properties

Modifying Jumper Pin Properties

Modifying Jumper Properties

Using Jumpers

Jumper Name Properties Dialog Box

Jumper Properties Dialog Box

Connecting a Net with a Plane

Setting Pins and Vias as Thermals

Jumper Properties Dialog Box

Use the Jumper Properties dialog box to modify jumper location, label, and size.

Accessing

- **Select a Jumper > Right-click > Properties**

Figure 45-208. Jumper Properties Dialog Box

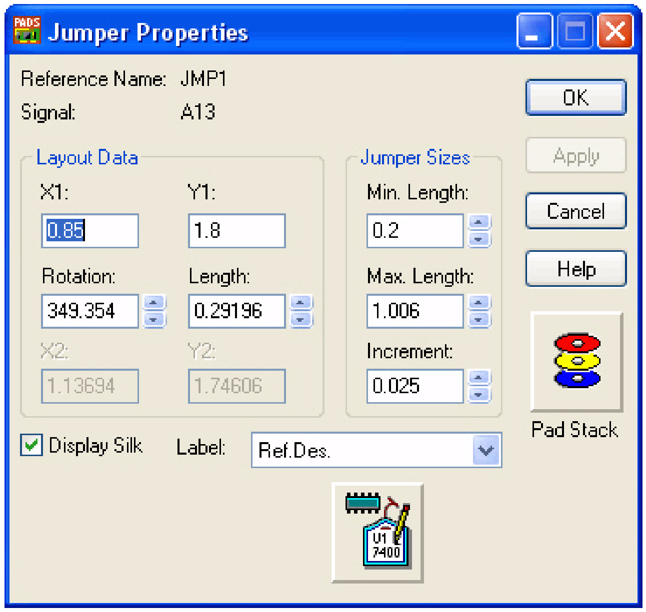
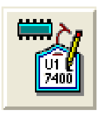


Table 45-194. Jumper Properties Dialog Box contents

Name	Description
Reference Name	The reference designator of the jumper.
Signal	The designation of the net to which the jumper belongs.
X1/Y1 boxes	The X and Y location of pin one. Type in these fields to change the location.
Rotation	The jumper rotation. Type in this field to change the rotation.
Length	The jumper length. Type in this field to change the length. The value must be within the minimum and maximum length values.
X2/Y2	The X and Y location of pin two. Type in these fields to change the location.
Minimum Length	Specifies the minimum length of the jumper.
Maximum Length	Specifies the maximum length of the jumper.
Increment	Specifies the increment at which you can stretch the jumper between the minimum and maximum lengths.

Table 45-194. Jumper Properties Dialog Box contents (cont.)

Name	Description
Display Silk	Enables the display of a silkscreen outline for the jumper. **Tips:** • For CAM output, you must enable Component Outlines for the layer on which the jumper resides before jumper outlines will plot. • The outline for jumpers is set at 10 mils; you cannot modify this setting.
Label	Lists existing labels for reference designator, part type, and attributes. To edit an existing label, click a label from the list and click the button in this tab. A label is selected instead of the component, and the corresponding Labels Properties dialog box appears where you can modify the label. To create a new label, click <new> from the list and click the button in this tab. The Add New Part Label dialog box appears where you can set up the new label. **Tip:** When modifying the Properties of a jumper name, Reference Designator is the only available label.
	Opens the appropriate Label Properties dialog box if an existing label is selected in the Label list. **Tip:** When the current color for labels is set to the background color, this option is unavailable. To activate Label, assign a non-background color to labels in the Display Colors Setup dialog box. Opens the Add New Part Label dialog box if <new> is selected in the Label list.
Pad Stack button	Opens the jumper pin in the Jumper Parameters Properties Dialog Box. You can change the pad stacks for individual pins in the jumpers. **See also:** Editing a Pad Stack in the PCB Decal Editor

Related Topics

Modifying Jumper Name Properties

Modifying Jumper Pin Properties

Modifying Jumper Properties

Using Jumpers

Jumper Name Properties Dialog Box

Jumper Pin Properties Dialog Box

Jumpers Dialog Box

Use the Jumpers dialog box to set up and modify jumpers and jumper pad stacks. You can create and modify SMD jumpers (single layer jumpers) on the top or bottom mounting layer.

Accessing

- **Setup Menu** > **Jumpers (Jumpers Dialog Box)**
- Select a Jumper > right-click > **Properties** > **Pad Stack** button (Jumper Parameters Properties Dialog Box)

The Jumpers dialog box controls change depending on what you have selected for the Pad Style. The three major differences are:

- Figure 45-209: Jumpers Dialog Box - Pad
- Figure 45-210: Jumpers Dialog Box - Thermal
- Figure 45-211: Jumpers Dialog Box - Antipad

Figure 45-209. Jumpers Dialog Box - Pad

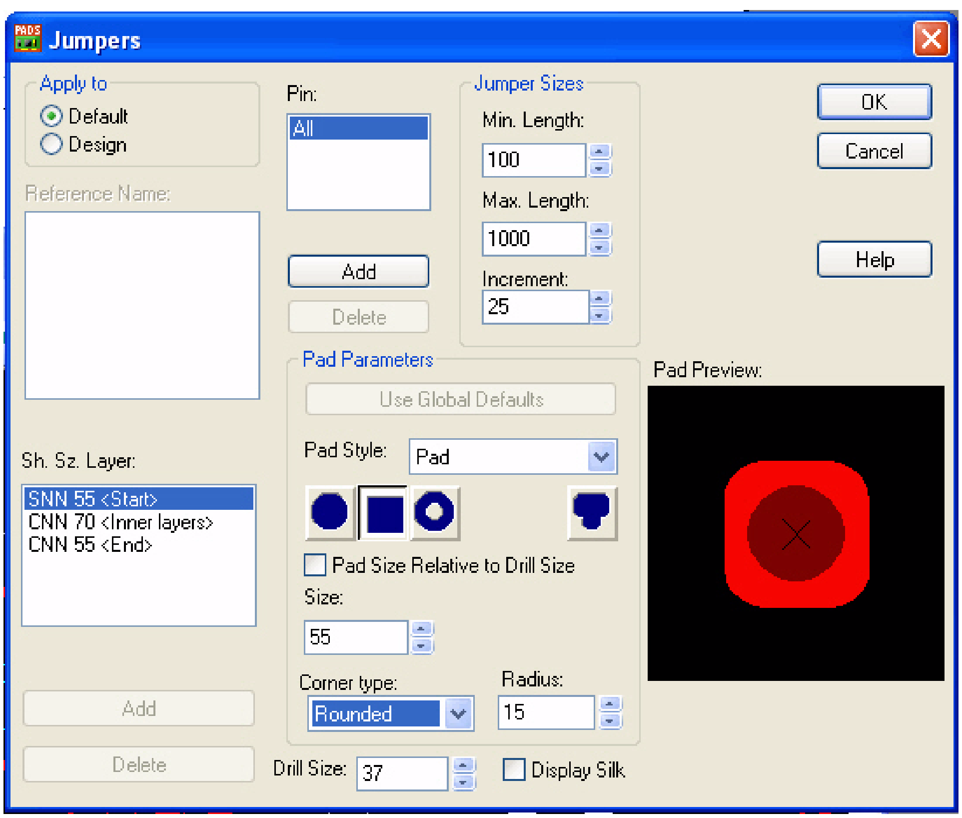


Figure 45-210. Jumpers Dialog Box - Thermal

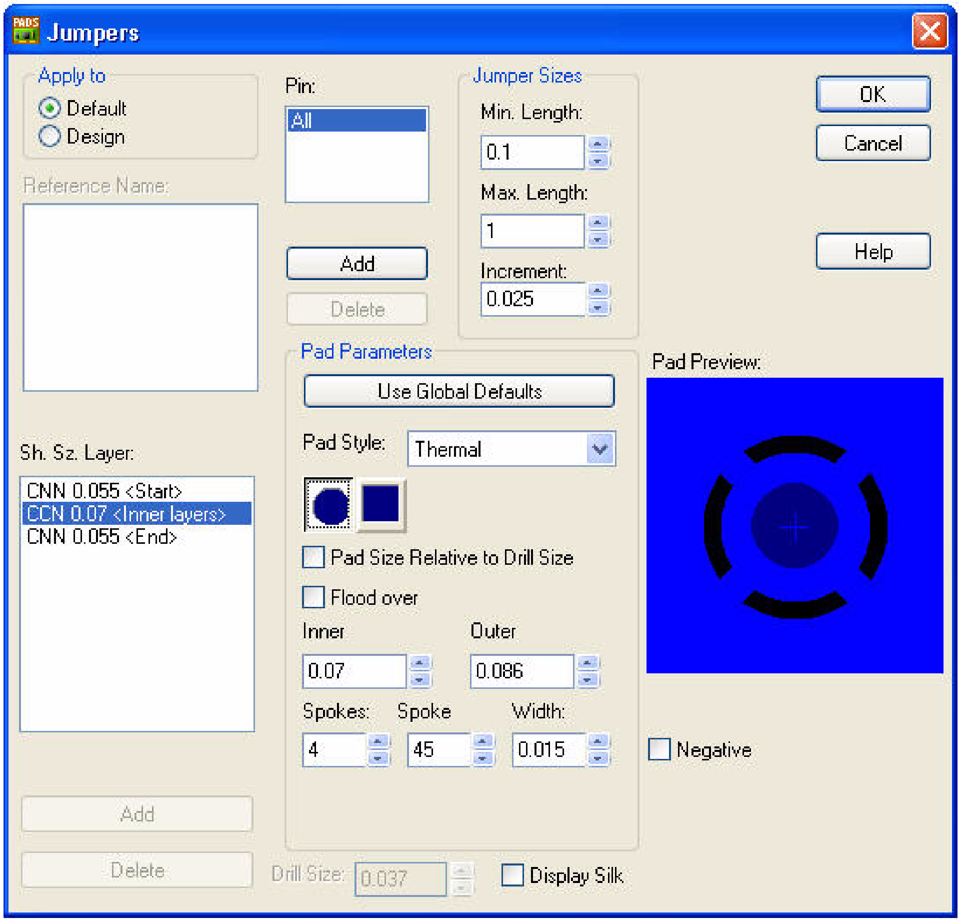


Figure 45-211. Jumpers Dialog Box - Antipad

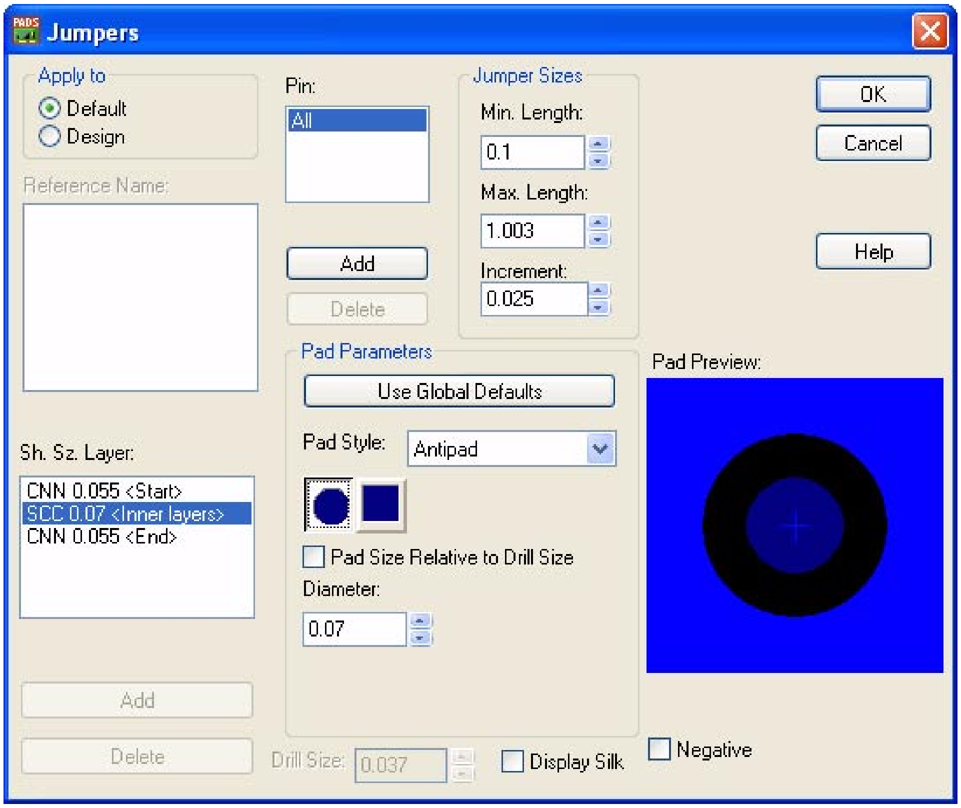


Table 45-195. Jumpers Dialog Box Contents

Name	Description
Apply to	**Default**—Specifies that you are setting up the default jumper. **Design**—Specifies that you are setting up the jumper for this specific design.
Reference Name	Lists the available reference names.
Shape, Size, and Layer	Lists the layers on which you can make jumper pad stack changes. **Exceptions:** When modifying Design jumpers, you can add individual layers of the design to the list for customizing the design jumper. Use the Add or Delete button to maintain the Shape/Size/Layer list.
Add	Opens the Add Layer dialog box.
Delete	Removes the selected shape
Pin list	Lists the pins available to you.

Table 45-195. Jumpers Dialog Box Contents (cont.)

Name	Description
Add Pin	Opens the Add Pin dialog box where you can add an existing pin to the Pin list.
Delete Pin	Deletes the selected pin from the Pin list.
Min. Length	Specifies the minimum length of the jumper.
Max Length	Specifies the maximum length of the jumper.
Increment	Specifies the increment at which you can stretch the jumper between the minimum and maximum lengths.
Use Global Defaults	Sets thermal and antipad shapes to those specified in the Thermals tab in the Options dialog box. **Restriction:** This option is available when the Pad Style list is set to Thermal or Antipad. **See also:** Design Rule Versus Pad Stack - Thermals and Antipads

Table 45-195. Jumpers Dialog Box Contents (cont.)

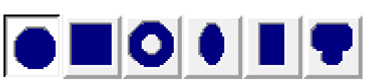
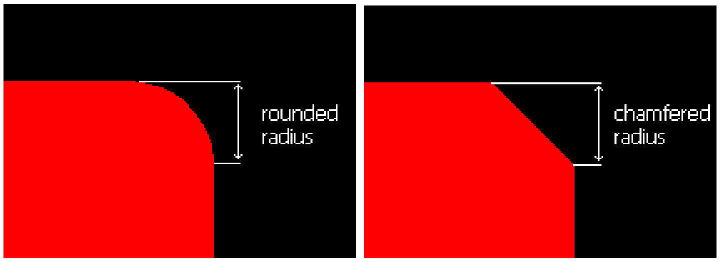
Name	Description
Pad Style	Specifies the style of pad: normal pad, thermal pad, or antipad. Thermal and Antipad display configuration controls the size and shape of thermals and antipads used on split/mixed layers and CAM negative planes (for RS-274X output). **See also:** Design Rule Versus Pad Stack - Thermals and Antipads **Tips:** • Beginning with PADS 9.2, the size, shape and orientation of thermals for slotted pads are no longer derived from the length, drill size and orientation of the slotted hole, but are inherited from the normal pad. • Set the Inner Diam and Outer Diam values to be the same for a solid connection to the plane (flood over). The current pad diameter is used as the inner diameter with the outer diameter set at the default same-net pad to corner rule. For antipads, diameters are initially set to follow the current default pad to copper design rule. If you select Use Design Rules for Thermals and Antipads in the Split/Mixed Plane page of the Options dialog box, the outer diameter is ignored and the clearance rule is used instead, except when the outer diameter is less than the inner diameter. Inner and outer diameter options always, however, control flood over. The length of an antipad having a non-zero drill size equals the slot length minus the drill size, plus the width. **Tip:** You cannot create antipads on outer layers. When you select an outer layer in the Size, Shape, Layer list, Antipad is unavailable.
Shape buttons 	Assigns a pad shape to the layer selected in the Size, Shape, Layers list. You can assign via pads as round, square, annular, oval or rectangular, odd. • Annular lets you specify an inner pad diameter, bringing the inner diameter of the pad inside the drill outline. A pilot dimple at the center of the copper pad appears as an aid to hand fabrication of some prototypes. • Odd is useful for drawn items such as moiré pads or registration marks. It is a circular outline only, and is usually used for marking areas for airgap checking. This setting should not be confused with trying to create a custom shaped (odd shaped) pad. See Creating a Custom Pad Shape for more information.

Table 45-195. Jumpers Dialog Box Contents (cont.)

Name	Description
Pad size relative to drill size	Displays inner and outer pad sizes relative to the drill size.
Flood over	Specifies that the pad requires no thermal relief and should be flooded over, irrespective of any default flooding options for a normal pad of this shape. **Restriction:** Available only when **Pad style** is Thermal and a pad shape button is selected.
Pad Parameters area	The options for sizing the pad shape vary according to the shape you choose. **Round**—Diameter (if hole is not slotted), Width (if hole is slotted). If the pad style is thermal: Inner Diam, Outer Diam, Spokes, Spoke Angle, Spoke Width. **Square**—Size, Width (if hole is slotted), Corner type and Radius. If the pad style is thermal: Inner Size, Outer Size, Spokes, Spoke Angle, Spoke Width. **Annular**—Diameter and Inner Diameter **Odd**—Diameter **Figure 45-212. Radius Examples** 
Drill Size	Specifies the drill size if the jumper is a through hole jumper. **Tip:** Type a drill size of zero if you want a surface mount jumper with round pads.
Display Silk	Specifies to display the silkscreen outline for the jumper. **Tips:** • For CAM output, you must enable Component Outlines for the layer on which the jumper resides before jumper outlines will plot. • The outline for jumpers is set at 10 mils; you cannot modify this setting.
Pad Preview	Shows pad shape and size for the current options.
Negative	Changes the preview area to a negative view. **Restriction:** Thermal Pad Style only.

Related Topics

Setting Up Jumpers

Latium Checking Setup Dialog Box

Use the Latium Checking Setup dialog box to set the rules to check using PADS Router.

Accessing

- **Tools** menu > **Verify Design** > **Latium Design Verification** check > **Setup button**

Figure 45-213. Latium Checking Setup Dialog Box

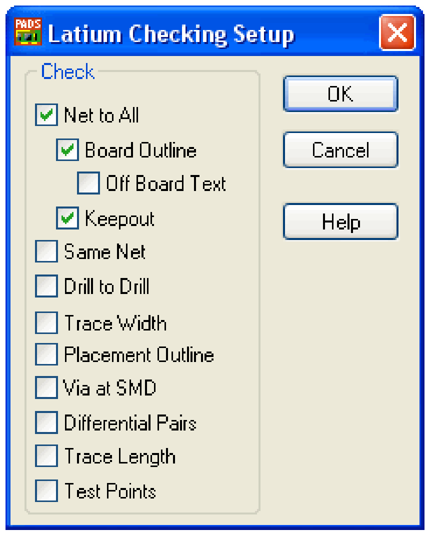


Table 45-196. Latium Checking Setup Dialog Box contents

Name	Description
Net to All	Checks clearance rules on each net or hierarchical level against any other obstacle type.
Board Outline	Checks clearance rules for the board outline and board cut outs.
Off Board Text	Checks for off-board text and to flag all instances of off-board text as clearance errors.
Keepout	Checks for keepout restriction violations.
Same Net	Checks clearances between objects along the same net, as specified in the Clearance Rules dialog box.
Drill to Drill	Checks clearances between all drill holes. Pad stack drill size plus the drill oversize value calculate the diameter for plated holes. **Tip:** Drill to drill errors are reported for only one layer in a drill pair.

Table 45-196. Latium Checking Setup Dialog Box contents (cont.)

Name	Description
Trace Width	Checks traces in excess of minimum and maximum widths, specified in the Clearance Rules dialog box.
Placement Outline	• In default layer mode, check outline against outline on layer 20, not on electrical layers. • In increased layer mode, check outline against outline on layer 120. **Tip:** You can create outlines on layer 20 (or 120) that do not exactly match the actual component outline. By setting a larger outline on this layer, you can leave an area near a component open for other purposes. This check ensures this area is left open.
Via at SMD	Checks for via at SMD restriction violations.
Differential Pairs	Checks for differential pair restriction violations.
Trace Length	Checks for length restriction violations.
Test Point	Checks test points on the design. Test Points checks for probe clearances, minimum via/pad sizes for probing, SMD pin probing, test points on component pin on the component side, test point count per net settings and nail diameter settings, and compares the settings against the setting in the DFT Audit program.

Related Topics

Setting Up Latium Checking

Layer Association Dialog Box

Use the Layer Association dialog box to specify the layer to use for the CAM Document from among all the layers available for the document type that you choose.

Accessing

- Add/Edit CAM Document dialog box > select a document type from the Document Type list (not Custom)

Figure 45-214. Layer Association Dialog Box

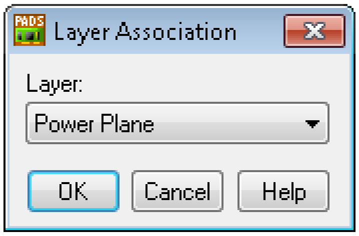


Table 45-197. Layer Association Dialog Box Contents

Name	Description
Layer	Select the layer to use for the CAM Document Type you have chosen to create. **Restriction:** This list displays only the layers that are available for the type of CAM document you are creating.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Layer Thickness Dialog Box

Use the Layer Thickness dialog box to define electrical layer and dielectric material layer thickness and dielectric constant information. When you verify your design, the electrodynamic check uses this information.

Traces on high-speed printed circuit boards can act like transmission lines that “broadcast” interference to adjacent conductors. Using the high-speed rules module, you can use Rules to set clearances on a net class, net, or pin to pin connection basis; then use high-speed checking to report on properties such as impedance, delay, track length, daisy chaining, and parallel routing. These issues cause interference and create costly problems in prototyping. You can run checks against the entire board or against specific nets.

Requirements

- Set these definitions before you run an electrodynamic check.
- You must have specified your plane layers in the Layers Setup dialog box before you run an electrodynamic check. For a two-layer board, temporarily identify one of the layers as a plane layer.

Accessing

- **Setup** menu > **Layer Definition** > **Thickness** button

Figure 45-215. Layer Thickness Dialog Box

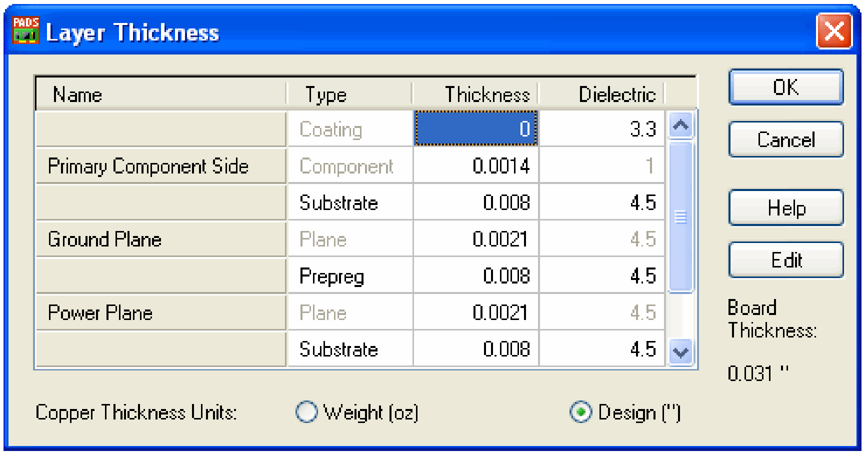


Table 45-198. Layer Thickness Dialog Box Contents

Name	Description
Name	The name of the layer.
Type	Specifies the type of layer. The only layers available for edit are dielectric. You have the choice of Substrate or Prepeg in this list.
Thickness	Specifies the thickness of the layer. **Tip:** If no coating is required, set thickness to zero.
Dielectric	Specifies the dielectric constant value.
Edit	Makes the selected cell available for editing. **Exception:** Cells that are grayed out are unavailable for editing.
Weight (oz)	Specifies to view and edit copper thicknesses by ounces per square foot.
Design (")	Specifies to view and edit copper thicknesses in the same unit of measure as the current database.
Board Thickness	The total value of material and layer thicknesses in the current design units.

Related Topics

Setting Layer Thickness

Layers Setup Dialog Box

Use the Layers Setup dialog box to define each layer.

Accessing

- **Setup** menu > **Layer Definition**

Figure 45-216. Layers Setup Dialog Box

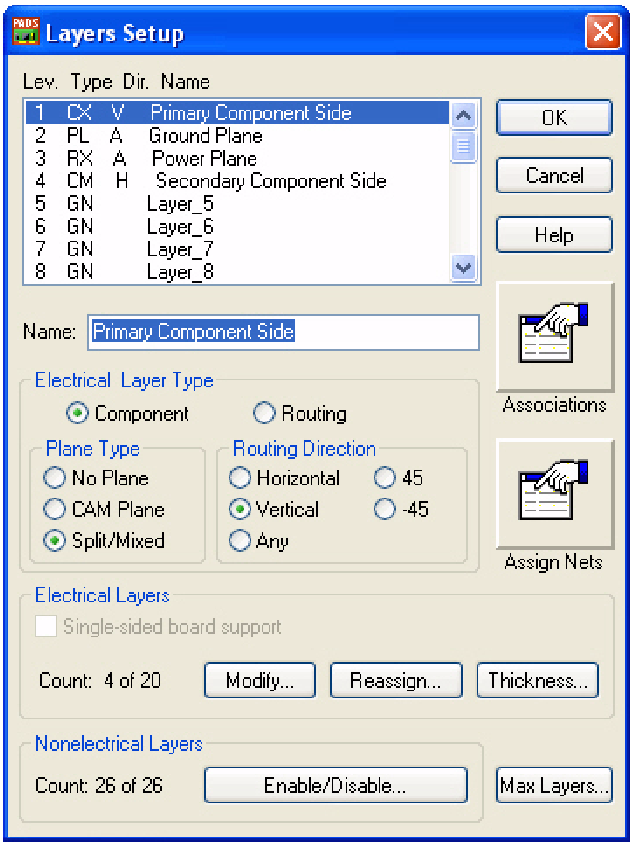


Table 45-199. Layers Setup Dialog Box Contents

<table>
<tr><th>Name</th><th>Description</th></tr>
<tr><td>Layers list</td><td>Level column—Displays the layer number. Layer one and the last electrical layer are used for components and routing and are automatically assigned as component layers for the top and bottom of the board. When you add additional electrical layers the last electrical layer becomes the new bottom layer.
See also: Modifying the Number of Electrical PCB Layers

Layers in the list below the last electrical layer are nonelectrical or documentation layers. Apply text or drafting lines for specific purposes like assembly drawings or solder or paste mask output to these layers.
Type column—Identifies the layer type:
CM—Component and No Plane
RT—Routing and No Plane
PL—Routing and CAM Plane
CP—Component and CAM Plane
CX—Component and Split/Mixed Plane
RX—Routing and Split/Mixed Plane
Direction column—Identifies the specified routing direction, Horizontal (H), Vertical (V), Any (A), 45 (/), or -45 (\).
Name column—Specifies the name of the selected layer.</td></tr>
<tr><td>Name</td><td>The name for the layer which defines its function. You can assign a unique name to each layer to identify it. This name appears in the Name column of the Layers list and in the Layer list in the standard toolbar.</td></tr>
</table>

Table 45-199. Layers Setup Dialog Box Contents (cont.)

Name	Description
Electrical Layer Type area	The Component and Routing options are available only when you've selected a top or bottom layer. If you want to place components on the selected outer layer, you must select Component. • **Component**—Sets the layer as a placement and routing layer. When a top or bottom layer is set as a component layer, you can use the Associations button to "map" (called associating) which documentation layers go with the selected layer. • **Routing**—Sets the layer as a routing layer. If you want to prevent the placement of components on an outer layer, select Routing; you cannot place components on this layer. **Restriction:** You can set only the top and bottom layers as component layers, indicating they will be used for placement; you cannot set inner layers as Component layers. All electrical layers can be set to routing, plane, or split/mixed plane layers.

Table 45-199. Layers Setup Dialog Box Contents (cont.)

Name	Description
Plane Type area	Assigns a layer type to the layer selected in the Layers list. When a layer is selected as a plane or split/mixed plane layer, use the Assign Nets button to assign which nets to connect to it. The available types are: • **No Plane**—Prevents planes from being added to the layer. The No Plane layer is available for routing. If you select No Plane, you can only create Copper and Copper Pour areas on the layer. • **CAM Plane**—Sets the entire layer to be solid copper and connected to only one net. The CAM Plane layer is a negative image, and the copper does not appear in the design as it normally does for all other copper objects. You can not manipulate the shape/outline of the copper on this layer since it is generated automatically and covers the entire layer. This is an outmoded layer type. You can not route traces on a CAM Plane layer. Copper Pours and Plane areas can not be created on CAM Plane layers. • **Split/Mixed Plane—**Enables one or more planes on the layer, and enables routing on the layer. Routes can be placed within or without plane areas. Plane areas avoid traces within their outline by a clearance area defined in the design rules. Copper Pours can not be placed on Split/Mixed layers. Plane areas are created on Split/Mixed plane layers and are similar to but more feature-packed than Copper Pours. **Tip:** If a No Plane layer is changed to a Split/Mixed or CAM Plane layer, nets belonging to the new plane are excluded from associated nets.
Routing Direction area	You must assign a primary routing direction to all electrical layers. Choose from: • Horizontal • Vertical • Any • positive 45 degrees • negative 45 degrees **Tip:** Nonelectrical layers are not assigned a routing direction. The routing direction affects the manual and autorouting performance. For example, if you select Horizontal but most of the traces on the layer need to be vertical, route editing performance is slow. Also, selecting Any can adversely affect route editing performance.

Table 45-199. Layers Setup Dialog Box Contents (cont.)

Name	Description
Associations	Opens the Component Layer Associations Dialog Box. **Tip:** The Associations button appears when you select a component layer in the Layers list.
Assign Nets	Opens the Plane Layer Nets dialog box. **Tip:** The Assign Nets button appears when you select the Ground or Power layer in the Layers list.
Single-sided board	Specifies the following: • Connectivity checking will not report connectivity errors for component pins with non-plated drill holes. Components and jumpers placed on the top layer are considered as connected to pads on the bottom layer with solder joints. • In CAM output, all through-hole pins and vias are treated as non-plated regardless of definition in pad stacks. **Tips:** • Available only for boards with two electrical layers. • When selected, Modify button is unavailable. **See also:** Designating a Board as Single-sided
Modify	Opens the Modify Electrical Layer Count dialog box. **See also:** Modifying the Number of Electrical PCB Layers
Reassign	Opens the Reassign Electrical Layers dialog box. **See also:** Reassigning Electrical Layers
Thickness	Opens the Layer Thickness dialog box. **See also:** Setting Layer Thickness
Enable/Disable	Opens the Enable/Disable Layers dialog box. **See also:** Hiding or Displaying Non-electrical Layers
Max Layers	Opens the Increase Maximum Layer Number dialog box. **Tip:** When you change to increased layer mode, layer numbers for all nonelectric layers increase by 100. **See also:** Increasing the Maximum Number of Available Layers

Related Topics

Setting Up an Outer Layer

Setting Up an Inner Layer

Setting Up a Documentation Layer

Layer Modes in the *Concepts Guide*

Leader Segment Properties Dialog Box

The Leader Segment Properties dialog box displays coordinate information for the selected dimensioning arrow and provides several areas for modifications.

Requirement: You must select the first segment of the leader, the section with the arrow, to display this dialog box.

The Leader Segment Properties dialog box remains open until you click OK or Cancel. Selecting another leader segment while the dialog box is open updates the information for the selected object.

Accessing

- Select a dimensioning leader segment > right-click > **Properties**

Figure 45-217. Leader Segment Properties Dialog Box

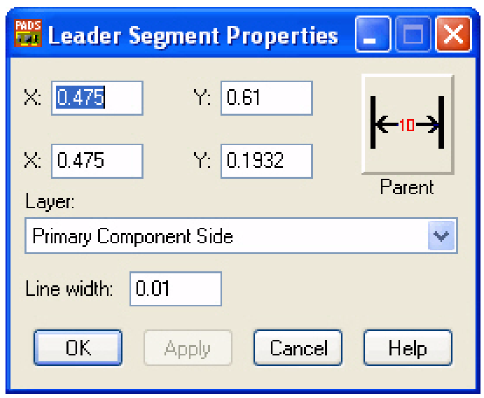


Table 45-200. Leader Segment Properties Dialog Box contents

Name	Description
X and Y	Lists the x and y coordinates of the dimension object. The coordinates are calculated from the bottom of one of the extension lines or from the radius point of an arc. Type new values to change the location.
Parent button	Opens the Dimension Properties dialog box for the dimension object with which the selected object is associated.
Layer list	Lists the current working layer. Select a new layer from the list.
Line Width	Lists the current line width used for the dimension object. Type a new value to change the line width.

Related Topics

Dimensioning Process

Library List Dialog Box

Use the Library List dialog box to specify the libraries available to the design, library search order, and other search-related options. Operations in this dialog box affect the contents of the Library list in the Library Manager dialog box.

Accessing

- **File** menu > **Library** > **Manage Lib. List** button

Figure 45-218. Library List Dialog Box

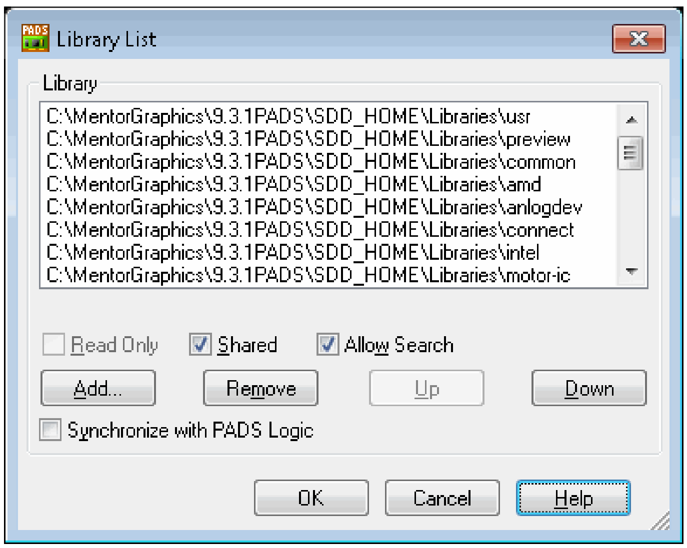


Table 45-201. Library List Dialog Box Contents

Name	Description
Library list	The libraries currently listed in the Library Manager Library list.
Read Only	A status indicator only; this box is always unavailable.
Shared	Shares the library over the network. This enables more than one user to access the library file at the same time.
Allow Search	Includes the library when performing operations that involves libraries, such as adding parts.
Add	Adds a library to the Library list.

Table 45-201. Library List Dialog Box Contents (cont.)

Name	Description
Remove	Removes a library from the Library list.
Up/Down	Moves the order of the libraries in the Library list.
Synchronize with PADS Logic	Specifies to push the library settings from PADS Layout to PADS Logic.

Related Topics

Setting Library Availability and Search Options

Library Manager Dialog Box

Use the Library Manager dialog box to create libraries, to display the contents of libraries, and to manage the contents of libraries.

Tips:

- The Library Manager supports up to 65,536 components.
- The picture in the Library Manager dialog box displays the selected item for decals, CAE decals, and lines. For parts, the picture displays the first-assigned decal.

Accessing

- **File** menu > **Library**

Figure 45-219. Library Manager Dialog Box

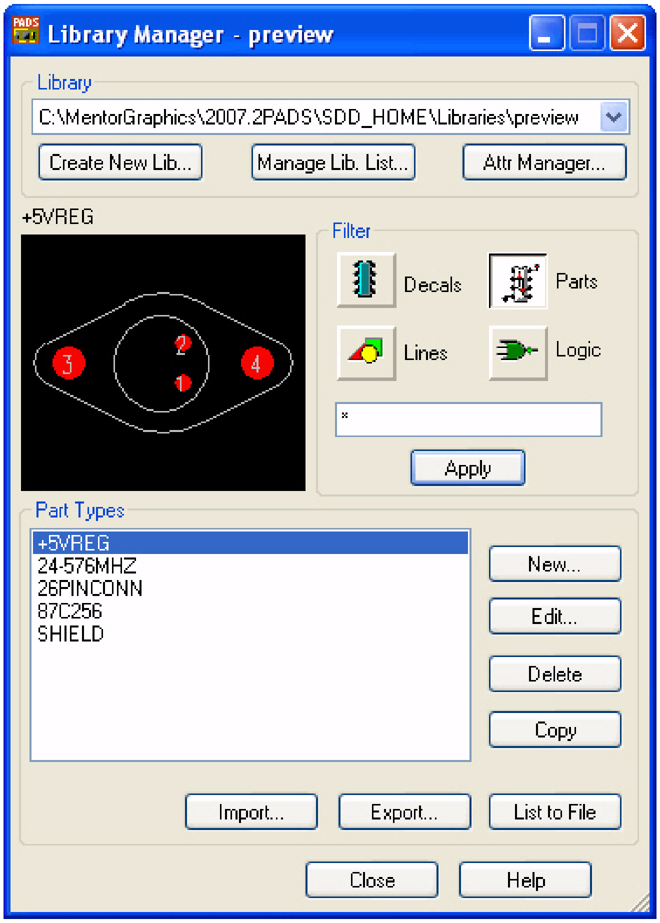


Table 45-202. Library Manager Dialog Box Contents

Name	Description
Library list	The list of libraries available to you.
Create New Lib	Opens the New Library window where you can specify a new library name and location.
Manage Lib. List	Opens the Library List Dialog Box.
Attr Manager	Opens the Manage Library Attributes Dialog Box.
Preview area	Shows the item selected in the Filter list.
Filter area	Narrows down the Filter list by Decals, Parts, Lines, or Logic. You can further narrow the list using wildcards in the Filter box. **Tip: Add an asterisk "*" to the box to display all items.**
Filter list	The results from your filter area selections.

Table 45-202. Library Manager Dialog Box Contents (cont.)

Name	Description
New	The action taken is dependent on the filter. • **Decals**—Opens the PCB Decal Editor on a new decal. • **Parts**—Opens the Part Information Dialog Box, Gates Tab on an unnamed part. • **Lines**—Unavailable. There is no special library lines editor. Use drafting tools to create or edit lines and save them to the library. **See also:** Drafting • **Logic**—Unavailable. Use PADS Logic to create or edit CAE decals. **Restriction:** This button is unavailable when the Library is set to *All Libraries*. **See also:** Adding Items to a Library
Edit	The action taken is dependent on the filter. • **Decals**—Opens the PCB Decal Editor on the selected decal. • **Parts**—Opens the Part Information Dialog Box, Gates Tab on the selected part. • **Lines**—Unavailable. There is no special library lines editor. Use drafting tools to create or edit lines and save them to the library. **See also:** Drafting • **Logic**—Unavailable. Use PADS Logic to create or edit CAE decals. **Restriction:** This button is unavailable when the Library is set to *All Libraries*. **See also:** Editing Items in a Library
Delete	Removes the selected item from the library. **Restriction:** This button is unavailable when the Library is set to *All Libraries*. **See also:** Deleting Items from a Library
Copy	Copies the selected item to another name or another library. **Restriction:** This button is unavailable when the Library is set to *All Libraries*. **See also:** Copying a Library Item
Import	Import library data from an ASCII file. The file type is dependent on the filter. **Restriction:** This button is unavailable when the Library is set to *All Libraries*. **See also:** Importing Library Data

Table 45-202. Library Manager Dialog Box Contents (cont.)

Name	Description
Export	Export library data to an ASCII file. The file type is dependent on the filter. **Restriction:** This button is unavailable when the Library is set to *All Libraries*. **See also:** Exporting Library Data
List to File	The action taken is dependent on the filter. • **Decals**—Generates a list of PCB Decals in a single library. • **Parts**—Generates a list of Parts in a single library or all libraries along with chosen attributes. • **Lines**—Generates a list of line items in a single library. • **Logic**—Generates a list of CAE decals or Logic symbols in a single library. **Restriction:** When the Library is set to All Libraries, this button is unavailable for all but Parts.

Related Topics

Creating a Library

Setting Library Availability and Search Options

Deleting All Items in a Library

Managing Library Attributes

Creating a Report of the Parts in a Library

Creating a Report of Decals, Lines or Logic Symbols in a Library

Log Test Dialog Box

Use BLT to replay session playback media created by BMW.

Accessing

- type **BLT** > press **Enter**

Tip: If nothing happens, close PADS Layout and restart it.

Figure 45-220. Log Test Dialog Box

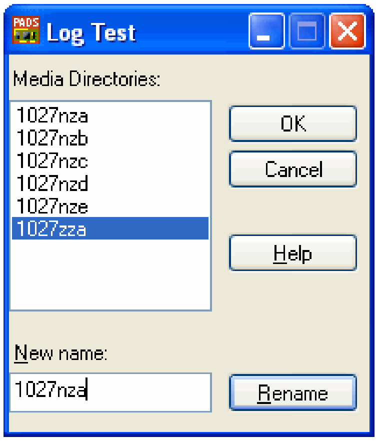


Table 45-203. Log Test Dialog Box contents

Name	Description
Media Directories	Lists the session playback media files.
New name	Specifies to rename the selected media directory.
Rename	Renames the selected media directory to the name in the New name box.

Related Topics

Replaying Session Playback Media with BLT

Logic Families Dialog Box

Use the Logic Families dialog box to add, delete, or edit logic families, and to specify their reference designator prefix.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **General** tab > **Families** button

Figure 45-221. Logic Families Dialog Box

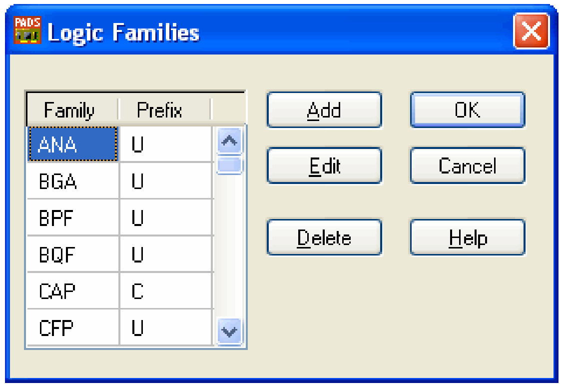


Table 45-204. Logic Families Dialog Box Contents

Name	Description
Family	The name of the Logic family. **Restriction:** The logic family name is limited to 3 character.
Prefix	The prefix of the Logic family. Multiple characters are allowed.
Add	Inserts a row at the bottom of the list where you can add a new Logic family.
Edit	Makes the selected cell available for editing. **Tip:** You can also double-click a cell to edit the contents.
Delete	Removes the selected row.

Make Reuse Dialog Box

Use the Make Reuse dialog box to specify a reuse type and reuse name for the physical design reuse you are creating.

Accessing

- **Select the objects you want to include > Right-click > Make Reuse**

Figure 45-222. Make Reuse Dialog Box

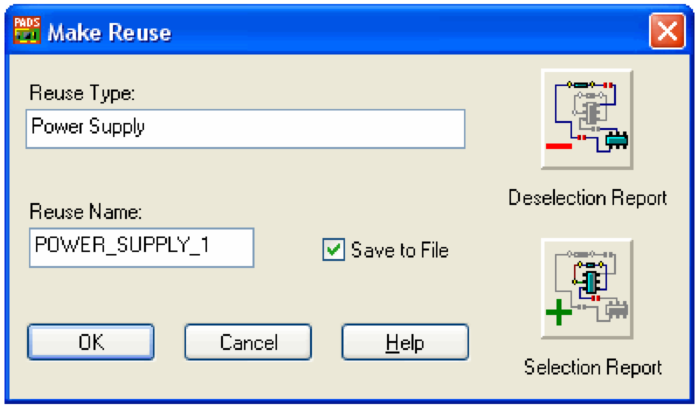


Table 45-205. Make Reuse Dialog Box contents

Name	Description
Reuse Type	Describes the physical design reuse and its function. The reuse type is similar to a library part type. Type a reuse type, up to 255 characters long, for the physical design reuse. Illegal characters are slashes (\ /), colon (:), asterisk (*), question mark (?), quotation marks ("), less than and greater than signs (< >), and pipe (|). You can include spaces in the type, but you can't use them as the first or last character. The reuse type is checked to ensure it is not already in use. If it is already used, an error message appears and you can specify a different type. **Tip:** There is no way to display the reuse name and reuse type in the design.
Reuse Name	Indicates a name that uniquely identifies this instance of the physical design reuse. Type a name, up to 15 characters long, for the physical design reuse. Illegal characters are comma (,), brackets ({ }), asterisk (*), period (.), and space. The default name is based on the reuse type. For example, if the reuse type is Power Supply, the default reuse name is Power_Supply_1. The reuse name is checked to ensure it is not already in use. If it is already used, an error message appears and you can specify a different name. **Tip:** There is no way to display the reuse name and reuse type in the design.

Table 45-205. Make Reuse Dialog Box contents (cont.)

Name	Description
Save to File	Saves the physical design reuse to a file for use in another design. The content of the current start-up file is also saved with the physical design reuse. **See also:** Creating Start-up Files If this is selected, the Reuse Save As dialog box appears when you click OK. Indicate the folder in which to save the newly created physical design reuse. The default folder is \PADS Projects\Reuse. Reuse files have a .reu extension and can be opened by clicking Open from the File menu. You must change the file type to **PADS Layout Reuse Files** (*.reu).
Deselection Report	Creates the report file report.rep in the *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs* folder and opens the file in the default text editor. The file contains a list of items removed from the selection because they were not valid for inclusion in the physical design reuse. **Tip:** The deselection report and the selection report are created using the same file name. If you want to save this file, do so in the default text editor using a different file name.
Selection Report	Creates the report file report.rep in the *C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs* folder and opens the file in the default text editor. The file contains a list of items included in the physical design reuse. **Tip:** The deselection report and the selection report are created using the same file name. If you want to save this file, do so in the default text editor using a different file name.

Related Topics

Reusing Designs or Parts of Designs

Manage Library Attributes Dialog Box

Use the Manage Library Attributes dialog box to manage attributes on a library-by-library basis. You can add, delete, and rename attributes for all parts or decals in an individual library or in all libraries. You can also display all the attributes in a library, whether the attributes apply to all items or to individual items.

Accessing

- **File** menu > **Library** > **Attr Manager** button

Figure 45-223. Manage Library Attributes Dialog Box

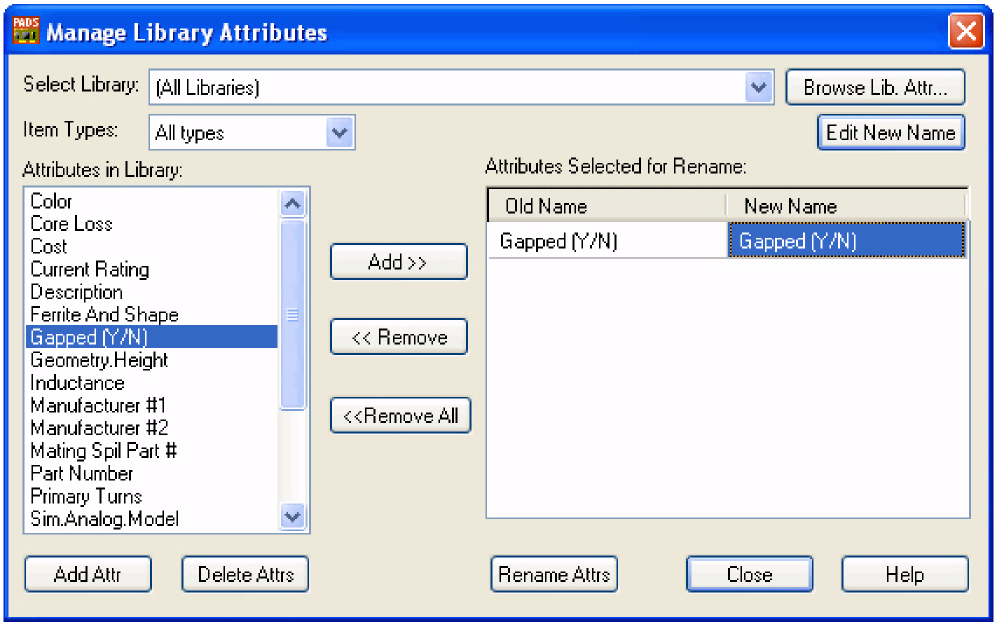


Table 45-206. Manage Library Attributes Dialog Box Contents

Name	Description
Select Library list	The list of libraries available to you.
Item Types list	Filters the type of items in the Attributes in Library list.
Browse Lib. Attr	Opens the Browse Library Attributes Dialog Box.
Edit New Name	Makes the selected attribute Name editable. **Tip:** This is available only when an attribute in the New Name column in the Attributes Selected for Rename list is selected.
Attributes in Library list	The list of attributes in the selected library. **Tip:** This is available only when an attribute in the New Name column in the Attributes Selected for Rename list is selected.
Add >>	Adds the selected attribute to the Rename list.
<< Remove	Removes the selected attribute from the Rename list.
<< Remove All	Removes all of the attributes from the Rename list.
Attributes Selected for Rename list	The list of attributes you've selected to rename.
Add Attr	Opens the Add New Attribute to Library Dialog Box.

Table 45-206. Manage Library Attributes Dialog Box Contents (cont.)

Name	Description
Delete Attrs	Deletes the selected attribute from the selected library.
Rename Attrs	Renames all of the attributes you gave a new name to in the selected library.

Markups Dialog Box

Use the Markups dialog box to view or log issues concerning the design. You can add 2D line markups to issues, to outline or highlight their location. You can also link design objects to markups. Then you can export the issues alone or export the design and any issues for viewing and logging additional information in visECAD.

Accessing

- **Edit** menu > **Markups**

Figure 45-224. Markups Dialog Box

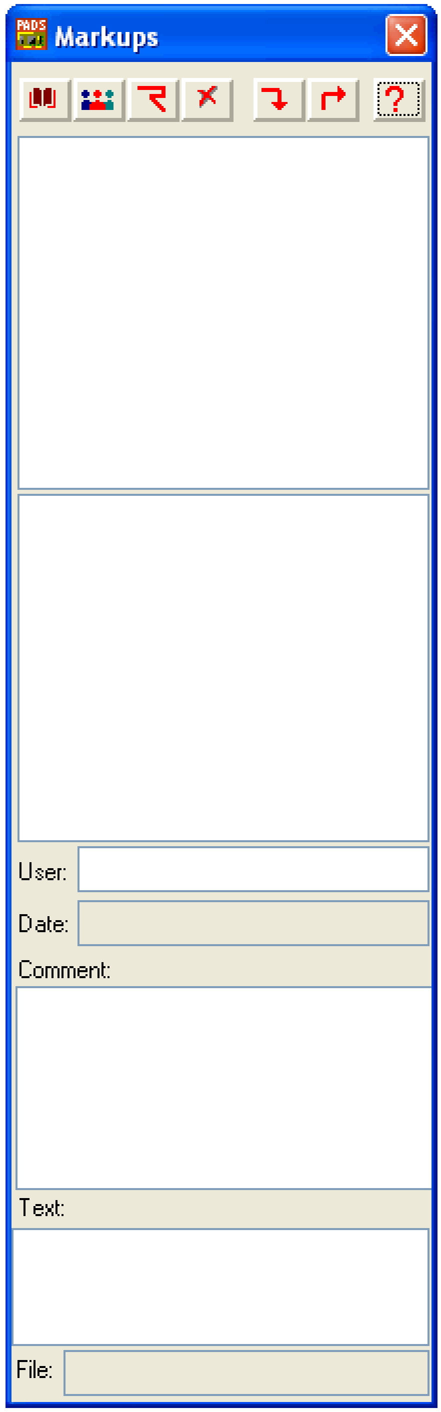


Table 45-207. Markups Dialog Box Contents

Name	Description
Add Topic	Click to add a new topic to the Collaboration Data tree.
Add Issue	Click to add a new issue to the tree under the currently active topic
Add Markup	Click to add a new markup to the currently active issue.
Delete	Delete the active item in the Collaboration Data tree
Import	Click to browse for, and Import a collaboration data file (*.clb, or *.cle).
Export	Click to Export the encrypted collaboration data to a file name of your choosing (*.cle).
Collaboration Data Tree	This is the top tree in the dialog box and it displays the collaboration data in a hierarchical form.
Elements Tree	This lists the elements linked to any markup.
User	Type your name in this box to associate your name with the active Collaboration Tree item.
Date	Displays the date and time of creation of the currently active collaboration item.
Comment	Type to add comments for the currently active collaboration item.
Text	This displays the text of a markup. This is unavailable in PADS Layout, but will display text associated with markups created in other software.
File	When you export or import a collaboration file, this displays the path and filename.

Related Topics

Adding Markups

Importing Markups

Exporting Markups

Media Wizard Dialog Box

BMW (Basic Media Wizard) is a tool that you can use to record and play back PADS Logic, PADS Layout and PADS Router sessions. It is particularly useful as a means of supplying information to PADS Technical Support engineers trying to identify and resolve any problematical behavior you may encounter.

Accessing

- type **BMW** > press **Enter**

Figure 45-225. Media Wizard Dialog Box

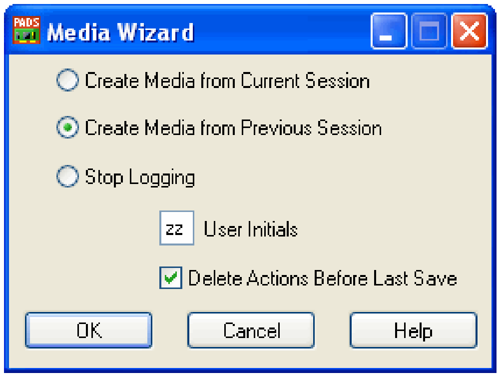


Table 45-208. Media Wizard Dialog Box contents

Name	Description
Media Wizard area	Specifies what you want the Media Wizard to do: • **Create Media from Current Session—**Use this procedure when the session you are recreating did not cause a PADS tool crash. • **Create Media from Previous Session—**Use this procedure when the session you are recreating caused the PADS tool to crash, and the automatic procedure described in Automatically Creating Session Playback Media for a Crashed Session cannot be used due to one of the restrictions listed in that section. • **Stop Logging**—Specifies to stop the Media Wizard from logging any further actions.
User Initials	Specifies your initials. They are included in the playback media filenames to identify the files as yours.
Delete Actions Before Last Save	Specifies to delete all entries in the session log file between the first Open and the last Save command. You can do this to eliminate any actions you may have performed before beginning the series of actions that produced the problematical behavior. This makes it easier for the Tech Support engineer to identify the problem.

Related Topics

BMW and BLT

Missing Height Dialog Box

Use the Missing Height dialog box to specify missing heights when exporting the design to IDF.

Accessing

The Missing Height dialog box appears when the Geometry.Height attribute does not exist or is set to zero height for any part type and decal pair exported to IDF.

Figure 45-226. Missing Height Dialog Box

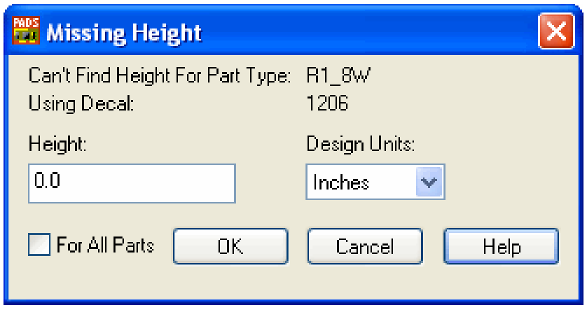


Table 45-209. Missing Height Dialog Box contents

Name	Description
Can't Find Height For Part Type	Displays the part type with the missing height.
Using Decal	Displays the decal associated with the part type.
Height	Specifies the package and mounting height for the part type and decal pair. **Tip:** If you specify the height as zero, the mechanical design system may prompt you to enter a height when you import the IDF file.
Design Units	Lists the design units.
For All Parts	Specifies to apply the height for all part type and decal pairs.

Related Topics

Specifying Missing Heights During Export to IDF

Mixed Plane Setup Dialog Box

Use the Mixed Plane Setup dialog box to set the type of plane checking.

Accessing

- **Tools menu > Verify Design > Plane check > Setup button**

Figure 45-227. Mixed Plane Setup Dialog Box

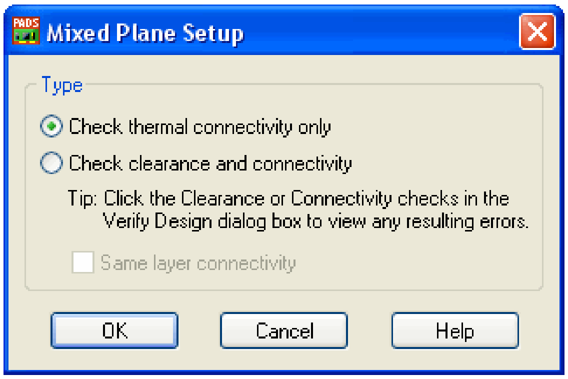


Table 45-210. Mixed Plane Setup Dialog Box contents

Name	Description
Type area	• **Check thermal connectivity only—Specifies to** check your design for split/mixed or CAM plane thermal connectivity. Use this check to find pins or vias that do not have the thermal attribute set, or to find pins that are not within a plane area (thermals will not connect). You can set the thermal attribute in Jumper Pin, Pin, or Via Properties dialog boxes. • **Check clearance and connectivity—Specifies to** check your design for split/mixed plane clearance and net connectivity errors. If the split/mixed plane is not connected in the design, a Plane Connect will be performed before the check is run.
Same Layer Connectivity	Ensures that plane areas are continuous on the split/mixed plane layer. Plane areas of a particular net must have copper contact with each other without going to another layer.

Related Topics

Setting Up Plane Checking

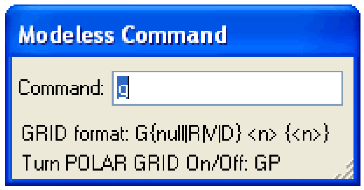
Modeless Command Dialog Box

You can set or change some settings and functions at any time using a shortcut key for the command called a modeless command.

Accessing

- Type the modeless command (shortcut key) for the command you want.

Figure 45-228. Modeless Command Dialog Box



Modeless Commands

The following is a complete listing of all the modeless commands. In the table:

<X,Y> = coordinates

<s> = text

<n> = number.

Table 45-211. Modeless Commands

Shortcut key	Description
C	Complementary format. Type C and press Enter to change the display to a complementary format, which views thermals and antipads on plane layers. Type C a second time and press Enter to restore the normal non complementary view.
D	Display last current layer on/off.
DO	Drill outline on/off.
E	Cycles through the End Via modes: • End No Via (where a trace ends in space) • End Via (where a trace ends with a via) • End Test Point (where a trace ends with a test point via
I	Database Integrity Test.
L <n>	Change the current layer to <n>. <n> can be a number or name, for example (L 2) or (L top).

Table 45-211. Modeless Commands (cont.)

Shortcut key	Description
N <s>	Highlights nets one by one. To place the highlighted net <s> at the top of the stack, repeat the command, for example, N GND.
N-	Unhighlights each highlighted net in reverse order.
N	Removes all highlighting from the nets.
NN	Toggles the visibility of net names. **Tips:** • This command toggles the Net Name check box (object type - column) in the Display Colors Setup Dialog Box. • When the Net Name column check box is enabled, net name visibility is still restricted by the color tiles on each layer and the *Show net names on Traces, Vias, Pins* check boxes. See also the NNP, NNT, and NNV modeless commands. • Sizing and frequency of placement is controlled by the settings in the Display Options.
NNP	Toggles the display of net names on pins. **Tip:** This command toggles the Net names on...Pins check box in the Display Colors Setup Dialog Box. Sizing is controlled by the settings in the Display Options. **Restriction:** The display of net names also depends on the Net Name visibility check box and the presence of a non-background color tile on the layer in the Display Colors Setup Dialog Box.
NNT	Toggles the display of net names on traces. **Tip:** This command toggles the Net names on...Traces check box in the Display Colors Setup Dialog Box. Sizing and frequency of placement is controlled by the settings in the Display Options. **Restriction:** The display of net names also depends on the Net Name visibility check box and the presence of a non-background color tile on the layer in the Display Colors Setup Dialog Box.
NNV	Toggles the display of net names on vias. **Tip:** This command toggles the Net names on...Vias check box in the Display Colors Setup Dialog Box. Sizing is controlled by the settings in the Display Options. **Restriction:** The display of net names also depends on the Net Name visibility check box and the presence of a non-background color tile on the layer in the Display Colors Setup Dialog Box.
O <r>	Toggles outline mode on and off. Displays an outline of pads, traces and all lines. Traces are displayed at their true width to the limits of the Minimum Display Width setting in the Global / General Options. **See also:** To Use Outline View Mode

Table 45-211. Modeless Commands (cont.)

Shortcut key	Description
OH	Toggles outline mode with High resolution. Displays all objects with their true shape.
OL	Toggles outline mode with Low resolution. Displays rounded objects with squared corners.
PO	Pour outline on/off. This toggles the display mode of copper pours between pour outline and hatch outline. See the SPO modeless command for use with plane areas.
PN	Toggles pin number display on/off. **Tip:** This command toggles the Pin Num. column check box in the Display Colors Setup Dialog Box. Sizing is controlled by the settings in the Display Options.
Q	Quick Measure with a dynamic ruler. Place the cursor at the starting point then type the "q" modeless command. Drag the cursor to create a line between the start and end point of your measurement. Snaps to the design grid when *Snap to grid* is on. Measurements are gridless when Grid Snap is off. Dynamically reports delta x, delta y, and delta x,y in current design units. You can also measure precise Euclidean distances between polar grid nodes using this command. **See also:** Measuring
QL	Quick Length. Area select the route items you want to measure; such as route segments, nets, or pin pairs. Type QL; and press Enter. A report of the route item lengths for routed, unrouted, and total length appears in the default text editor. **See also:** Measuring
R <n>	Reduce display width to <n>, for example, R 50.
RV	Toggles Make Like Reuse operations to compare or ignore value and tolerance attributes. **See also:** To Make a Like Reuse in Verb Mode, To Make a Like Reuse in Object Mode
SO <x> {<y>}	Sets the origin using relative coordinates. If a component, pin, drafting corner, text, via, circle, or line intersection is selected, SO sets the board origin to the origin of the selected item. If nothing is selected, coordinates must be given. For example, SO 25, SO 8.3 or SO 16-2/3, SO 25 25. The second parameter is optional. **See also:** Setting the Design Origin.
SOA <x> {<y>}	Sets the origin using absolute coordinates. Coordinates must be given. For example, SOA 25, SOA 8.3 or SOA 16-2/3, SOA 25 25. The second parameter is optional.

Table 45-211. Modeless Commands (cont.)

Shortcut key	Description
SPD	Display generated plane data for split/mixed planes. This command controls an option in the Options Dialog Box, Split/Mixed Plane Page.
SPI	Display plane thermal indicators. This command controls an option in the Options Dialog Box, Split/Mixed Plane Page.
SPO	Display plane polygon outlines for split/mixed planes. This command controls an option in the Options Dialog Box, Split/Mixed Plane Page.
T	Toggle Transparent Mode. **See also:** To Use Transparent View Mode
UM	Sets Design Units to mils.
UMM	Sets Design Units to millimeters (metric).
UI	Sets Design Units to inches.
X	Text outline on/off.
W <n>	Changes width to <n>, for example W 5.
Grids Commands	
G <x> {<y>}	Grid global setting. The second parameter is optional. Sets the Design and Via grids simultaneously, for example, G 25, G 8.3 or G 16-2/3, G 25 25.
GD <x> {<y>}	Dot grid setting, for example, GD 8-1/3, GD 25 25, or GD 100. The second parameter is optional.
GP	Polar grid on/off. Use the polar grid for Radial Move, circular component arrays, and to create radial drawings.
GP r a	Move to a point specified by polar coordinates (radius and angle).
GPR r	Move to a point specified by the radius, using the existing angle.
GPA a	Move to a point specified by the angle, using the existing radius.
GPRA da	Move to a point specified by the current angle (da) value, using the existing radius.
GPRR dr	Move to a point specified by the current radius (dr), using the existing angle.
GR <xx>	Design grid setting, for example, GR 8-1/3, GR 25 25, GR 25.
GV <xx>	Via grid setting, for example, GV 8-1/3, GV 25 25, or GV 25.

Table 45-211. Modeless Commands (cont.)

Shortcut key	Description
Tips: • - 8 = (eight); - 8-1/3 = (eight and a third); - 8.33 = (eight point 33) • You can search for, and move the cursor to, a location regardless of the current grid. But the Snap to grid settings of the Grids Options restrict you to performing any design activity on the grid.	
Search Commands	
S <s>	Search for reference designator/pin <s>, for example SU 1.1 or SU 1.
S <n> <n>	Search absolute at <n> <n>, for example S 1000 1000.
SR <n> <n>	Search relative X and Y, for example SR -200 100.
SRX <n>	Search relative X, for example SRX 300.
SRY <n>	Search relative Y, for example SRY 400.
SS <s>	Search and Select components by reference designator, for example SS U10. You can also search and select more than one component by typing multiple entries, for example SS U10 U5 R6. **Tip:** Spaces may be important in modeless commands. For example, SS W1 and S SW1 have different meanings. SS W1 tells PADS Layout to search and select W1, while S SW1 tells it to search for SW1.
SS <s>*	You can search and select using an asterisk, *. Type SS, a space, the character you want to search by, and an asterisk. For example, to search and select all components beginning with a C, type SS C*****. All components with a reference designator starting with C are selected. **Tip:** This command is useful for placing parts. For example, you can select all resistors using SS R* and then choose Move Sequential from the pop-up menu to place the selected parts.
SX <n>	Absolute move to <n>, current Y, for example SX 300.
SY <n>	Absolute move to <n>, current X, for example SY 400.
XP	Search for and select route segments using pixels instead of segment width, allowing you to adjust a trace whose corners are less than the width of the trace.
Quick Layer View Commands	
Z	Quick layer view. With no command arguments, Z displays the initial layer view.

Table 45-211. Modeless Commands (cont.)

Shortcut key	Description
Z {+<layer>} {-<layer>}	Add or remove layer from the current set of displayed layers. Examples: • Z +O makes the outside layers visible, but does not change visibility of other layers. • Z -O makes the outside layers invisible, but does not change visibility of other layers. • Z -2 +O makes invisible layer 2 and makes visible the outside layers.
Z <n-m>	View only the range of layers you type. For example, Z 2-4 displays layers 2, 3, and 4. Do not enclose the range with square brackets.
Z <layer n> {<layer m> ...}	View only the layers you type. For example, Z 2 4 d displays layers 2, 4, and the documentation layer.
Z *	View all layers. **Restriction:** Z supports only the asterisk * regular expression.
Z A	View the active layer. If the active layer is changed, it has no effect on the display.
Z ADB	View the Assembly Drawing Bottom layer.
Z ADT	View the Assembly Drawing Top layer.
Z B	View only the bottom layer.
Z C <-C>	View only the current layer. If the active layer is changed, only the new current layer is displayed. Unlike Z A, it puts you in a continuous mode where all layers are hidden except the active layer. When you change layers, the new layer becomes visible and all other layers are hidden. Use Z -C to exit the mode.
Z D	View all documentation layers.
Z E	View all electrical layers.
Z I	View all internal layers.
Z O	View only the outside layers, that is, the top and bottom layers.
Z PMB	View the Paste Mask Bottom.
Z PMT	View the Paste Mask Top.
Z SMB	View the Solder Mask Bottom.
Z SMT	View the Solder Mask Top.
Z SSB	View the Silkscreen Bottom.
Z SST	View the Silkscreen Top.

Table 45-211. Modeless Commands (cont.)

Shortcut key	Description
Z T	View only the top layer.
Z U	View unroutes that are visible on all layers.
Z Z	View all layers.
ZR <name>	Restore a quick layer view configuration. For example, ZR L23 restores the configuration stored as L23. **Tip:** Type ZR and press Enter to see a list of all layer configurations created by ZS in this session. **See also:** ZS.
ZS <name>	Saves the current set of displayed layers as a quick layer view configuration. For example, ZS L23 stores the current configuration as L23. The quick layer view configuration is available until you exit the program. **See also:** ZR.
Angles Commands	
AA	Any angle.
AD	Diagonal angle.
AO	Orthogonal angle.
Undo Commands	
UN [<n>]	Multiple undo command (1-100), <n> is optional. (Ex. UN 2) UN will undo 1 time.
RE [<n>]	Multiple redo command (1-100), <n> is optional. (Ex. RE 2) RE will redo 1 time.
Design Rule Checking (DRC) Commands	
DRP	Prevent.
DRW	Warn.
DRI	Ignore Clearance.
DRO	Turn off DRC Mode.
Routing Commands	
E	Toggle between end via, end no via, and end test point modes.
LD	Toggle between horizontal and vertical routing layer direction on the current layer.
PL <n> <n>	Paired Layer command, where <n> can be layer numbers or layer names. (for example, PL 1 2 or PL top bottom).
SH	Toggles Shove mode on or off.

Table 45-211. Modeless Commands (cont.)

Shortcut key	Description
V	Opens the Vias dialog box.
VA	Automatic via and mode selection.
VP	Use partial via.
VT <name>	Use through hole via. If multiple vias exist, you can also type the name of the via to use.
T	Toggle Transparent Mode.
Drafting Objects Commands	
HC	Circle shape draw mode.
HH	Path shape draw mode.
HP	Polygon shape draw mode.
HR	Rectangle shape draw mode.
Other Commands	
?	Show this help topic.
BMW	Opens the Basic Media Wizard dialog box. • BMW records session playback media for a problematic PADS Logic, PADS Layout, or PADS Router session. It can create playback media based on your last PADS session or your current session. This playback media can be replayed using the BLT modeless command. • BMW is also a command line option. **See also:** BMW and BLT
BMW ON	Starts BMW session logging.
BMW OFF	Stops BMW session logging.
BLT	Basic Log Test. Opens the Log Test Dialog Box. BLT finds and runs BMW session playback media. **See also:** BMW and BLT
F <s>	Open file <s>, where <s> is the path and name of the file to open.

Related Topics

Typing Modeless Commands

Shortcut Keys

Modify Electrical Layer Count Dialog Box

Use the Modify Electrical Layer Count dialog box to increase or decrease the number of electrical layers in the design.

Accessing

- **Setup** menu > **Layer Definition** > **Modify** button

Figure 45-229. Modify Electrical Layer Count Dialog Box

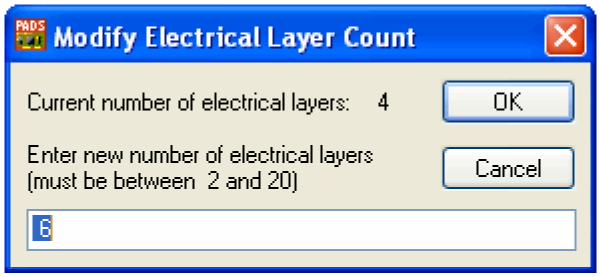


Table 45-212. Modify Electrical Layer Count Dialog Box Contents

Name	Description
Current number of electrical layers	Lists the current number of electrical layer.
Enter new number of electrical layers (must be between _ and _)	Type a new number of electrical layers. You can add or delete layers. The number must be between the two values above the text box. **Restriction:** You can’t delete layers that have data on them. For example, you want to reduce a 2 layer board to a 4 layer board. if you have 4 layers and they all have content, you must move the content from the layers to be deleted. The statement above the text box will state “must be between 4 and 20” since you can’t delete any layers because of data.

Related Topics

Modifying the Number of Electrical PCB Layers

Reassigning Electrical Layers

Reassign Electrical Layers Dialog Box

Layers Setup Dialog Box

NC Drill Options Dialog Box

Use the NC Drill Options dialog box to define NC Drill Output plotting options when adding an NC drill document or editing an existing one.

Accessing

- **File** menu > **CAM** > **Add** button > select **NC Drill** from the **Document Type** list > **Drill** button > **Options** button

  or

- **File** menu > **CAM** > select a document name > **Edit** button > select **NC Drill** from the **Document Type** list > **Drill** button > **Options** button

Figure 45-230. NC Drill Options Dialog Box

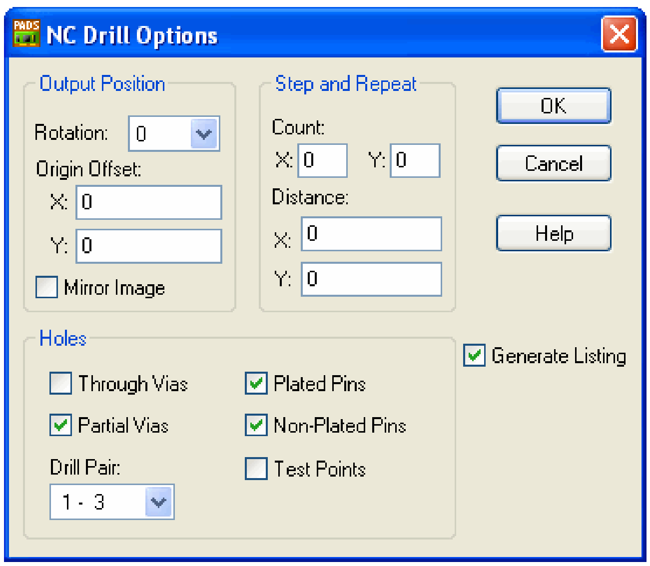


Table 45-213. NC Drill Options Dialog Box contents

Name	Description
Rotation	Define how to rotate the drill drawing. Your choices are 0, 90, 180, or 270 degrees of counterclockwise rotation.
Origin Offset X/Y	Specifies how much to move the board so the design origin coincides with the drill machine origin. A positive X shift moves the board to the right: a positive Y shift moves the board up.
Mirror Image	Specifies to mirror the image.
Count X/Y	Specifies how many times the Pattern is repeated.

Table 45-213. NC Drill Options Dialog Box contents (cont.)

Name	Description
Distance X/Y	Specifies the distance between adjacent patterns; the distances should be greater than or equal to the board dimensions to avoid any overlap. If all box values are zero, then there is no step and repeat.
Through vias	Specifies to drill through vias.
Partial Vias	Specifies to drill partial vias. **Restriction:** Available only if you've set up Drill Pairs.
Drill Pair	Specifies which layer pair to drill. **Restriction:** Available only if you've specified to drill partial vias.
Plated Pins	Specifies to drill plated pins.
Non-Plated Pins	Specifies to drill non-plated pins. Non-plated pins are usually mounting holes.
Test Points	Specifies to plot test point locations.
Generate Listing	Creates a listing of the drill hole sizes by location. An additional CAM file is produced with a .lst extension when you run the NC Drill CAM Document.

Related Topics

Creating an NC Drill File

Creating CAM Outputs to Manufacture Your PCB

NC Drill Setup Dialog Box

Use the NC Drill Setup dialog box to set Excellon output or Drill Listing parameters.

Accessing

- **File** menu > **CAM** > **Add** button > select **NC Drill** from the **Document Type** list > **Drill** button > **Device Setup** button

  or

- **File** menu > **CAM** > select a document name > **Edit** button > > select **NC Drill** from the **Document Type** list > **Drill** button > **Device Setup** button

Figure 45-231. Excellon Tab

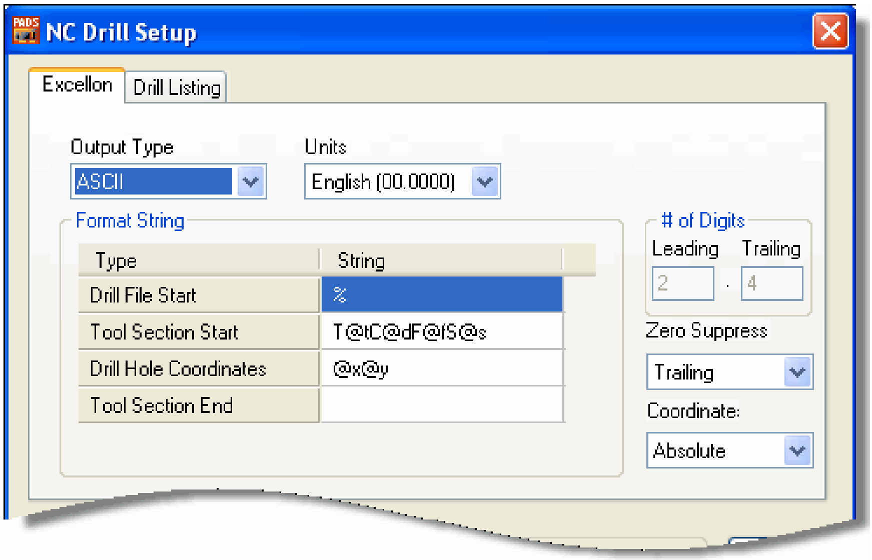


Figure 45-232. Drill Listing Tab

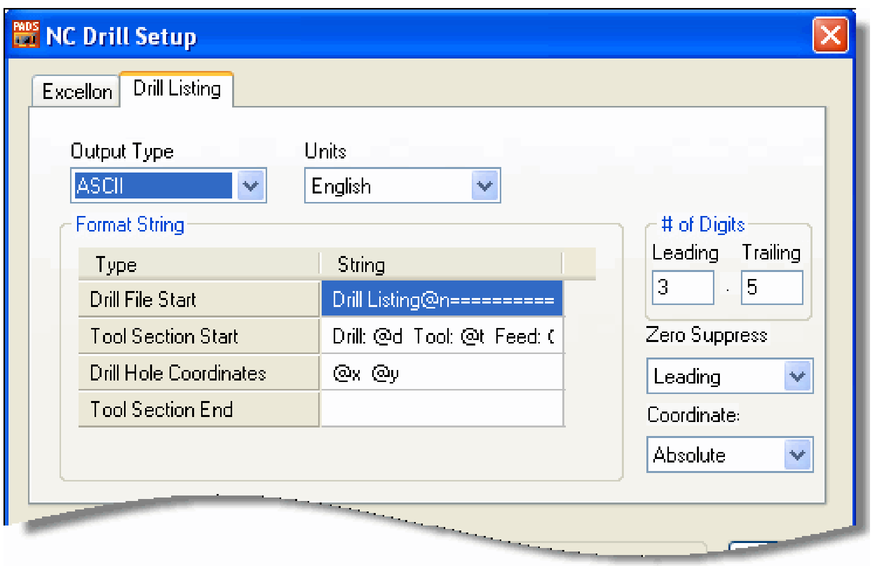


Figure 45-233. NC Drill Setup Dialog Box

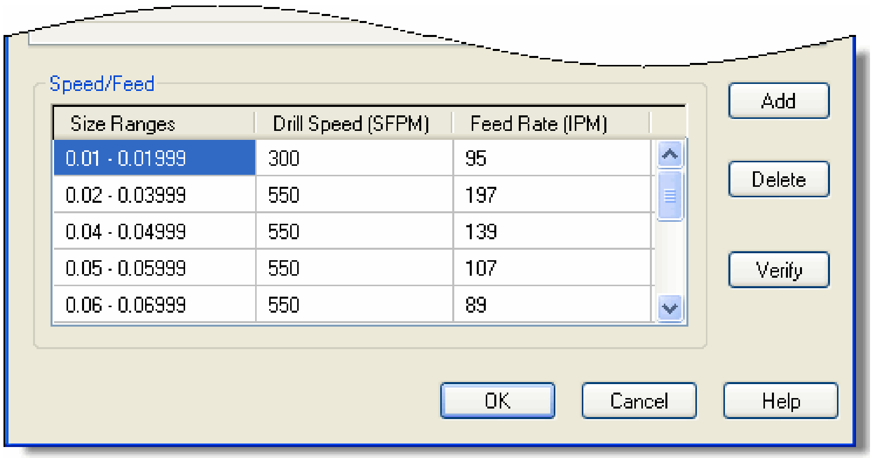


Table 45-214. NC Drill Setup Dialog Box contents

Name	Description
Output Type	Specifies the output file you want: ASCII or **EIA-244** **(**Electronic Industries Association RS-244 Standard format)
Units	Specifies the units to use: Metric or English. **Tip:** The Units list on the Excellon tab allows multiple selections with predetermined leading and trailing digits.
Type column	Displays the format string type.
String column	Lists the format string. Double click to edit. **Tips:** • Default strings on the Excellon tab are different than those on the Drill Listing tab. • See the header of the drill.dat file in the <product_name>/Settings folder for more detailed format information.
# of Digits area	Specifies the number of digits before (leading) and after (trailing) the decimal point position. This defines the output file coordinate precision. **Exception:** The # of Digits fields are unavailable on the Excellon tab since the Units list selections contain predetermined leading and trailing digits.
Zero Suppress	Specifies the zeros to suppress. • **None—**retains both leading and trailing zeros. • **Leading—**suppresses leading zeros. • **Trailing**—suppresses trailing zeros.

Table 45-214. NC Drill Setup Dialog Box contents (cont.)

Name	Description
Coordinate	Specifies the coordinate type. • **Absolute**—absolute coordinates. • **Incremental**—relative coordinates.
Speed/Feed table	Controls the drill speed and feed rates. Double-click to modify an existing entry. Edit the size range, drill speed (surface feet per minute), and feed rate (inches per minute) values in the boxes. **Tip:** Drilling feed rates can be set between 10 and 500 IPM (4 and 212 mm/s if metric) in increments of 1 IPM (1 mm/s if metric).
Add button	Adds a new row at the bottom of the table.
Delete button	Removes the selected row from the table.
Verify button	Ensures that you have values in the Size Range box which satisfy each drill on your board. A prompt tells you whether the drill sizes are accurate or if drill size ranges are missing from the Size Ranges box.

Related Topics

Creating an NC Drill File

Creating CAM Outputs to Manufacture Your PCB

Net Assignment Dialog Box

Use the Net Assignment dialog box to select nets to associate to copper being used to bridge two or more nets. This ensures that when online design rule checking is enabled, the bridge copper isn't flagged as a violation, but all other accidental physical connections between the nets are caught.

Accessing

- In the Add Drafting or Drafting Properties dialog box, click the **Bridge** check box and then click the **Nets to bridge** button.

Figure 45-234. Net Assignment Dialog Box

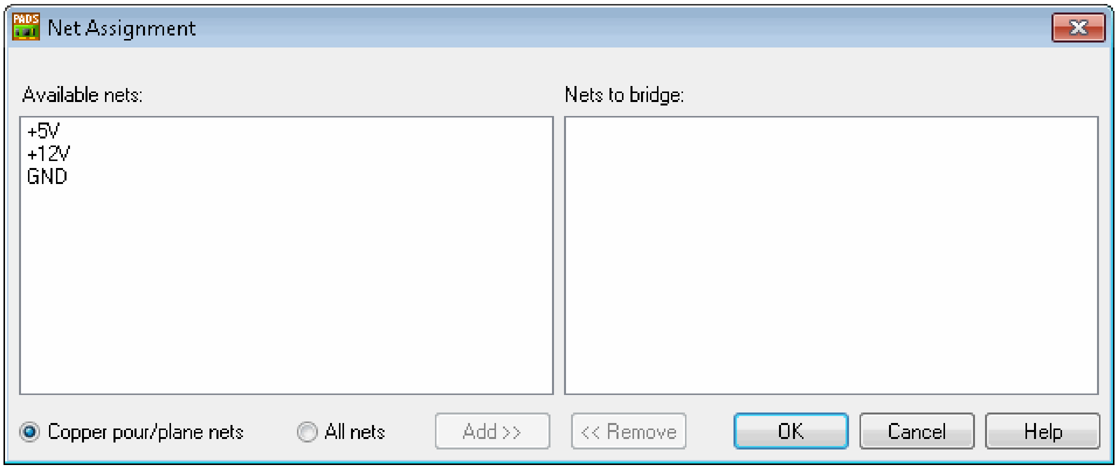


Table 45-215. Net Assignment Dialog Box Contents

Name	Description
Available nets	Specifies the nets that are available for adding to the Nets to bridge list to associate to the copper bridge. This list is filtered by selecting either *Copper pour/plane nets* or *All nets.*
Nets to bridge	Specifies the nets to associate to the copper bridge. The copper can bridge these nets with design rules enabled.
Copper pour/plane nets	Filters the Available nets list to display only copper pour and plane area nets.
All nets	Filters the Available nets list to display all the nets in the design.

Related Topics

Bridging Nets with Copper

Net Properties Dialog Box

The Net Properties dialog box displays the netname, the pin-to-pin connections in a list, routing information, and rules data. You also set route protection for nets in this dialog box.

The Net Properties dialog box remains open until you click **OK** or Cancel. If you select another net while the dialog box is open, the information updates for the selected net.

Tip: Several of the options in this dialog box are unavailable if the net is part of a physical design reuse or contains protected routes.

Accessing

- **Select a net > Right-click > Properties**

Figure 45-235. Net Properties Dialog Box

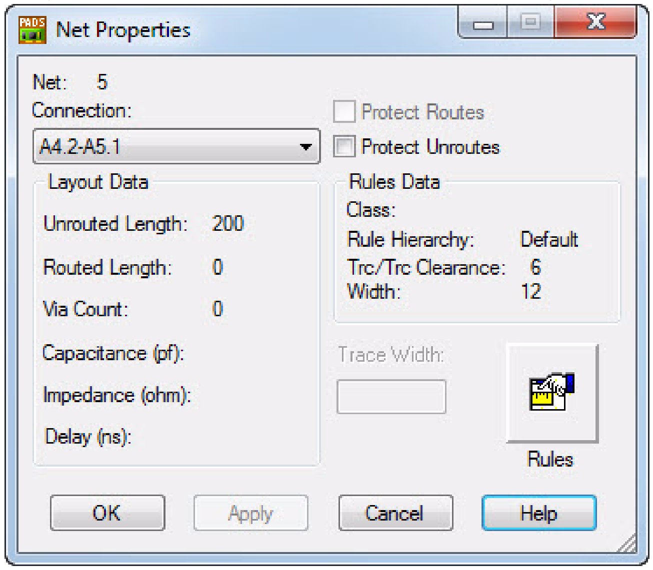


Table 45-216. Net Properties Dialog Box contents

Name	Description
Net	Displays the name of the net.
Connection	Lists the connections available in the design.
Layout Data area	Lists all layout data about the selected net.
Protect Routes	Protects selected routes or traces. **See also:** To Protect Routes
Protect Unroutes	Protects unrouted connections and the unrouted portions of partial routes. **See also:** To Protect Unroutes
Rules Data area	lists all of the rules that apply to the selected net.
Trace Width	Modifies the trace width. Type a new value in the box. This option is unavailable if route protection is on.

Table 45-216. Net Properties Dialog Box contents (cont.)

Name	Description
Rules button	Opens the Net Rules dialog box, at the proper hierarchy level, that applies to the pin pair. **Tip:** If rules are defined specifically for the pin pair, the Pin Pair Rules dialog box appears; if no rules are set for the pin pair or net, the Default Rules dialog box appears.

Related Topics

Modifying Net Properties

To Assign Colors to Nets

View Nets Dialog Box

Net Properties Dialog Box, Associated Nets

The Net Properties dialog box displays the netname, the pin-to-pin connections in a list, routing information, and rules data. You also set route protection for nets in this dialog box.

The Net Properties dialog box remains open until you click **OK** or Cancel. If you select another net while the dialog box is open, the information updates for the selected net.

Restriction: Several of the options in this dialog box are unavailable if the net is part of a physical design reuse or contains protected routes.

Accessing

- **Select a net** > **Right-click** > **Properties**

Figure 45-236. Net Properties Dialog Box, Associated Nets

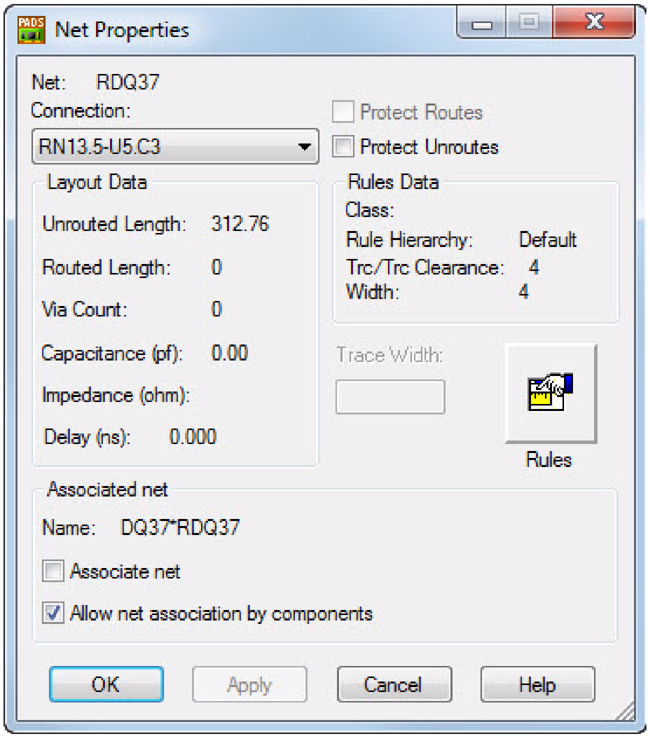


Table 45-217. Net Properties Dialog Box Contents, Associated Nets

Name	Description
Net	Displays the name of the net.
Connection	Lists the pin pair connections available in the net.

Table 45-217. Net Properties Dialog Box Contents, Associated Nets (cont.)

Name	Description
Layout Data area	Displays all layout data about the selected net. • **Unrouted Length**—Displays the length of the net that is unrouted. • **Routed Length**—Displays the length of the net that is routed. • **Via Count**—Displays the number of vias that are used in the net. • **Capacitance (pf)**—displays a baseline capacitance value of the net. This calculation uses the trace parameters, the board material values you set in the Layer Thickness Dialog Box, and requires at least one adjacent plane to display values. A maximum of two adjacent planes are used in the calculation. If the plane layer is a split/mixed plane layer, you must have a plane shape on the layer. The location and outline of the shape and cutouts of the plane area are all ignored in the calculation. You can set a Minimum and Maximum Capacitance rule in the HiSpeed Rules Dialog Box, but it is only verified when you run Tools > Verify Design after you set up a High Speed check to check Capacitance. It is not verified by online DRC. • **Impedance (ohm)**—displays a baseline impedance value of the net. This calculation uses the trace parameters, the board material values you set in the Layer Thickness Dialog Box, and requires at least one adjacent plane to display values. A maximum of two adjacent planes are used in the calculation. If the plane layer is a split/mixed plane layer, you must have a plane shape on the layer. The location and outline of the shape and cutouts of the plane area are all ignored in the calculation. You can set a Minimum and Maximum Impedance rule in the HiSpeed Rules Dialog Box, but it is only verified when you run Tools > Verify Design after you set up a High Speed check to check Impedance. It is not verified by online DRC. • **Delay (ns)**—displays a baseline delay value of the net. This calculation uses the trace parameters, the board material values you set in the Layer Thickness Dialog Box, and requires at least one adjacent plane to display values. A maximum of two adjacent planes are used in the calculation. If the plane layer is a split/mixed plane layer, you must have a plane shape on the layer. The location and outline of the shape and cutouts of the plane area are all ignored in the calculation. You can set a Minimum and Maximum Delay rule in the HiSpeed Rules Dialog Box, but it is only verified when you run Tools > Verify Design after you set up a High Speed check to check Delay. It is not verified by online DRC.
Protect Routes	Protects selected routes or traces from being moved. Similar to gluing components but you can allow protected traces to be edited by PADS Router in the Routing Rules. **See also:** To Protect Routes

Table 45-217. Net Properties Dialog Box Contents, Associated Nets (cont.)

Name	Description
Protect Unroutes	Protects unrouted connections and the unrouted portions of partial routes. If you are in a routing mode, you will not be able to select the unroute to begin routing. If you select one of the pins of the pair, you will not be able to complete routing of the pin pair. If you attempt to finish the trace on the pin pair, you will only see a partial route-completion target instead of the full route-completion target. **See also:** To Protect Unroutes

Rules Data area	Displays some rule information for the selected net.
	• **Class**—If the net is included in a Class rule, the Class name is displayed.
	• **Rule Hierarchy**—Displays one of the following from lowest priority to highest priority:
	  ◦ "Default" is shown if the net is using the Default rules.
	  ◦ "Class" is shown if the net is getting its rules from a Class rule. The class name will be shown immediately above.
	  ◦ "Net" is shown if the net has its own rules.
	• **Trc/Trc Clearance**—Displays the Trace to Trace clearance value from the Clearance Rules dialog box. See the Rule Hierarchy above to determine if the value is from the Default, Class, or Net level rules. Conditional rules are not shown.
	• **Width**—Displays the Recommended Trace Width from the Clearance Rules dialog box. See the Rule Hierarchy above to determine if the value is from the Default, Class, or Net level rules.

Trace Width	Modifies the trace width. Type a new value in the box. You are restricted to the range of Trace widths you have specified in the Clearance Rules dialog box. This option is unavailable if the Protect Routes checkbox is selected.
Rules button	Opens the Net Rules dialog box with the net preselected in order to apply Net Level rules.
Name:	If one or more nets belonging to a single associated net are selected, displays the name of the associated net to which the selected net(s) belong. If nets belonging to more than one associated net are selected, no associated net names are listed. An associated net's name is a sorted list of net names separated by asterisks (*).

Table 45-217. Net Properties Dialog Box Contents, Associated Nets (cont.)

Name	Description
Associate net	Select the checkbox to associate the selected nets. **Tip:** You can also set this checkbox (and associate nets) using the Associate Nets popup command for selected nets. See Creating Associated Nets. Clear the checkbox to disassociate any selected net that is part of an associated net. Clear both checkboxes to exclude the selected nets from inclusion in any associated net.
Allow net association by component	Select the checkbox to allow the selected nets to be included in associated nets by: • Specifying, in the Associated Nets dialog box, the refdes prefix of the nets' , or • Checking the Associate nets checkbox in the Component Properties dialog box. Clear the checkbox to disable association by component (including by refdes prefix). **Tip:** This checkbox is also cleared by the Disable Net Association popup command for selected nets. **Restriction:** Nets that have the Associate net checkbox checked are not disabled from association by clearing this check box. Clear both checkboxes to exclude the selected nets from inclusion in any associated net.

Related Topics

Modifying Net Properties

To Assign Colors to Nets

View Nets Dialog Box

Net Properties Dialog Box - Design Reuse

Use the Net Properties dialog box to resolve netname conflicts between netnames in a physical design reuse and netnames in the design. Only netnames from route and polygon elements in the physical design reuse appear in this dialog box.

Net properties are applied to route and polygon elements. For example, assigning a merge status for net GND merges all physical design reuse net and polygon objects using net GND with net GND in the design.

Tip: You can't rename and merge a net. For example, if you have GND in the physical design reuse and GND1 in the design, there is no available method to rename GND to GND1 and then merge. You must add GND and then manually merge the nets using ECO.

Accessing

- Select a reuse with nets > **Right-click** > **Properties** > **Net Properties** button

Figure 45-237. Net Properties - Design Reuse Dialog Box

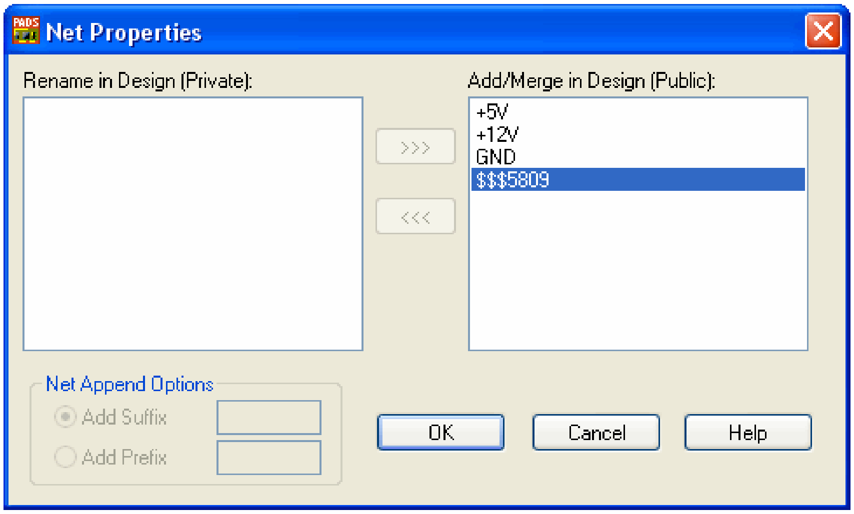


Table 45-218. Net Properties - Design Reuse Dialog Box contents

Name	Description
Rename in Design (Private)	Lists nets in the physical design reuse to rename in the design. Nets are renamed using the Rename Preferences settings. If nets exist that should remain intact, and not be merged with nets in the design, place them in this list. These nets are called private nets because they are internal to the reuse and are not connected to other nets in the design, nor do they have the same netname. If you want the net to be merged with a net in the design, move it to the Add/Merge in Design (Public) list.
Add/Merge in Design (Public)	Lists nets in the physical design reuse to either add to the design (because the net doesn't exist in the design) or to merge with a net in the design (because the net does exist in the design). These nets are called public nets because they are used in both the physical design reuse and the design. If you want the net to remain separate from the net in the design, move it to the Rename in Design list.

Table 45-218. Net Properties - Design Reuse Dialog Box contents (cont.)

Name	Description
Net Append Options	Indicates whether to rename a net using a prefix or a suffix. Affixes (prefixes and suffixes) can be up to four characters long. Illegal characters are brackets ({ }), asterisk (*), space, and period (.). You can, however, leave the box empty, and that is considered a valid entry. The value you type here is checked against the design to ensure that duplicate netnames are not created. If duplicates occur, an error message appears and you can specify a different affix. **Tip:** If you are not in ECO mode when you access this dialog box from the Reuse Properties dialog box, the Net Append Options are unavailable.
Arrow Buttons	Move selected nets between the list boxes. Click the >>> button to move selected nets in the Rename in Design (Private) list into the Add/Merge in Design (Public) list. Click the <<< button to move selected nets in the Add/Merge in Design (Public) list into the Rename in Design (Private) list. This is useful if you specifically want a net to remain separate from a net that exists in the design. When you access this dialog box from the Reuse Properties dialog box, these buttons are unavailable. This prevents you from modifying physical design reuse nets. You can however, change the suffix and prefix for netnames if you are in ECO mode.

Related Topics

Reusing Designs or Parts of Designs

Net Rules Dialog Box

Use the Net Rules dialog box to define design rules that apply to nets.

Accessing

- **Setup** menu > **Design Rules** > **Net** button

Figure 45-238. Net Rules Dialog Box

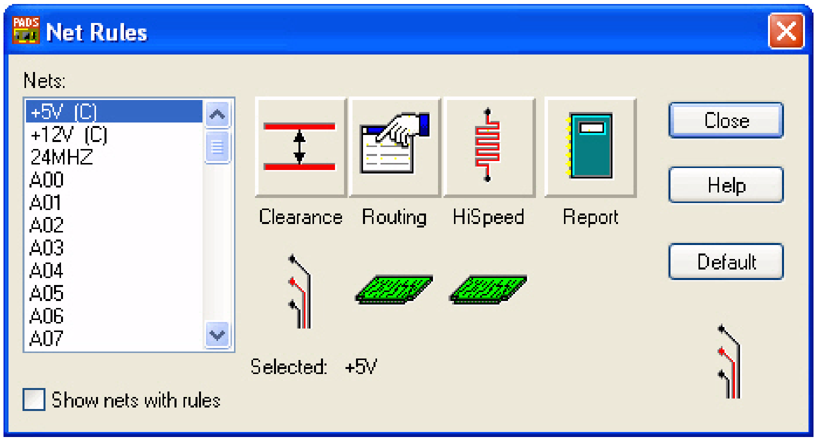


Table 45-219. Net Rules Dialog Box

Name	Description
Nets list	Lists all nets in the design.
Show Nets with rules	Specifies to show only nets that have rules.
Clearance	Opens the Clearance Rules Dialog Box.
Routing	Opens the Routing Rules Dialog Box.
HiSpeed	Opens the HiSpeed Rules Dialog Box.
Report	Opens the Rules Report Dialog Box.
Picture below rule button	The picture below each type of rule button identifies which rules hierarchy level is used for that rule type. The meaning of the picture corresponds to the button in the Hierarchy area of the Rules dialog box. For example, if you select a class in the Class list and a green polygon appears below the Clearance button, then the default values apply to the class. **See also**: Non-Default Rules Indicators
Selected	Lists the net(s) selected in the Nets list.
Default	Removes non-default rules from the selected nets, so that only default rules apply.

Related Topics

Creating Net Design Rules

Modifying Net Design Rules

Resetting Net Rules to Default Rules

Design Rule Hierarchy

Nudge Parts and Unions Dialog Box

Use the Nudge Parts and Unions dialog box to control the nudge process.

Accessing

- **Move a part or a union over another part or union. The dialog box opens.**

  Requirement: Automatic or Prompt mode must be set in the Nudge area of Tools > Options > Design page.

Figure 45-239. Nudge Parts and Union Dialog Box

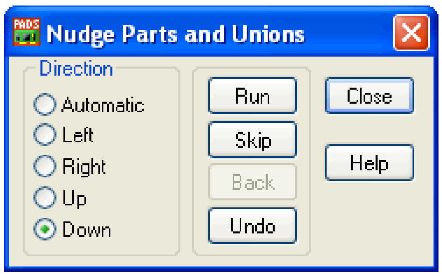


Table 45-220. Nudge Parts and Union Dialog Box

Name	Description
Direction	Indicates the direction you want to nudge the overlapping part.
Run	Performs a nudge.
Skip	Does not perform a nudge on the selected part and continues to the next overlapping part.
Back	Returns to the last skipped part.
Undo	Returns parts to their original positions, prior to nudging.

Related Topics

To Nudge Overlapping Parts

Object Attributes Dialog Box

Use the resizable Object Attributes dialog box to add, modify, or remove attributes of single objects or multiple objects of the same type.

Accessing

- Select object(s) > **right-click** > **Attribute**

Figure 45-240. Object Attributes Dialog Box

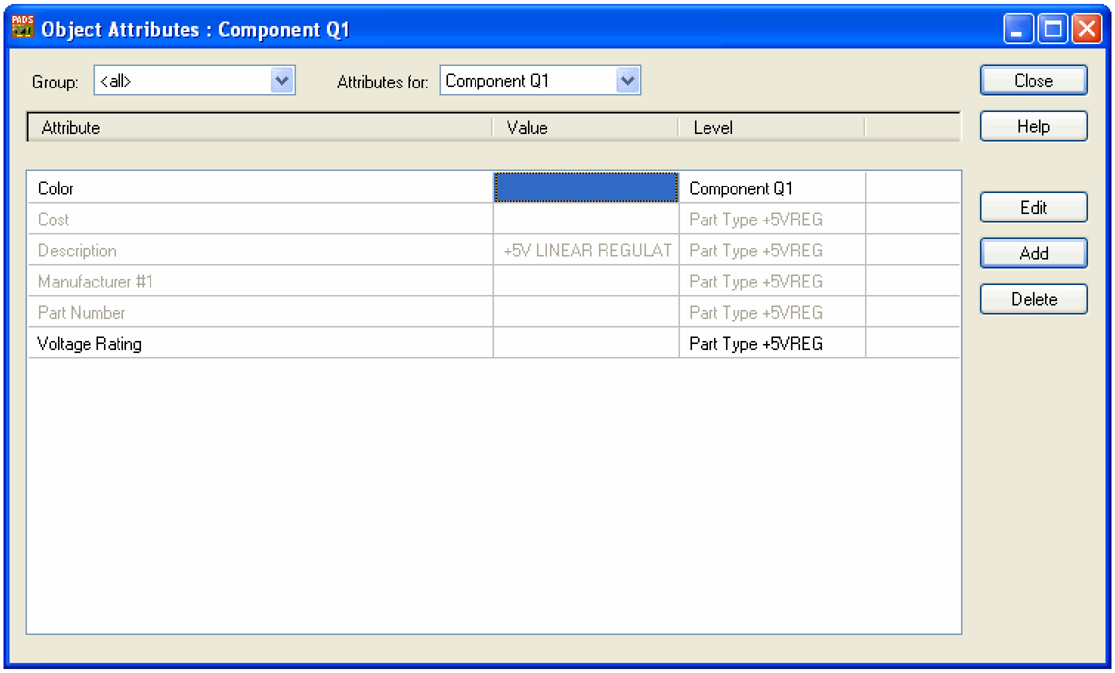


Table 45-221. Object Attributes Dialog Box

Name	Description
Group	Filters the Attributes list. You can choose an attribute group to view.
Attributes for	Specifies the attribute hierarchy level for which you want to assign an attribute. The hierarchy levels change, depending on the object you selected in step 1.
Attribute table	Lists the attributes assigned to the object.
Edit	Makes the selected cell available for editing.
Add	Adds a row to the bottom of the list where you can select a new attribute for the object.
Delete	Removes the selected attribute.

Related Topics

Working with Object Attributes

ODB++ Export Dialog Box

Use the ODB++ Export dialog box to export a design to an ODB++ file.

Accessing

- PADS Layout File menu > **Export** > **select ODB++ from Save as type: list**

Figure 45-241. ODB++ Export Dialog Box

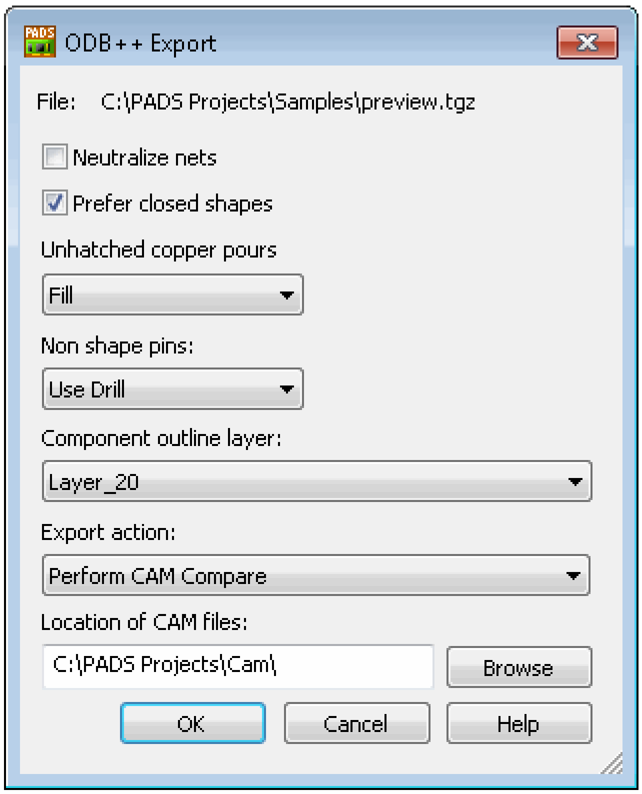


Table 45-222. ODB++ Export Dialog Box

Name	Description
Neutralize Nets	Renames nets numerically.

Table 45-222. ODB++ Export Dialog Box (cont.)

Name	Description
Prefer closed shapes	Specify how to determine component outlines: • Select to search for a closed shape to use as the component outline, beginning with the layer specified in the Component Outline Layer setting. If none is found on the Component Outline Layer, it then proceeds to layer 20, then layer 0 (all layers), then layer 1 (mounted side). If no closed shape is found, it creates a bounding box of all shapes found. • Clear this check box to just create a bounding box of all shapes that are found.
Unhatched copper pours	Specify what to do with unhatched copper pours and plane areas: • **Fill—**Flood all copper pours and plane areas before translating. • **Discard—**Discard unhatched copper pours and plane areas.
Non Shape Pins	Specify what to do with pins for which no outline is supplied or when you do not want to use the supplied pin shape. • **Discard**—Do not translate these pins. • **Add empty**—Translate these pins as-is (with no outline information). • **Use Drill**—Use the drill shape as the pin shape. • **Use Stack—**Use the pad stack shape as the pin shape. Specify what to do with pins regardless of whether an outline is supplied: • **Force Drill**—Use the drill shape instead of the pin shape. • **Force Stack—**Use the padstack shape instead of the pin shape. • **Force Piece**—If the part definition contains pin outlines supplied as “pieces”, use these instead of the pin shape.
Component Outline Layer	Specify the preferred component outline layer.
Export action	Choose an export action: • **Export only—**creates the .tgz file that you specified. • **View generated ODB++**—creates the .tgz file that you specified and loads your ODB++ data into the CAM Compare software to view the data. • **Perform CAM Compare**—creates the .tgz file that you specified and loads both the ODB++ data and your CAM documents into the CAM Compare software to allow you to compare them. **Tip:** When the CAM Compare software is active, press F1 to open the CAM Compare help documentation.

Table 45-222. ODB++ Export Dialog Box (cont.)

Name	Description
Location of CAM file	Specify the location of the CAM documents to use when comparing with the ODB++ data. **Restriction:** This is only available when you select Perform CAM Compare as the Export action.

Related Topics

Exporting ODB++ Files

Options Dialog Box, Design Page

Use the Design page to specify place and route options.

Accessing

- **Tools** menu > **Options** > **Design**

Figure 45-242. Design Page

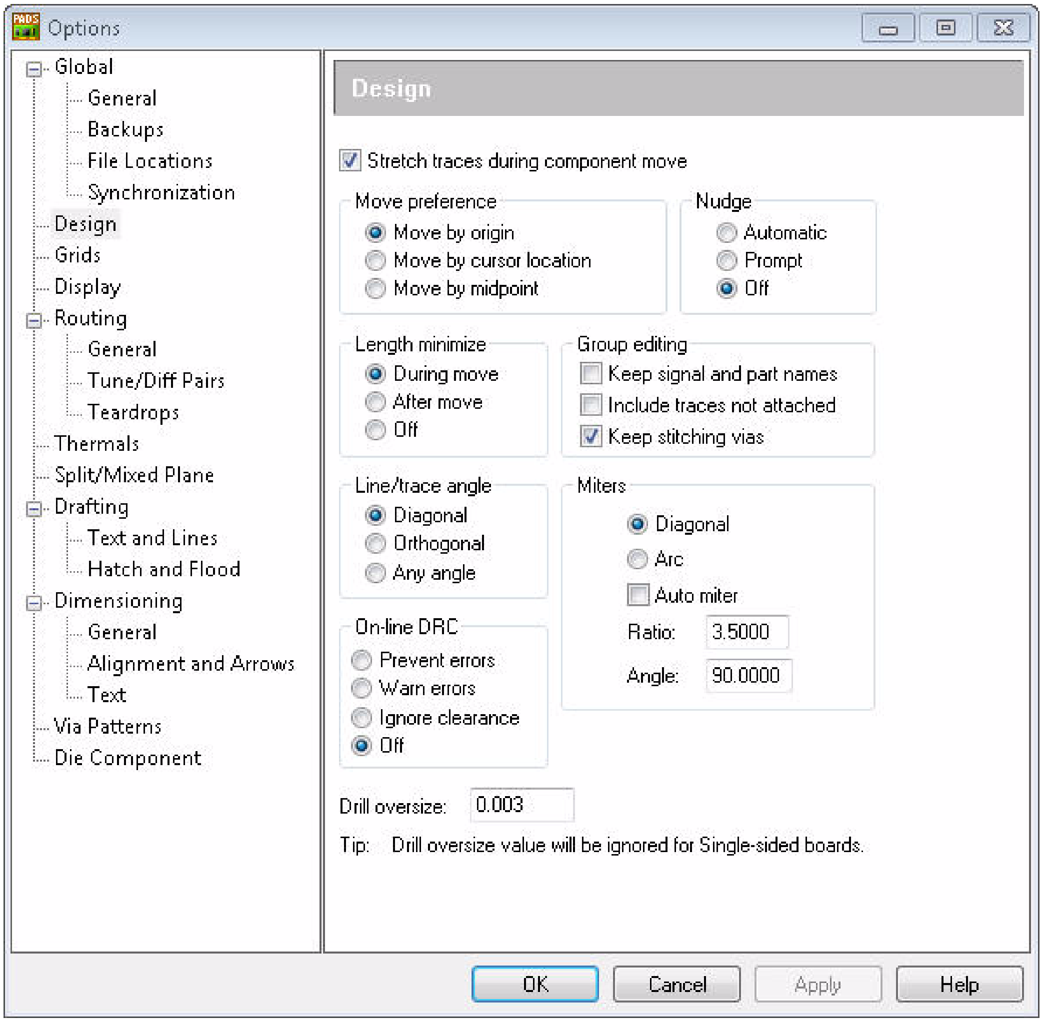


Table 45-223. Design Page Contents

Name	Description
Stretch traces during component move	Specifies whether to reroute traces, or create an unroute, for nets connected to the moved, or pin-swapped part. • Select the check box to create new routes between tacks at the old pin positions and the new pin positions after you complete the move. • Clear the check box to create an unroute connection after swapping pins or moving a part. **See also:** With Stretch Traces During Component Move in the *Concepts Guide*

Table 45-223. Design Page Contents (cont.)

Name	Description
Move preference area	Specifies which, of various locations on the part being moved, should attach to the pointer: • **Move by origin**—Pointer attaches to the part origin. • **Move by cursor location**—Pointer attaches to the part from its current location. **Example:** If you start the Move operation with the pointer at X=200,Y=500 and the selected part at X=0,Y=0, move the pointer to X=1200,Y=1500, the part moves to X=1000,Y=1000. • **Move by midpoint**—Pointer attaches to the center of a rectangle enclosing the part. **Tip:** While you are moving the part, you can modify the pointer attachment location from the following shortcut menus: • Move in the Layout Editor • Move Sequential in the Layout Editor • Move Terminal in the PCB Decal Editor • Radial Move in the Layout Editor and the PCB Decal Editor
Nudge area	Specifies how to use nudge to prevent parts from overlapping when you finish moving the part or using the Tools > Nudge Component command. • **Automatic**—Nudge automatically moves apart overlapping parts when you finish moving the part. • **Prompt**—Opens the Nudge Parts and Unions dialog box for each overlapping part when you finish moving the parts, enabling you to control the nudge operation. • **Nudge Off**—Disables nudge.
Length minimize area	Specifies when to recalculate, or minimize, unrouted net lengths when moving parts. • **During Move**—Recalculate length of unrouted pin pairs as you move a part. The closest viable connections are displayed in progress. • **After Move**—Recalculate length of unrouted pin pairs when you finish a move. This option consumes less display memory. • **Off**—Do not recalculate length of unrouted pin pairs. **Tip:** To specify the topology type (which determines length recalculation behavior) per net, per class, and so on, use the Routing Rules dialog box.

Table 45-223. Design Page Contents (cont.)

Name	Description
Group editing area	Specifies selection and editing options for multiple object operations. • **Keep signal and part names**—Maintains signal and part names when you insert data using Edit/Paste. • **Include traces not attached**—Selects all traces passing through the selection rectangle, even if they do not connect to any parts within the selection rectangle. • **Keep stitching vias**—Prevents the deletion of stitching vias, or free vias. **Tip:** This option distinguishes between stitching vias and regular routing vias. When this check box is selected, stitching vias are preserved during the following operations: • ECO delete connection, delete component, swapping a pin, swapping all pins, change component • Unroute For operations related to ECO, this option is respected whether they are performed interactively or by an ECO import operation. In general, this option is respected by any ECO operation that deletes traces and vias. When the this check box is cleared, stitching vias may be deleted during the operations listed above.
Line/trace angle area	Specifies angle options when adding or moving a line or trace (corner, or pad entry/exit). • **Diagonal**—Angle must be 45 degrees or a multiple of 45 degrees. • **Orthogonal**—Angle must be 90 degrees or a multiple of 90 degrees. • **Any Angle**—No angle restriction.

Table 45-223. Design Page Contents (cont.)

Name	Description

Miters area	• **Diagonal Miter**—Create miters as lines at 45-degree angles.
	• **Arc**—Create miters as arcs.
	• **Auto miter check box**—Creates miters automatically while adding drafting objects.
	• **Ratio**—Specifies the size of a diagonal miter or the radius of an arc miter.
	Tips:
	  • For diagonal miters, the distance from the virtual corner to either of the breakpoints is created by the Ratio multiplied by the line width. For example, a 10-mil trace and a ratio of 5 produces a 45-degree angle 50 mils apart from the virtual corner. For angles less than 90 degrees, the distance becomes longer. For angles greater than 90 degrees, the distance becomes shorter.
	  • For arc miters, the radius is set to the ratio multiplied by the trace width. For example, a ratio of one sets the arc radius equal to the trace width. Also, a 10-mil trace and a ratio of 5 produces a radius of 50 mils.
	• **Angle**—Specifies the maximum corner at which miters are created.

On-line DRC area	Specifies the response to design rule errors. • **Prevent Errors**—Prevents you from violating design rules. • **Warn Errors**—Prompts you with a warning when you try to violate the design rules, but does not prevent you from placing the part. • **Ignore Clearance**—Ignores clearance design rules. **Tip:** Allows parts to touch but not overlap each other. • **Off**—Disables design rule checking. You violate design rules. **Tip:** If you use the Layout Editor without Design Rule Checking (DRC) enabled, use Verify the Design to check the design in progress for clearance violations. **See also:** Design Rule Checking in the *Concepts Guide*

Table 45-223. Design Page Contents (cont.)

Name	Description
Drill oversize	Type a value to globally apply an oversize to plated holes for design rule checking. The oversize accounts for drill oversizing during the PCB fabrication process. **Restrictions:** • This value only applies to design rule checking and does not apply to the Drill Drawing table, or NC Drill outputs. • Your PCB fabricator normally oversizes plated holes to match the drill size that you specify in the pad stacks. • This value is ignored if Single-sided board is selected on the Layers Setup dialog box. **Tip:** The oversize is measured from the center of the pad, not the perimeter. For example, a 3 mil oversize represents 1.5 mils in all directions from the center of the pad. For more information, see the Drill Size and Plated settings in the Pad Stacks Properties Dialog Box.

Options Dialog Box, Die Component Page

Use the Die Component page to specify die part creation and modifying options.

Restriction: This information applies to only the BGA toolkit.

Accessing

- **Tools** menu > **Options** > **Die Component**

Figure 45-243. Die Component Page

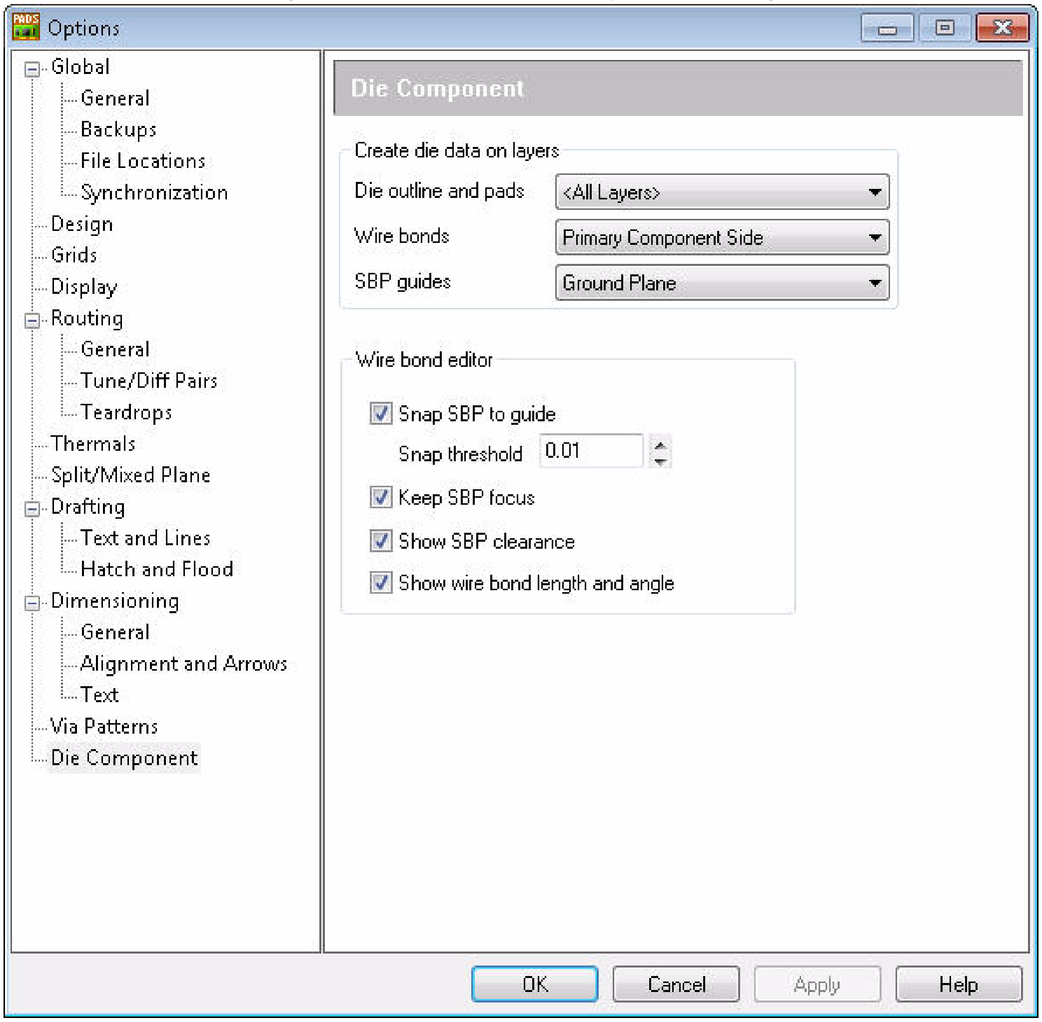


Table 45-224. Die Component Page Contents

Name	Description
Create die data on layers area	Specifies on which layers to create new die decal items, which are represented as decal drawings. Choose the layer for the die outline and pads, the wire bonds, and the substrate bond pad guides.
Wire bond editor area	
Snap SBP to Guide	Enables snap-to-guide mode when moving substrate bond pads.
Snap threshold	Specifies the distance in current design units at which a moved substrate bond pad snaps to the nearest substrate bond pad guide.

Table 45-224. Die Component Page Contents (cont.)

Name	Description
Keep SBP focus	Automatically rotates the substrate bond pad to match the direction of the wire bond.
Show SBP clearance	Displays the clearance outline when you move, add, or spin substrate bond pads, or when you add fanouts.
Show wire bond length angle	Displays the wire bond length and angle when you move, add, or spin substrate bond pads, or when you add fanouts or wire bonds.

Related Topics

To Move a Die Component Substrate Bond Pad

To Add Substrate Bond Pads

To Spin a Substrate Bond Pad

To Add a Fanout

To Add Wire Bonds

Options Dialog Box, Dimensioning /Alignment and Arrows Page

Use the Dimensioning / Alignment and Arrows page to specify the appearance of dimensioning alignment tools and arrows. Options on this page affect the appearance of text, lines, and so on, used to show clearances.

Accessing

- **Tools** menu > **Options** > **Dimensioning** / **Alignment and Arrows**

Figure 45-244. Dimensioning / Alignment and Arrows page

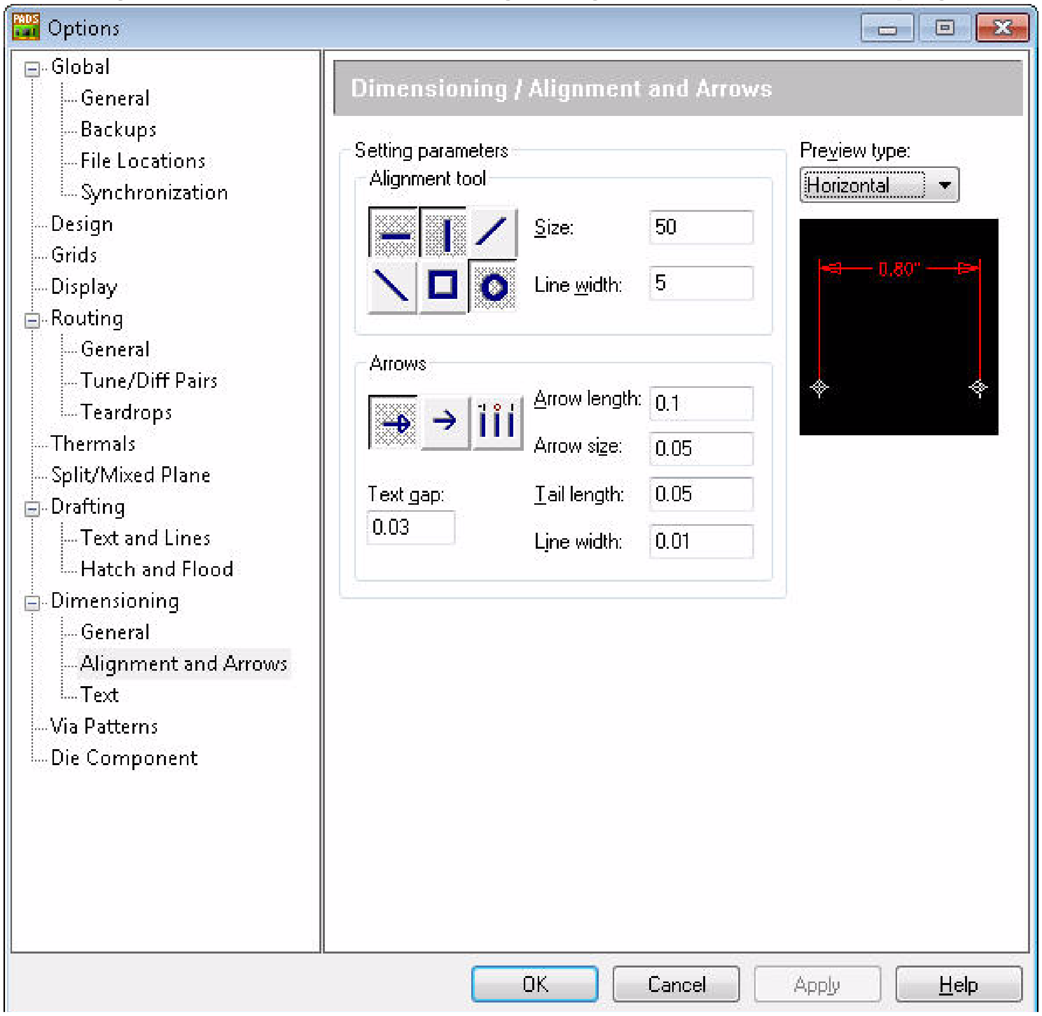
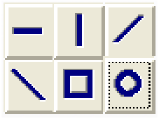
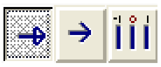


Table 45-225. Alignment and Arrow page Contents

Name	Description
Preview type	Specifies the way the image is shown in the Preview area. You have the choice of: • Horizontal • Vertical • Aligned • Angular • Circular
Alignment tool area	

Table 45-225. Alignment and Arrow page Contents (cont.)

Name	Description
	Specifies the shape of the dimensioning alignment tool appears on the screen. Choose any combination of these shapes.
Size	Specifies the alignment tool shape length or diameter.
Line Width	Specifies the alignment tool line width.
Arrows area	
	Specifies how dimensioning arrows appear on the screen. **Tip:** The right-most shape represents datum lines. With datum lines, no arrows are drawn and the dimensions are placed above the extension line. Datum lines are created when using Baseline Dimensioning.
Arrow Length	Specifies the distance between the arrow tip and the end of the arrowhead. **Restriction:** Unavailable for Datum Lines.
Arrow Size	Specifies the width of the arrowhead. **Restriction:** Unavailable for Datum Lines.
Tail Length	Specifies the minimum length of the arrow tail. **Restriction:** Unavailable for Datum Lines.
Line Width	Specifies the width of the tail and arrow lines. **Restriction:** Unavailable for Datum Lines.
Text Gap	Specifies the distance between the tail, which is the line extending behind the arrowhead, and the measurement text. **Restriction:** Unavailable for Datum Lines.

Options Dialog Box, Dimensioning / General Page

Use the Dimensioning / General page to specify the layers used to display dimensioning text and lines, the appearance of extension lines, and how to measure circles. Options on this page affect the appearance of text, lines, and so on, used to show clearances.

Accessing

- **Tools** menu > **Options** > **Dimensioning** / **General**

Figure 45-245. Dimensioning / General page

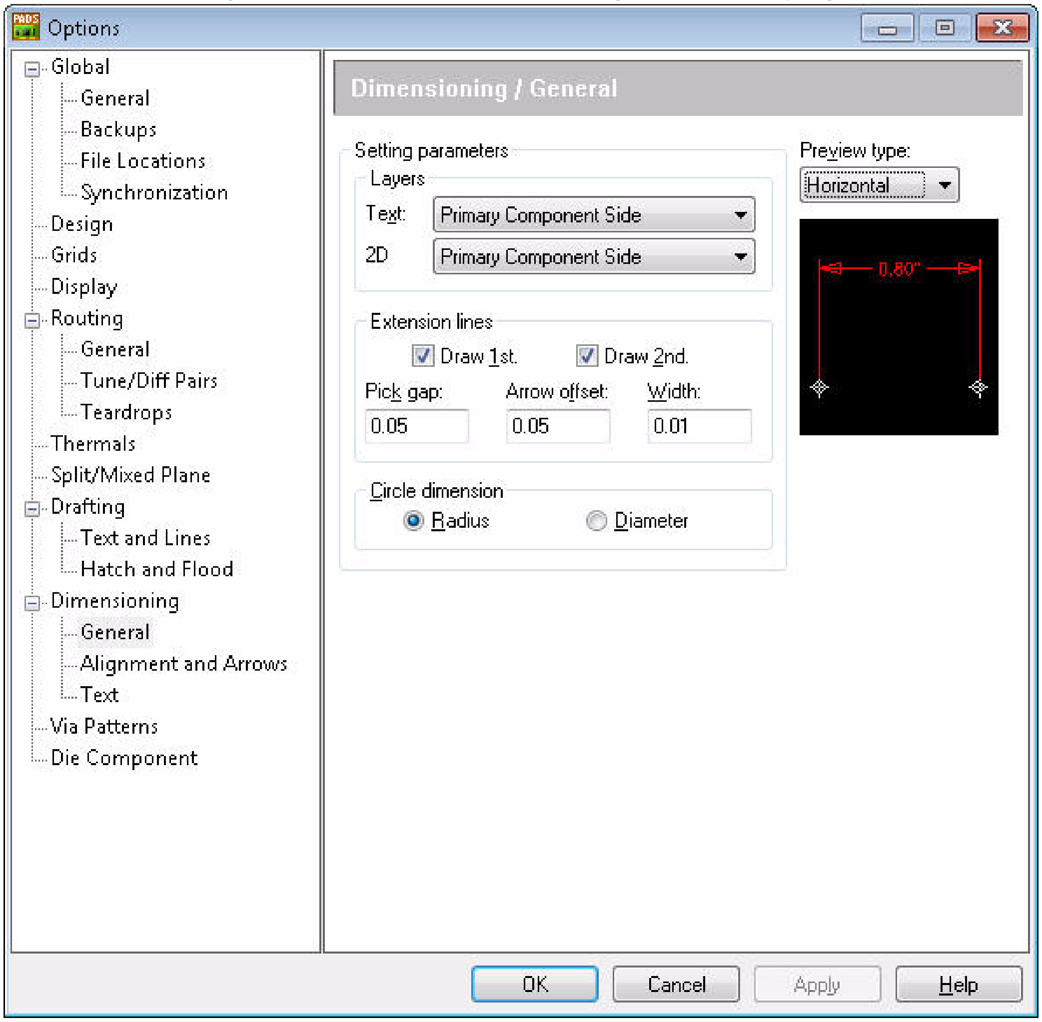


Table 45-226. Dimensioning / General page contents

Name	Description
Preview type	Specifies the way the image is shown in the Preview area. You have the choice of: • Horizontal • Vertical • Aligned • Angular • Circular

Table 45-226. Dimensioning / General page contents (cont.)

Name	Description
Layers area	Specifies the layers to use for dimensioning text and lines. **Tips:** • Dimensioning items ignore the current layer set in the Layer list on the main toolbar, so you must specify layers here. • Use the Dimension Properties dialog box to reset the layer for a selected dimension object or any of its dimension elements. • Items placed on layer 0 appear on all layers.
Extension lines area	
Draw 1st	Specifies that you want to display an extension line for the first point you select.
Draw 2nd	Specifies that you want to display an extension line for the second point you select.
Pick gap	Specifies the distance between the selection point and the tip of the extension line. **Tip:** All numerical values are in current design units.
Arrow offset	Specifies the overhang of the extension line beyond the arrowhead. **Tip:** All numerical values are in current design units.
Width	Specifies the width of the extension line. **Tip:** All numerical values are in current design units.
Circle dimension area	Specifies the circle measurement method you want.

Options Dialog Box, Dimensioning / Text Page

Use the Dimensioning / Text page to specify the appearance of dimensioning text used to show clearances.

Accessing

- **Tools** menu > **Options** > **Dimensioning** / **Text**

Figure 45-246. Dimensioning / Text page

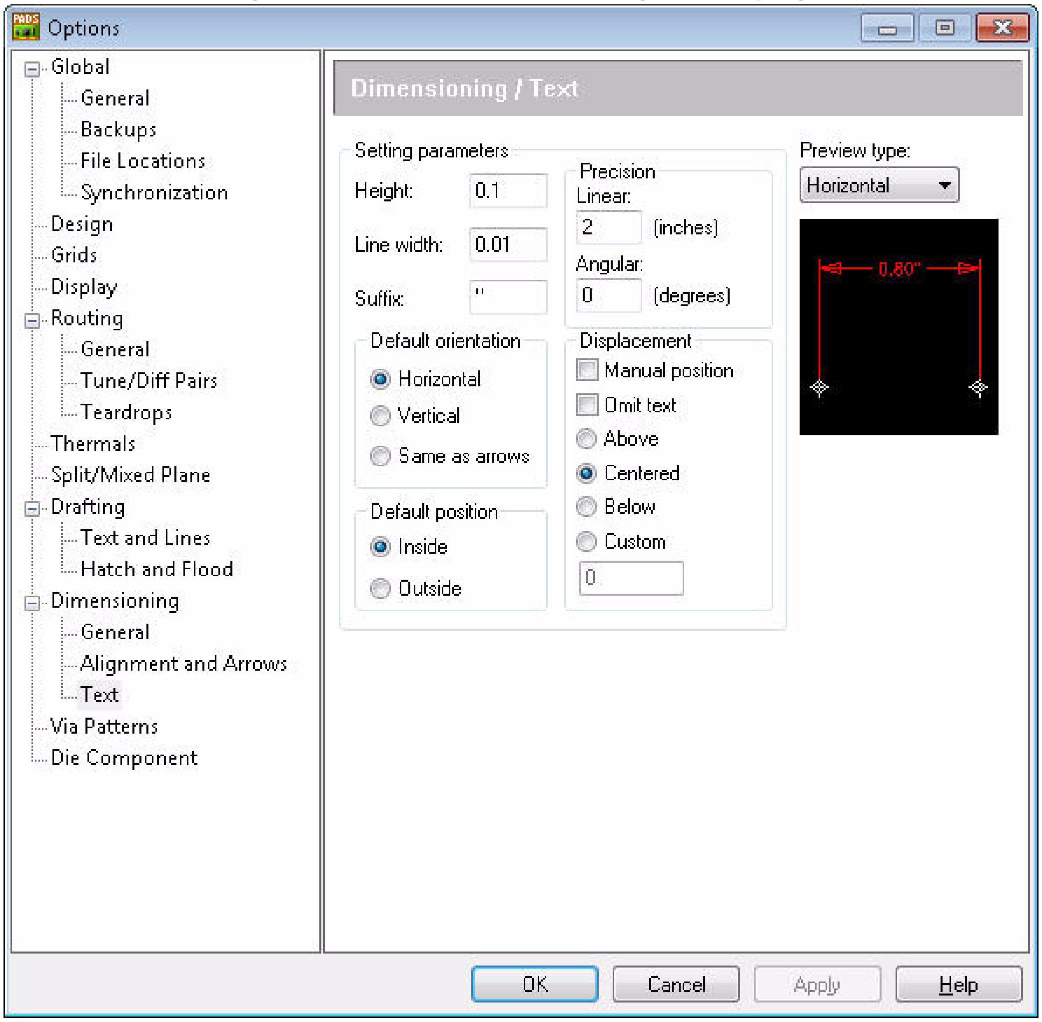


Table 45-227. Dimensioning / Text page contents

Name	Description
Preview type	Specifies the way the image is shown in the Preview area. You have the following choices: • Horizontal • Vertical • Aligned • Angular • Circular
Setting parameters area	
Height	Specifies the height of text characters.
Line Width	Specifies the width of one character.

Table 45-227. Dimensioning / Text page contents (cont.)

Name	Description
Suffix	Specifies the suffix characters that follow the dimensioning measurement.
Precision area	
Linear	Specifies linear measurement precision: type the number of decimal places, in current design units.
Angular	Specifies angular measurement precision: type the number of decimal places, in degrees.
Default orientation area	Specifies how you want to orient the dimensioning text relative to the screen or to the dimensioning arrows. • **Horizontal**—Orient text horizontally on the screen, regardless of the arrow angle. • **Vertical**—Orient text vertically on the screen, regardless of the arrow angle. • **Same As Arrows**—Orient text parallel to the arrows.
Default position area	Specifies to position text outside the extension lines, or inside the extension lines if possible.
Displacement area	
Manual Position	Specifies to position text manually, by attaching the text to the pointer when you add the dimension.
Omit Text	Specifies to create dimensioning lines and arrows without displaying dimensioning text.
Displacement	Specifies to position text relative to the arrow axis. You have the following choices: • **Above**—Position text above the arrow axis. • **Centered**—Position text on the arrow axis. • **Below**—Position text below the arrow axis. • **Custom**—Position text at the position you specify. **Tip:** Zero centers the text, positive numbers place the text above the arrow, and negative numbers place the text below the arrow.

Options Dialog Box, Display Page

Use the Display page to set the text size of displayed net names and pin numbers, and the maximum allowable gap between net names on traces.

Accessing

- **Tools** menu > **Options** > **Display**

Figure 45-247. Display page

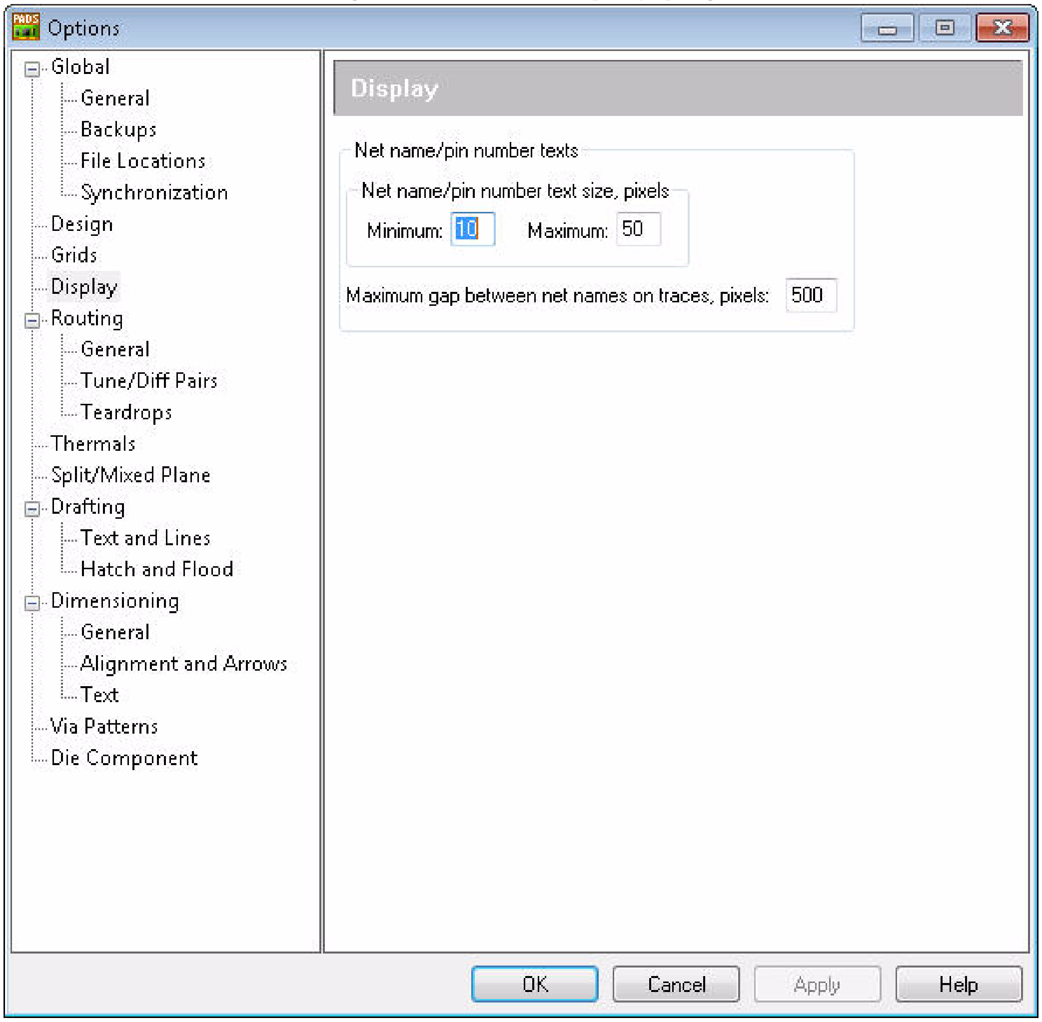


Table 45-228. Display Page Contents

Name	Description
Minimum	Specifies the minimum size of displayed net names and pin numbers, in pixels.
Maximum	Specifies the maximum size of displayed net names and pin numbers, in pixels.
Maximum gap between net names on traces, pixels	Specifies the maximum allowable gap between displayed net names in traces, in pixels.

Tip: The display of net names is controlled by the Net Name column check box, by the color tiles on each layer, and the *Show net names on Traces, Vias, Pins* check boxes in the Display Colors Setup Dialog Box.

Options Dialog Box, Drafting / Hatch and Flood Page

Use the Drafting / Hatch and Flood page to view and edit options for copper shapes.

Accessing

- **Tools** menu > **Options** > **Drafting** / **Hatch and Flood**

Figure 45-248. Drafting / Hatch and Flood page

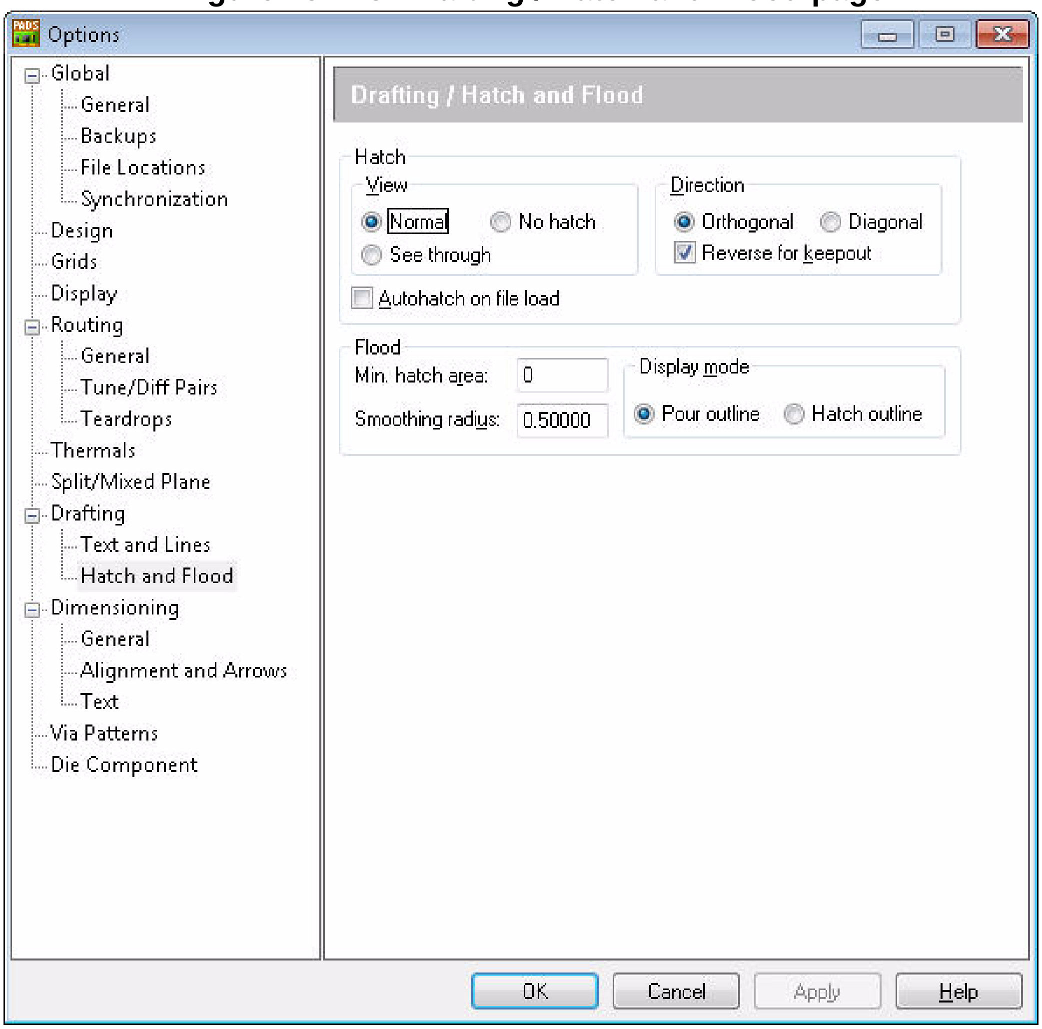


Table 45-229. Drafting / Hatch and Flood page contents

Name	Description
Hatch View area	Specifies the display of the hatching. Choose from: • **Normal**—Display hatching • **No Hatch**—Remove hatching • **See Through**—Display hatching with non-intersecting lines
Hatch Direction area	Specifies the hatch orientation in the workspace. Choose from: • **Orthogonal**—Hatching consists of diagonal lines • **Diagonal**—Hatching consists of vertical and horizontal lines Reverse for keepout check box—Distinguishes keepout hatching from all other hatching by reversing the orientation. **Example:** If this option is enabled and the hatch orientation is Orthogonal, hatching for keepout areas is diagonal.
Autohatch on file load	Specifies to hatch the copper pours and plane areas when you load the file.
Flood area	
Min. hatch area	Specifies the smallest island area created by flooding in squared current design units. **Example:** If you do not want to display islands smaller than four square design units, type 4 into the box.
Smoothing radius	Specifies the radius of copper pour corners in current design units
Display mode area	• **Hatch Outline**—Display the copper pour outline and hatching • **Pour Outline**—Display the copper pour outline with no hatching **Tip:** If you attempt to flood a copper pour area with a Hatch outline, you are warned, “To generate new thermals turn off hatch display prior to flooding.” You must change the outline to a Pour outline before flooding.

Options Dialog Box, Drafting / Text and Lines Page

Use the Drafting / Text and Lines page to view and edit options for drafting objects.

Accessing

- **Tools** menu > **Options** > **Drafting** / **Text and Lines**

Figure 45-249. Drafting / Text and Lines page

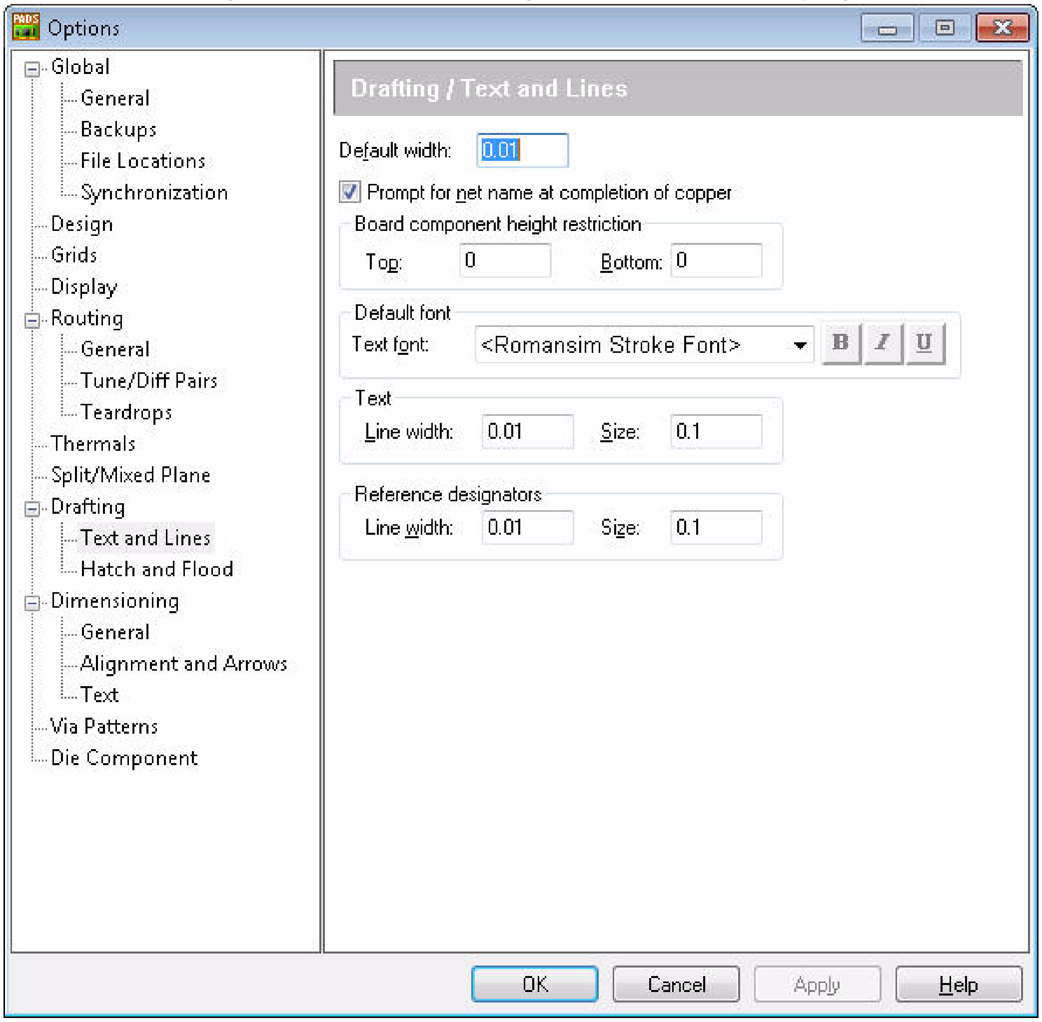


Table 45-230. Drafting / Text and Lines page contents

Name	Description
Default width	Specifies the default width to use when adding drafting objects. **Tips:** • To change the width of an existing shape, select the shape, right-click, and click Properties. • To search for and change all shapes of a similar width, on the Edit menu, click Find.

Table 45-230. Drafting / Text and Lines page contents (cont.)

Name	Description
Prompt for net name at completion of copper	Specifies whether you are prompted to assign a net to the new copper. When you complete drafting a copper shape, the Add Drafting dialog box opens automatically. Assign the net to the copper by using the Net assignment list to select the net from a list, or by using the Assign Net by Click button to click on an object in the design with the net name you want to assign.
Board component height restriction area	Specifies the height restriction in current design units for the top and bottom side layers. Sets the maximum height for all design components that use the Geometry.Height attribute. **See also:** Restricting Heights on Component Layers. **Tips:** • You can set height restrictions for individual components by creating a Component keepout with a height restriction. **See also:** Restricting Heights in Areas of Component Layers. • Use On-line DRC to prevent placement of components that exceed height restrictions. Use Verify the Design to check for violations after placement. • **See also:** Attribute Dictionary in the *Concepts Guide*
Default font area	Specifies the default font and style for all newly created text strings and labels. **Select <PADS Stroke Font>** to use the default stroke font installed on your system or choose a font name from the list of fonts installed on your system. **Restriction**: Type 1 fonts are not supported. **Tips:** • The font in use is highlighted at the top of the list. • The fonts listed above the horizontal line are used in the design. • If you select a system font, you can also click one or more buttons to specify a font style: **B** for bold, **I** for italic, or **U** for underlined.

Table 45-230. Drafting / Text and Lines page contents (cont.)

Name	Description
Text area	**Specify text options for design drafting text.** • **Line width**—Specifies the text line width in current design units.  • **Size**—Specifies the text height in current design units. 

Table 45-230. Drafting / Text and Lines page contents (cont.)

Name	Description
Reference designators	**Specifies the default width and height for reference designator labels, pin numbers, and pin names.** The PCB Decal Editor Add New Decal Label dialog box and the Layout Editor Add New Part Label dialog box use the default width value set on this tab. • **Line width**—Specifies the reference designator line width in current design units. **Restriction:** Reference designator width is stored in the library, but the pin number is not.  • **Size**—Specifies the reference designator height in current design units.  **Tip:** If you modify the default width and height, the physical appearance of existing pin numbers updates to the new values, but the physical appearance of existing labels does not change. To change the label, select the label, right-click, and click Properties.

Options Dialog Box, Global / Backups Page

Use the Global / Backups page to specify design data backup options.

Accessing

- **Tools** menu > **Options** > **Global / Backups**

Figure 45-250. Global / Backups page

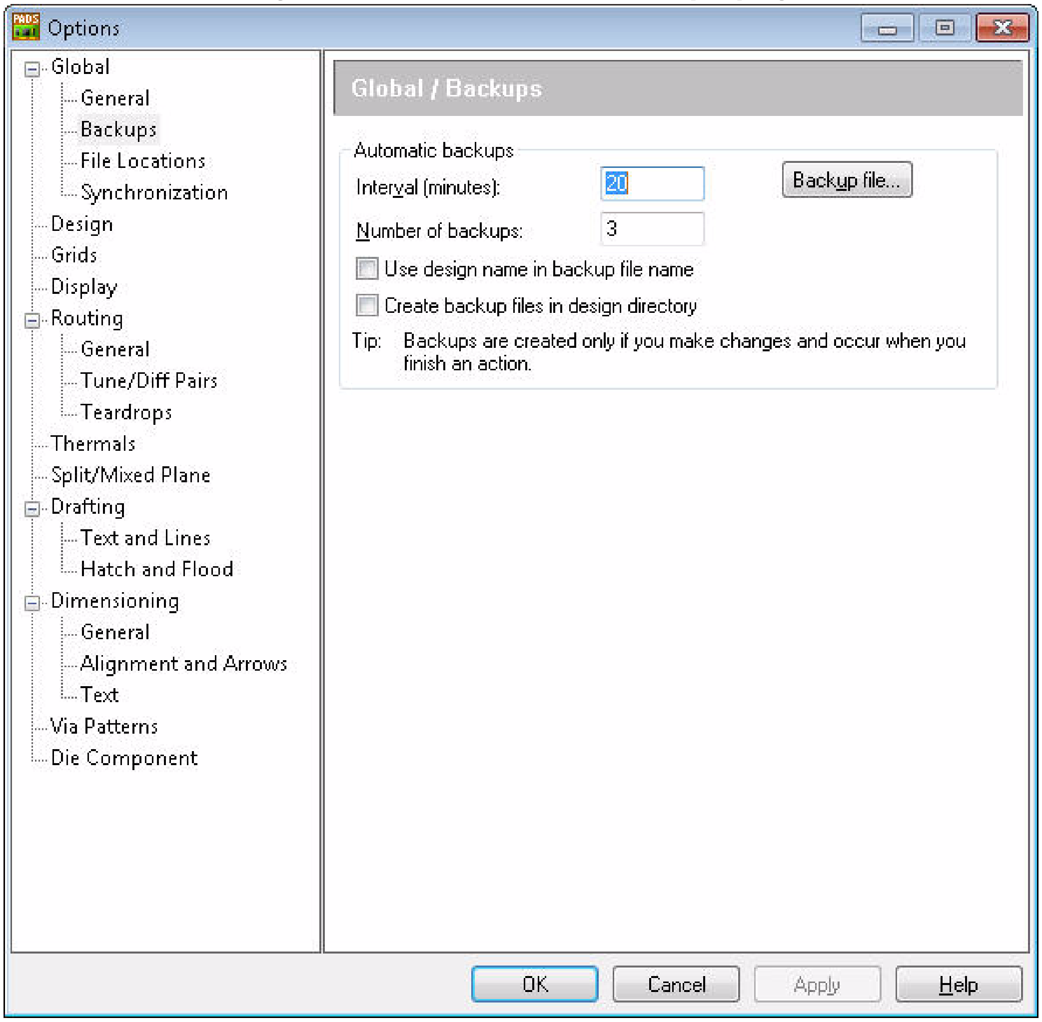


Table 45-231. Global / Backups page Contents

Name	Description
Automatic backups area	
Interval (minutes)	Specifies the time in minutes between backups.
Number of backups	Specifies the quantity (1-9) of different backup files to create. **Tip:** Backup files are named <design_name>.#, where # is a sequential number. For example, Layout1.pcb, Layout2.pcb, and so on.
Backup file	Opens the Backup File dialog box where you can change to folder or name of the backup file.

Table 45-231. Global / Backups page Contents (cont.)

Name	Description
Use design name in backup file name	Specifies to use the design name in your backup file name. Click to clear if you want to use Layout as the file name.
Create backup files in design directory	Specifies to **place all of your backup files in the same directory as the design**. Click to clear if you want your backup files in one, common backup directory.

Options Dialog Box, Global / File Locations Page

Use the Global / File Locations page to specify default design location options.

Accessing

- **Tools** menu > **Options** > **Global / File Locations**

Figure 45-251. Global / File Locations page

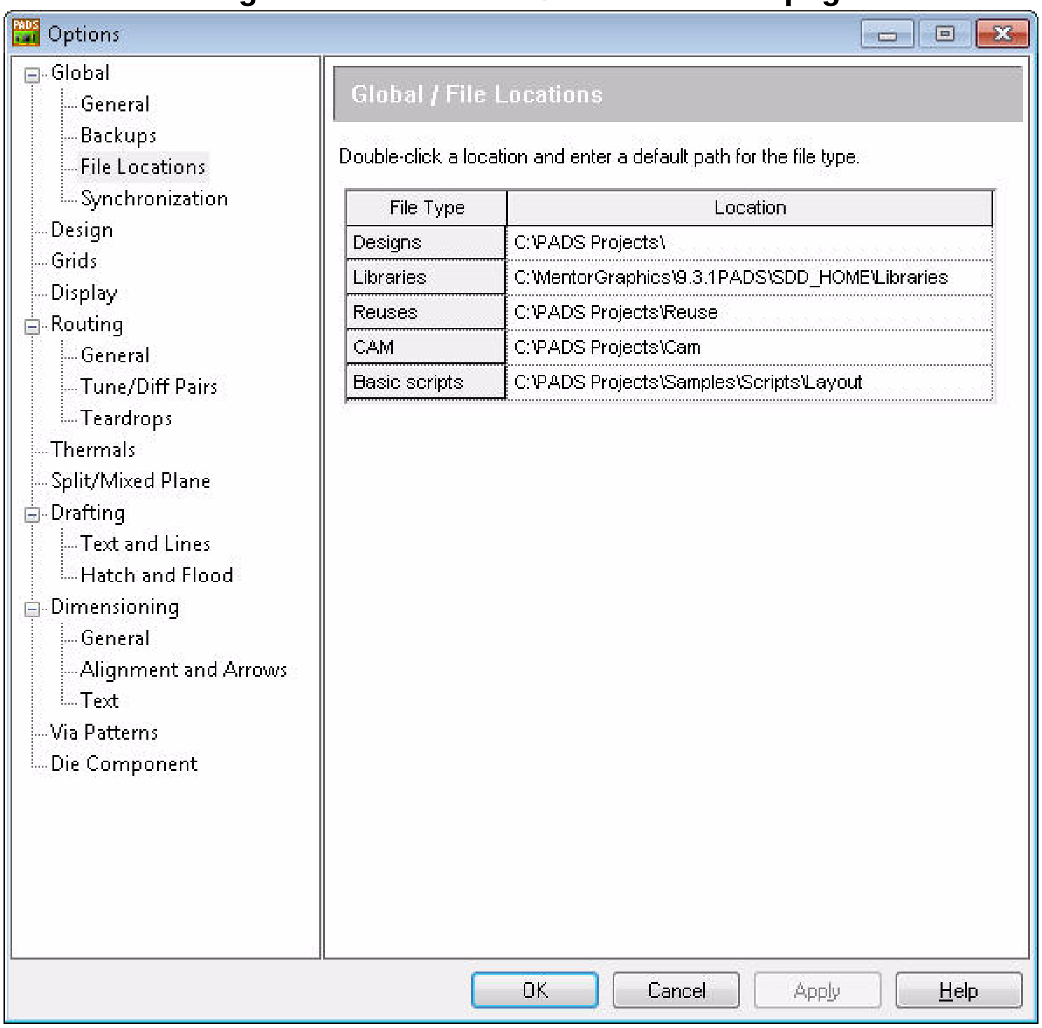


Table 45-232. Global / File Locations page contents

Name	Description
File Type	Specifies the type of file for which to set the location.
Location	Specifies the location of the corresponding file type.

Options Dialog Box, Global / General Page

Use the Global / General page to specify various workspace and design unit options.

Accessing

- **Tools** menu > **Options** > **Global / General**

Figure 45-252. Global / General Page

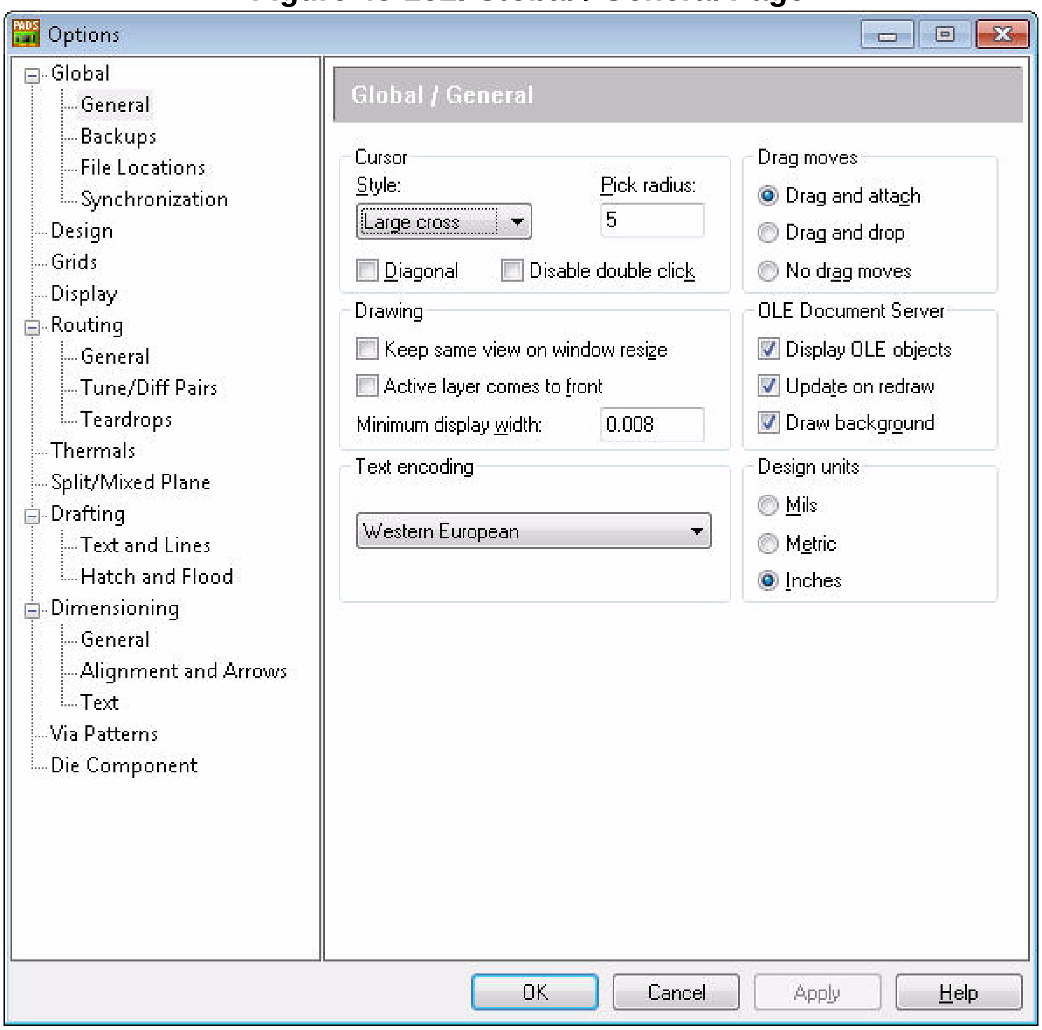


Table 45-233. Global / General Page Contents

Name	Description
Cursor area	
Style	Specifies the pointer shape. Choose from: • **Normal**—Arrow • **Small cross**—Small plus sign + • **Large cross**—Large plus sign + • **Full screen**—Full screen crosshair
Pick radius	Specifies the selection accuracy - the maximum distance in pixels that the pointer can be from an object and still select it. **Tip:** Larger values mean the pointer can select objects further away, but this can cause you to select incorrect objects.
Diagonal	Specifies to rotate the pointer shape diagonally, so that it resembles an x instead of a plus sign +. **Tip:** This option is unavailable for the Normal pointer shape.
Disable double click	Specifies to disable double-click operations, such as adding vias, opening the Properties of objects, or completing a drafting object polygon.
Drag moves area	Specifies the behavior when you try to drag objects with the pointer. • **Drag and Attach**—Attaches the object to the pointer when you click it and begin to drag it. While the object is attached to the pointer, release the left button and move the object to the new location. Click to finish the move. After placement the object remains selected. • **Drag and Drop**—Same as Drag and Attach, but the move is finished when you release the left button. • **No Drag Moves**—Prohibits drag type moves. **Tips:** • Because it is possible to accidentally start a move sequence when performing an area select, enable No Drag Moves to make area selection in dense areas of the design. • When No Drag Move is enabled, you can move the selected object by right-clicking, click Move, move the object to the new location, and click to finish the move.
Drawing area	

Table 45-233. Global / General Page Contents (cont.)

Name	Description
Keep same view on window resize	Specifies to maintain the area view of the design when you resize the PADS Layout window, by automatically zooming in or out.
Active layer comes to front	Specifies to display the active layer on top of all other layers. **Tip:** Specify the active layer in the Layers list on the main toolbar.
Minimum display width	Specifies the minimum width in current design units of lines you want to draw at actual width. Lines smaller than this width are drawn only as center lines to save memory and redraw time. **Tips:** • Set this value to zero to display all lines at actual width. • Set this to a value larger than the trace width. This displays traces as center lines and makes selecting small trace segments easier. Use this in combination with a reduced Pick radius value for optimal results.
OLE Document Server area	
Display OLE objects	Specifies to display linked or embedded objects inserted in PADS Layout. **Tip:** You may want to disable this option to decrease redraw times when PADS Layout contains many linked or embedded objects.
Update on redraw	Specifies to update the linked or embedded object in the container application. **Restriction:** This option applies only when you are modifying the PADS Layout object in a separate window and you click the Redraw button in the separate window. **Tip:** To increase performance, disable this option.
Draw background	Specifies to draw the PADS Layout background color in the linked or embedded object. When this option is disabled, the background of the PADS Layout object is transparent and you can see through the object at the container application's background.

Table 45-233. Global / General Page Contents (cont.)

Name	Description
Text encoding area	Specifies the language you want. Text encoding determines how text characters are interpreted. Each character in any text string has a unique digital signature. Within each system font are graphics (for instance, an image of the letter A) associated with each digital signature. Most system fonts have multiple images for the same character in order to accommodate regional differences in font styles. **Tip:** Changing the text encoding option may result in blank spaces or non-printing characters. **Example:** When you load a design created in Japan with Japanese text encoding on an American system, the file may have text encoding set to Japanese. If you change it from Japanese to English, Japanese kanji characters may be interpreted (and displayed) as non-printing characters. **Restriction:** The default text encoding cannot be changed. It is automatically set by the Regional and Language settings of the operating system.
Design units area	Specifies the design measurement units. • Mils— 1/1000 of an inch, accurate to 2 places after the decimal • Metric— 1mm (1/1000 of a meter), accurate to 5 places after the decimal • Inches— accurate to 5 places after the decimal Designs typically contain a mixture of metric and English (imperial/inch) components. And depending on the ratio of components in your design, you may want to use one measurement instead of the other. You can also switch design units as often as you like. This is done smoothly and efficiently since the accuracy of each unit of measure is the same.

Options Dialog Box, Global / Synchronization Page

Use the Global / Synchronization page to specify the PADS Layout and PADS Router syncronization options

Accessing

- **Tools** menu > **Options** > **Global / Synchronization**

Figure 45-253. Global / Synchronization page

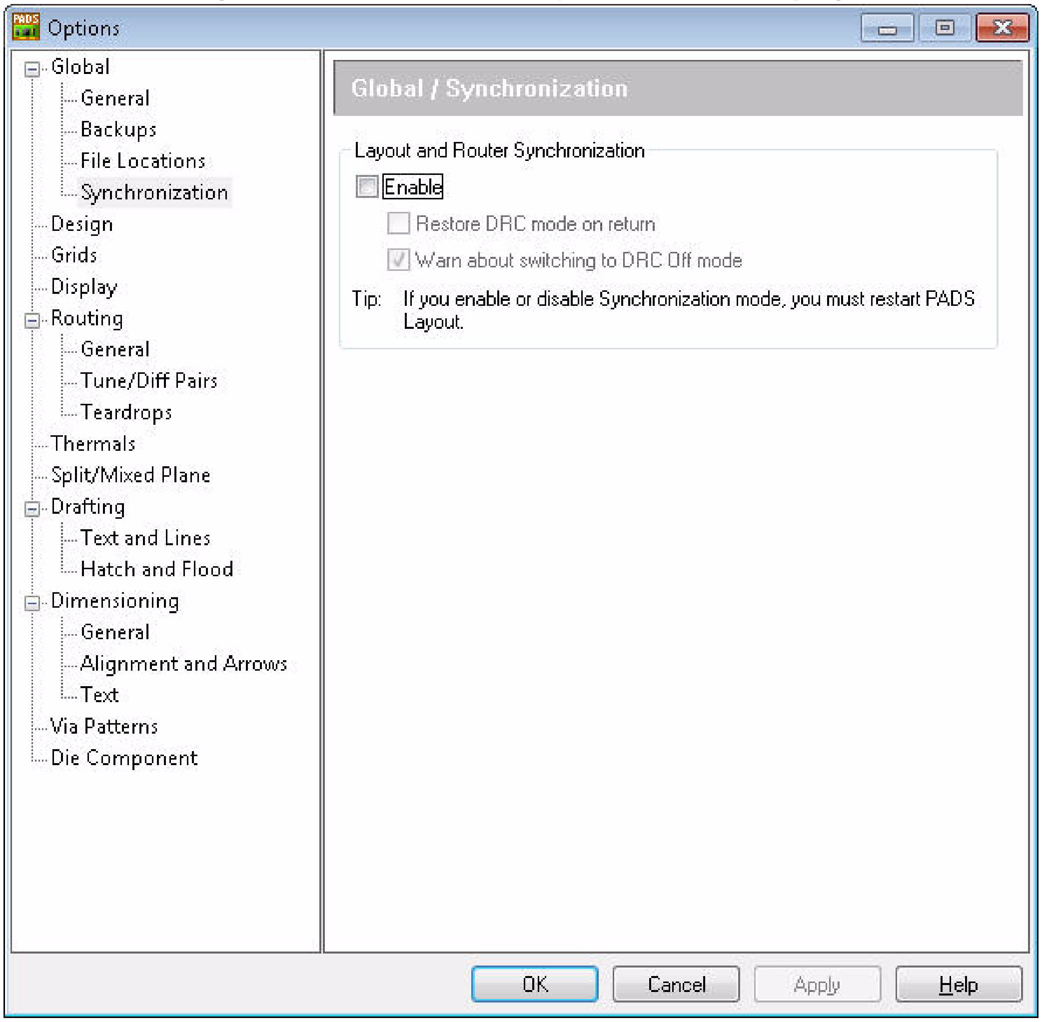


Table 45-234. Global /Synchronization page contents

Name	Description
Layout and Router Synchronization area **See also:** Synchronization Mode	
Enable	Specifies to turn Layout and Router Synchronization mode on. Click to clear to turn Synchronization mode off. **Warning:** You must restart Layout for the change in Synchronization mode to take effect.

Table 45-234. Global /Synchronization page contents (cont.)

Name	Description
Restore DRC mode on return	Specifies, if you return to PADS Layout from PADS Router after switching, to change the DRC mode back to what you had before you switched: DRC Prevent, Warn, or Ignore. **Tips:** • Switching to PADS Router automatically places your design in PADS Layout in DRC Off mode; the DRC mode in PADS Router is not affected. • Depending on the size of your design, restoring the DRC mode may take a few minutes; therefore, it is recommended to not restore your DRC mode upon return.
Warn about switching to DRC Off mode	Specifies to show a warning to remind you that Synchronization mode puts PADS Layout into DRC Off mode.

Options Dialog Box, Grids Page

Use the Grids page to specify grid options.

Accessing

- **Tools** menu > **Options** > **Grids**

Figure 45-254. Grids page

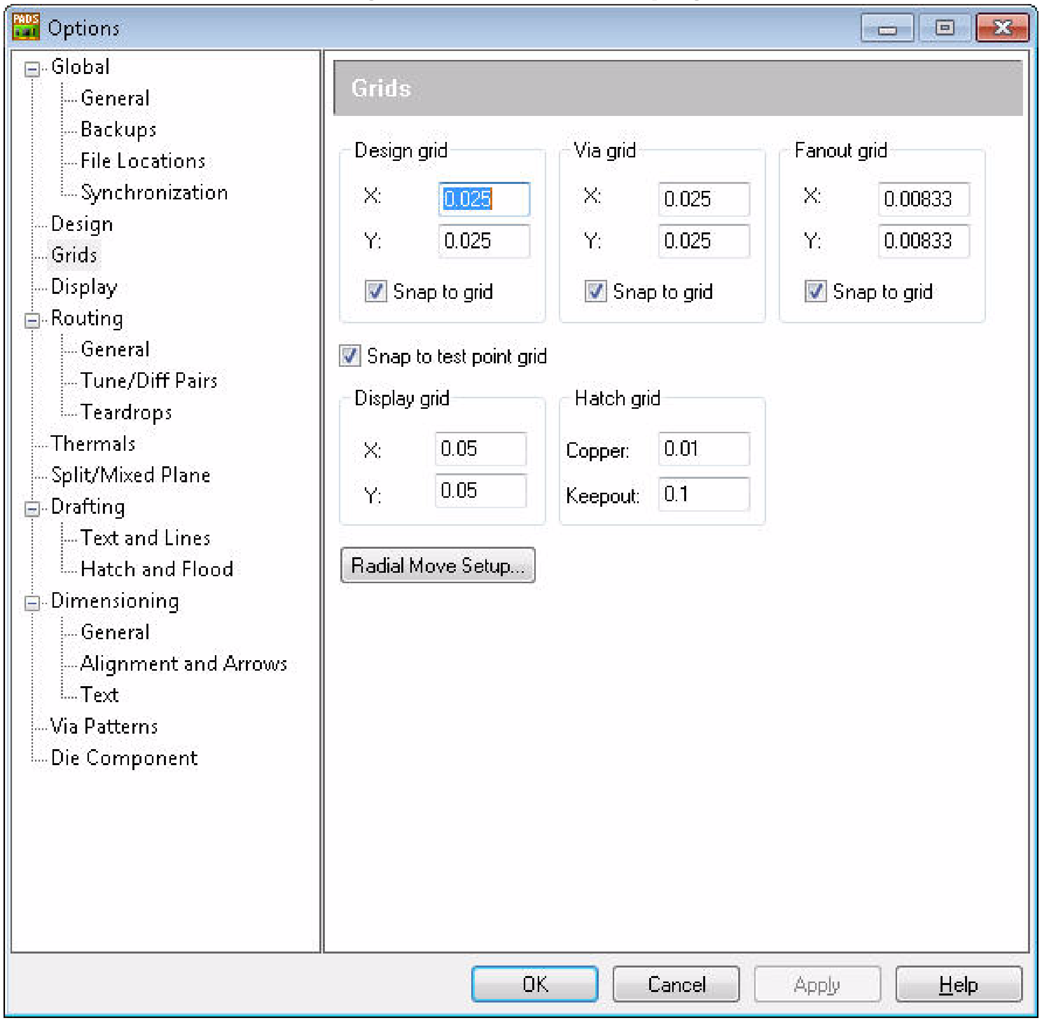


Table 45-235. Grids page contents

Name	Description
Design grid area	Establishes the minimum snap distance for the pointer when editing (placing objects, drawing/editing drafting objects). • **X**—Specifies the distance between grid lines on the X axis in current design units. • **Y**—Specifies the distance between grid lines on the Y axis in current design units. • **Snap to grid**—Specifies to snap objects from grid point to grid point, instead of freely and smoothly as you move or place the object. **Tip:** If the Snap to Grid check box is selected, you cannot place a part off the grid. **See also:** To Set the Design Grid
Via grid area	Establishes the minimum distance between vias. • **X**—Specifies the distance between grid lines on the X axis in current design units. • **Y**—Specifies the distance between grid lines on the Y axis in current design units. • **Snap to grid**—Specifies to snap objects from grid point to grid point, instead of freely and smoothly as you move or place the object. **Tip:** If the Snap to Grid check box is selected, you cannot place a via off the grid.
Fanout grid area	Controls the placement of substrate bond pads on a die and the placement of fanout vias. **Tip:** This data is passed to PADS Router. • **X**—Specifies the distance between grid lines on the X axis in current design units. • **Y**—Specifies the distance between grid lines on the Y axis in current design units. • **Snap to grid**—Specifies to snap objects from grid point to grid point, instead of freely and smoothly as you move or place the object.
Snap to test point grid	Enable snap to test point grid in PADS Router.

Table 45-235. Grids page contents (cont.)

Name	Description
Display grid area	Specifies the visible dot grid. • **X**—Specifies the distance between grid lines on the X axis in current design units. • **Y**—Specifies the distance between grid lines on the Y axis in current design units. **Tip:** If you want to make the dot grid invisible, set the X and Y values to 0. **See also:** To Set the Display Grid
Hatch grid area	Specifies the distance between hatch lines of copper and keepout areas. • **X**—Specifies the distance between grid lines on the X axis in current design units. • **Y**—Specifies the distance between grid lines on the Y axis in current design units. **Tip:** Copper, copper pour, and plane areas are filled with lines on the hatch grid. When the Drafting option Default width matches the Copper hatch grid, the result is a solid. When the default width less than the grid value, the result is a hatch pattern.
Radial Move Setup	Opens the Radial Move Setup dialog box to specify the polar grid for radial moves. **See also:** To Set Up a Polar Grid.

Options Dialog Box, Routing / General Page

Use the Routing / General page to specify routing options.

Accessing

- **Tools** menu > **Options** > **Routing / General**

Figure 45-255. Routing / General Page

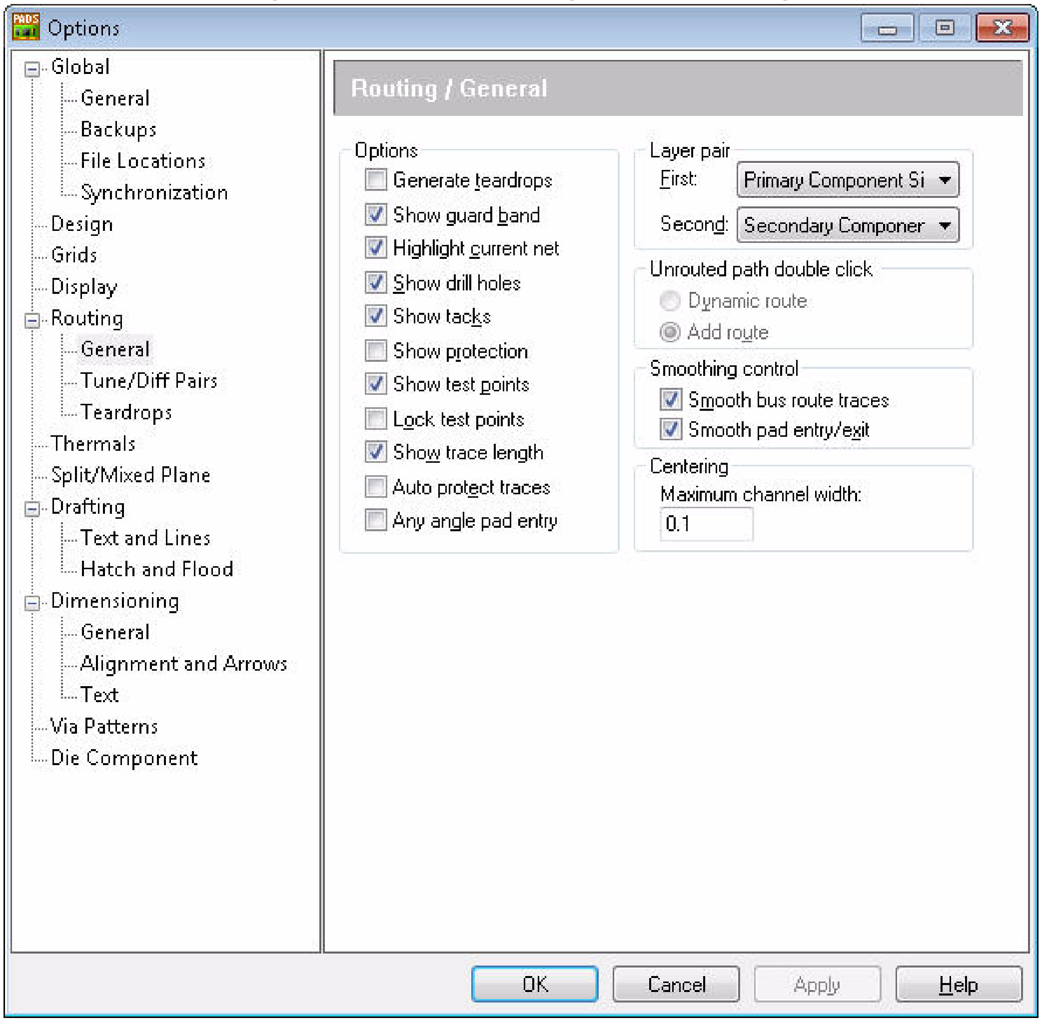


Table 45-236. Routing / General page contents

Name	Description
Options area	
Generate teardrops	Specifies to automatically create teardrops to existing and new trace segments that enter or exit a pad or via.
Show guard band	If On-line DRC is set to prevent, specifies to display an octagon shape at the tip of the current route to illustrate a clearance violation.
Highlight current net	When you start routing the selected pin pair, specifies to highlight the net with a complementary color.

Table 45-236. Routing / General page contents (cont.)

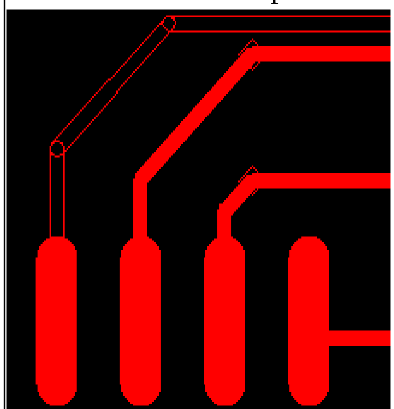
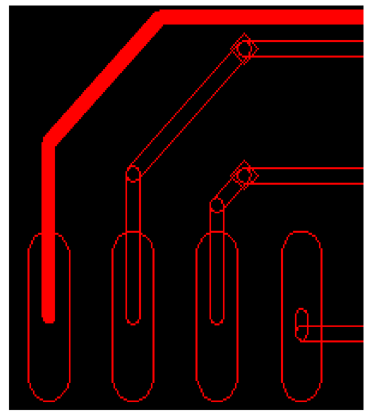
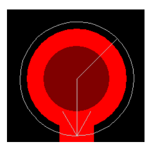
Name	Description
Show drill holes	Specifies to display the inside diameter of all pads. The modeless command “DO” toggles this setting. The drill hole is given a complementary color.
Show tacks	Specifies to display tacks on routes. **Tip:** If displayed tacks hinder routing or viewing the design, disable this option.
Show protection	Specifies to display protected routes as outlines when Outline View mode is off, and as normal traces when Outline View mode is on. In the image below, the left-most trace is protected and the other traces are not protected.   Outline View mode off Outline View mode on
Show test points	Specifies to display test points. **Tip:** When the via or pin is flagged as a test point, an arrow is drawn on it in the design: 
Lock test points	Specifies to prevent moving test points when moving components. Locked test points are not deleted when you do any of the following: • Unroute pin pairs or nets • Delete trace segments, vias, or jumpers • Change the layer for a segment

Table 45-236. Routing / General page contents (cont.)

Name	Description
Show trace length	Specifies to display the trace length monitor at the pointer. The status bar always reports trace length. **Tips:** • You can also use the modeless command Ctrl+PageUp to toggle the display the trace length monitor. • Displaying or hiding the trace length monitor with the modeless command does not end the current routing command, so you can continue to route.
Auto protect traces	Specifies to protect traces from smoothing, stretching, moving, shoving, or ripup operations. **Tip:** This option affects manual routing (Route or Add Route command) and dynamic routing.
Any angle pad entry	Specifies to allow traces to enter or exit a pad at any angle, despite the current Line/trace angle setting on the Design options tab.
Layer pair area	Specifies the pair of layers to use for routing when manually adding vias. **See also:** Using the Layer Pair, To Change the Layer While Routing **Tips:** • When you add a via while manually routing, the layer automatically switches to the other layer in the layer pair. • If you specify the layer pair as the two layers on which you expect to do most of your routing, you can reduce the number of times you manually specify layers when adding a via. • If the current layer is not in the layer pair when adding a via, the layer automatically switches to the layer specified in the First list.
Unrouted path double click area	Specifies a routing operation to start when you double-click unroutes. • **Dynamic Route**—Start a dynamic route • **Add Route**—Start a manual route **Restriction:** These options are unavailable unless On-line DRC is set to Prevent Errors in the Design options (or using the “DRP” modeless command.
Smoothing control area	
Smooth bus route traces	Specifies to run smoothing after bus routing. **Tip:** This option affects only the Bus Route command, and it inhibits the global smoothing pass for all traces of the current bus.

Table 45-236. Routing / General page contents (cont.)

Name	Description
Smooth pad entry/exit	Specifies to have traces that enter a pad at a 90-degree angle converted to a 45-degree angle during a trace segment smoothing operation.
Centering area	Maximum channel width—Limits the number of channels in which traces are automatically centered by specifying a maximum channel width. Traces in channels larger than this width are not eligible for centering. **Tip:** This option is used only by PADS Router.

Options Dialog Box, Routing / Teardrops Page

Use the Routing / Teardrops tab to specify the display and physical appearance of teardrops. Use the Routing / General page to enable teardrops.

Accessing

- **Tools** menu > **Options** > **Routing / Teardrops**

Figure 45-256. Routing / Teardrops page

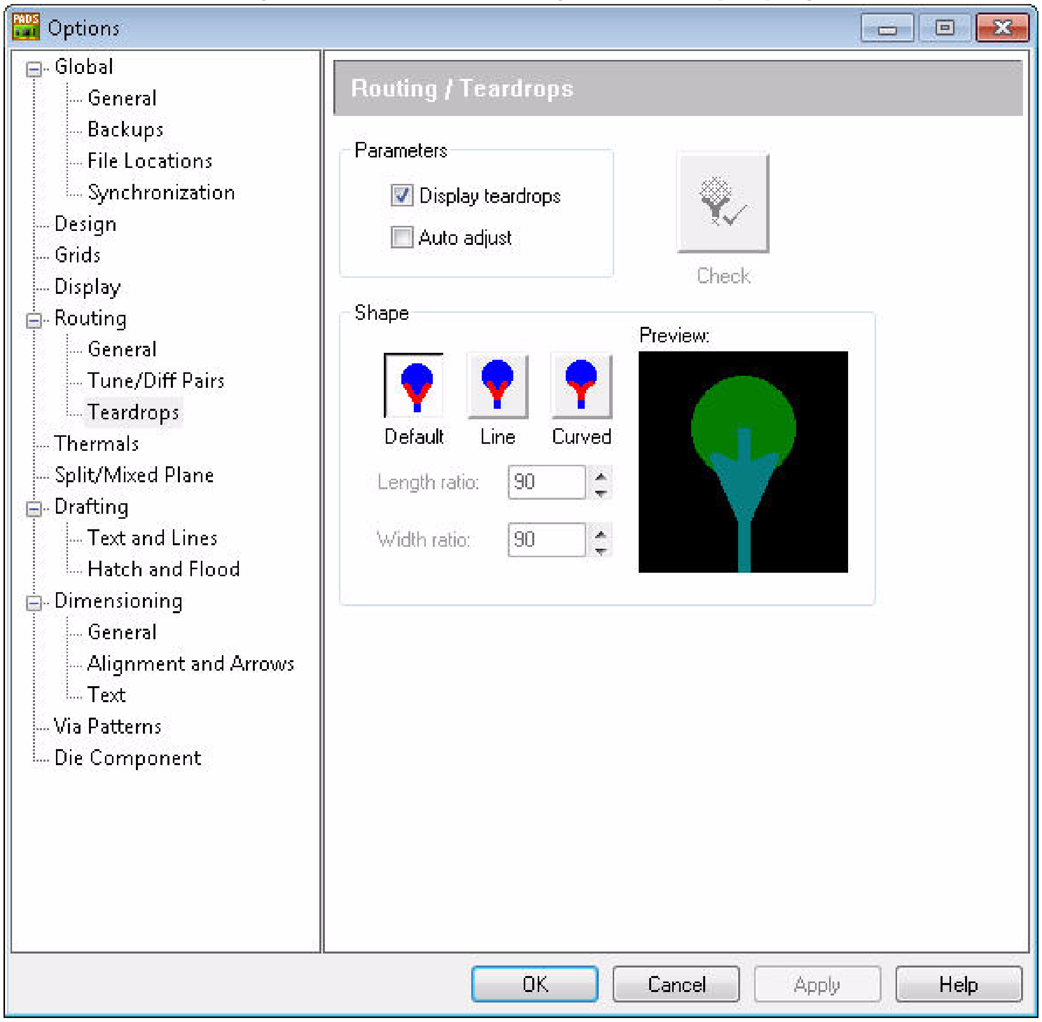


Table 45-237. Routing / Teardrops page contents

Name	Description
Parameters area	

Table 45-237. Routing / Teardrops page contents (cont.)

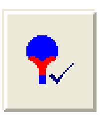
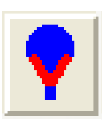
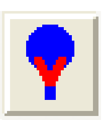
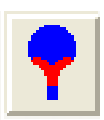
Name	Description
Display teardrops	Specifies whether to display teardrops. Hiding the teardrops could reduce the redraw time. Although you hide the teardrops in the design, they will be visible in CAM output. **Tip:** You create the teardrops using the *Generate teardrops* check box on the Routing / General page. **Warnings:** • If you clear this checkbox and hide the teardrops, you will not be able to verify any clearance errors created by the teardrops. Clearance checking only checks objects that are displayed. • If you flood an area while hiding the teardrops, the flood procedure will not account for the teardrops. If you don't flood the shape again prior to generating your CAM documents, you will create a CAM document without enough clearance around all the teardrops.
Auto adjust	Specifies to have PADS Layout attempt to adjust the length of teardrops on traces where the trace corner is inside the pad or via, or where the segment is too short to contain the specified length ratio. **Restriction:** You can specify teardrop length and width ratios only if you select the Line or Curved teardrop shape.
	Opens the Check Teardrop dialog box where you can check the design for teardrop errors and report them. **Restriction:** The Check button is unavailable unless the Generate teardrops check box on the Routing tab is selected.
Shape area	
  	Specifies the detailed physical appearance of teardrops. The Preview area shows the current teardrop shape. • **Default**—Standard shape. **Restriction:** You cannot set the length or width ratio for the Default shape. • **Line**—Outer teardrop edges are straight. • **Curved**—Outer teardrop edges are curved. **Tip:** If the board has high-frequency analog circuits or very dense connections, you may want to specify either the Line or Curved shape.

Table 45-237. Routing / Teardrops page contents (cont.)

Name	Description
Length ratio	Specifies the length ratio in percent of pad diameter. **Restrictions:** • You cannot set a ratio over 1000. • Not available for the Default shape. **Tip:** This option sets the length of the teardrop relative to the attached pad. The formula used to calculate the teardrop length follows: `teardrop length = (pad diameter) * (length ratio in %)` **Example:** If the length ratio is 200 (200% of the pad diameter) and the pad diameter is 60 mils, then the length of the teardrop is 120 mils.
Width ratio	Specifies the width ratio in percent of pad diameter. Restrictions: • You cannot set a ratio over 100. • Not available for the Default shape. **Tip:** This option sets the width of the teardrop relative to the attached pad.

Options Dialog Box, Routing / Tune/Diff Pairs Page

Use the Routing / Tune/Diff Pairs page to specify options for using accordions in routing to length constraints and tuning differential pairs.

Restriction: While you can define these options in either PADS Layout or PADS Router, the settings are only used in PADS Router.

Use routing to length constraints to adjust the length of length-controlled traces. This feature automatically maintains length-based design rules for nets, classes, pin pairs, groups, and differential pairs during autorouting. These settings use the tune pass to adjust or rip up traces based on their compliance with minimum and maximum trace length rules. The pass increases net and pin pair lengths to satisfy length rules by introducing accordion patterns to the trace.

Tip: You can set length rules in Class, Net, Pin Pair, Differential Pair, and (PADS Router only) Matched Length Properties dialog boxes.

Accessing

- **Tools** menu > **Options** > **Routing / Tune/Diff Pairs**

Figure 45-257. Routing / Tune/Diff Pairs page

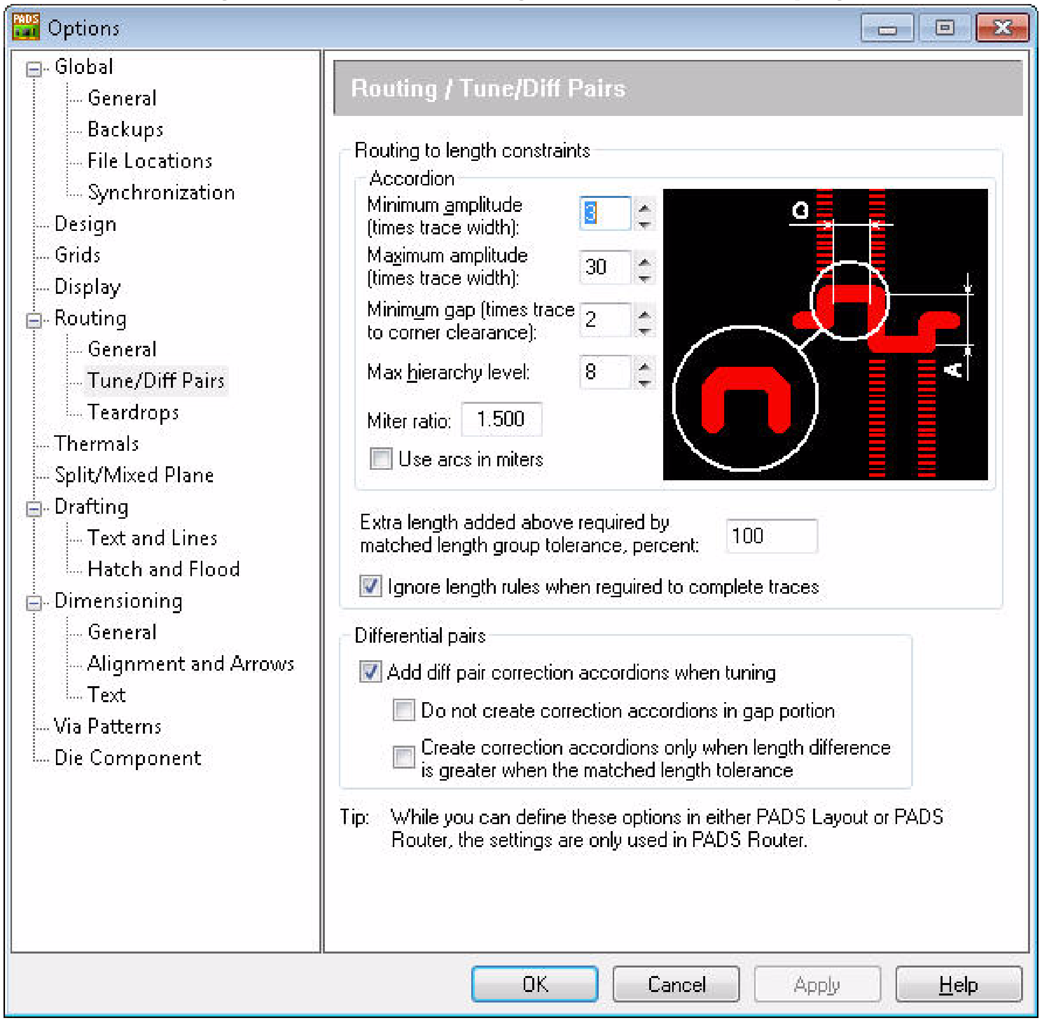


Table 45-238. Tune/Diff Pairs page contents

Name	Description
Accordion area	
Minimum amplitude	Specifies the minimum height (for horizontal accordions) or width (for vertical accordions).
Maximum amplitude	Specifies the maximum height (for horizontal accordions) or width (for vertical accordions).

Table 45-238. Tune/Diff Pairs page contents (cont.)

Name	Description
Minimum gap	Specifies the edge to edge distance between accordions. The gap is equal to the value of the parameter times the trace-to-corner clearance. However, if the trace to corner clearance equals 0, then the gap is equal to the trace width times the value of the gap parameter.
Max hierarchy level	Specifies how many steps are used to create the accordion. **See also:** Maximum Hierarchy Level
Max ratio	Specifies the miter ratio for accordion corners.
Use arcs in miters	Specifies to use an arc instead of a diagonal segment in the accordion.
Extra length added above required by matched length tolerance, percent	Specifies how much extra length is added above the required matched length group tolerance (in percent of the tolerance). **Example:** If you type 0, the tuned net will be <Leader length - tolerance> length. If you type 100, the net will get the same length as the group leader. The leader net is the net in the matched length group that has the longest length.
Ignore length rules when required to complete traces	Specifies to ignore length rules so you can complete traces.
Differential pairs area	
Add diff pair correction accordions when tuning	Specifies to use an accordion to make differential pairs the same length.
Do not create correction accordions in gap portion	Specifies that you do not want to allow correction accordions where two traces go together at the gap.
Create correction accordions only when length difference is grater than the matched length tolerance	Ensures that correction accordions are not created when the differential pair net length difference is less than the tolerance of the matched length group.

Options Dialog Box, Split/Mixed Plane Page

Use the Split/Mixed page to specify options for split and mixed planes. Split/mixed planes use the same global thermal attributes as copper pours, but in addition they use custom thermals from the pad stacks. All split/mixed plane options are stored in the design file.

Accessing

- **Tools** menu > **Options** > **Split/Mixed Plan**

Figure 45-258. Split/Mixed Plane page

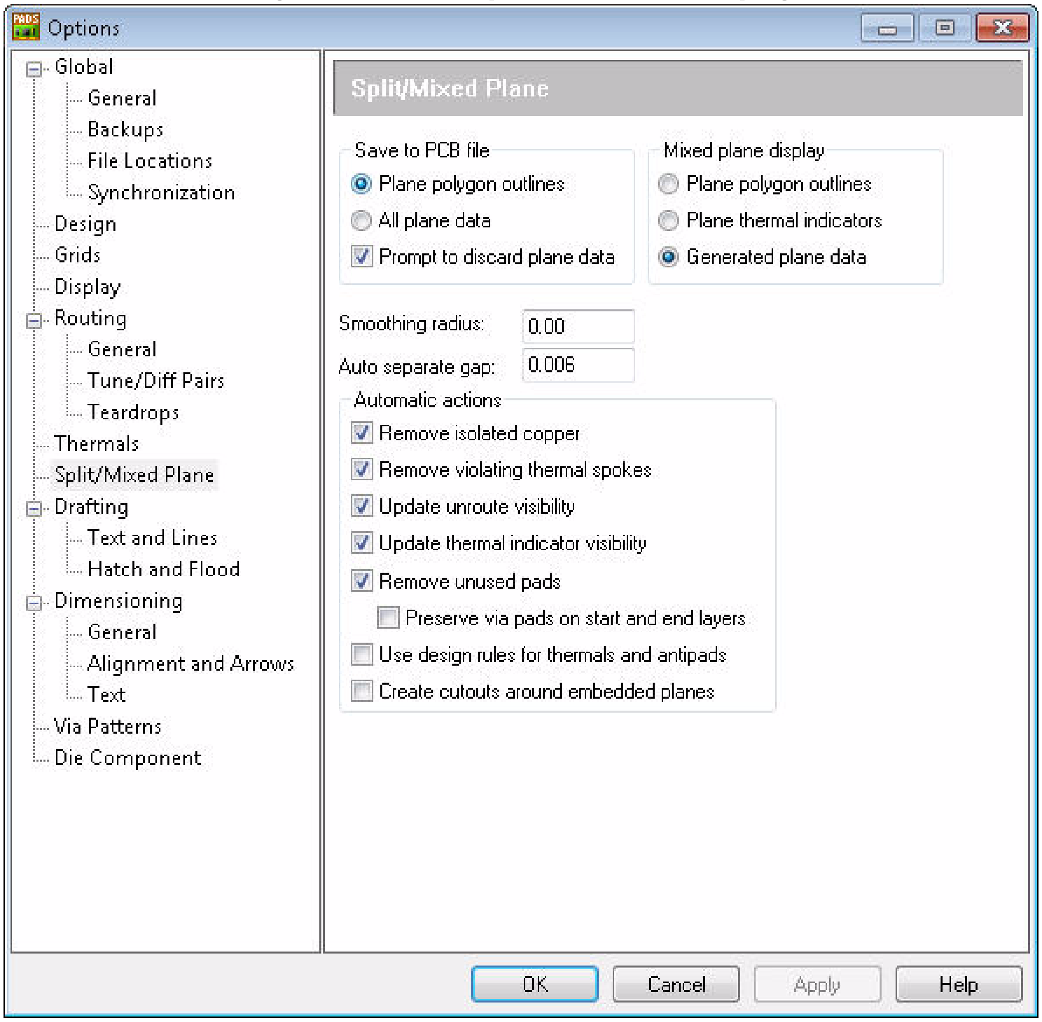


Table 45-239. Split/Mixed Plane page Contents

Name	Description
Save to PCB file area	Specifies whether to save various forms of plane data in the design file. • **Plane Polygon Outlines**—Saves only the mixed plane area polygons in the PADS Layout file. Discards generated thermals and antipads. **Tip:** Saving only the plane polygon outlines results in a smaller PADS Layout file and a faster design load time. • **All Plane Data**—Saves all data associated with mixed planes in the PADS Layout file. **Tip:** The fill or hatching of plane areas is never saved in the design file.
Prompt to discard plane data	Specifies to display the Discarding Plane Data dialog box every time you save the design. **Tip:** This option is unavailable if the All Plane Data option is selected.
Mixed Plane display area	Specifies which plane data to display in PADS Layout. • **Plane Polygon Outlines—**Displays only the plane area polygons. • **Plane Thermal Indicators—**Displays the plane area polygons, thermal reliefs, and antipads. • **Generated Plane Data**—Displays all data associated with mixed planes.
Smoothing radius	Specifies how much smoothing to apply to sharp corners of mixed planes. **Tips:** • Zero radius results in no smoothing. Larger radius values produce smoother and rounder corners. • The smoothing radius affects only plane area data types and is independent of the smoothing radius for copper pours.
Auto separate gap	Specifies the plane-to-plane gap used by the Auto Separate command and Create Plane Area commands. **Tip:** The default value is the Board-to-Trace clearance.
Automatic actions area	
Remove isolated copper	Remove copper areas on a mixed plane that are not connected to nets. **Tip:** This option also appears on the Thermals tab. Changing this option here also changes the option on the Thermals tab.

Table 45-239. Split/Mixed Plane page Contents (cont.)

Name	Description
Remove violating thermal spokes	Removes thermal spokes that cause clearance violations on split/mixed plane layers
Update unroute visibility	Makes unroutes invisible when a connection is made to the split/mixed plane
Update thermal indicator visibility	Updates the plane thermal indicator visibility status of pads
Remove unused pads	Removes unused pads and replace them with antipads. **Tip**: CAM planes using custom antipads defined in the Pad Stacks require this setting, otherwise the default antipad setting will be used instead.
Preserve via pads on start and end layers	For partial vias, specifies to not remove the Start pad or End pad if the pad is on a plane layer. **Tip:** This check box is unavailable if Remove Unused Pads is disabled.
Use design rules for thermals and antipads	When flooding during a plane connection operation, specifies to use the hierarchical Pad to Copper clearance rule for thermals and the Drill to Copper clearance rule for antipads. **Tips:** • Enabling this option ignores the outer width/diameter/size settings for custom thermals and the width/diameter/size settings for custom antipads. Also, the external outlines of thermals and antipads are not displayed. • This option does not affect custom thermals for which the outer width/diameter/size is less than, or equal to, the inner width/diameter/size in the flood over settings.
Create cutouts around embedded planes	Automatically creates cutouts around plane areas if the area is placed inside another. **Tip:** The cutout is combined with the external area.

Options Dialog Box, Thermals Page

Use the Thermals page to specify options for copper pours automatically generated for existing pour outlines.

Restriction: These options only apply to Split/Mixed layer thermals when there are no custom thermals in the pad stack. Copper pours never use custom thermals from the pad stack.

Accessing

- **Tools** menu > **Options** > **Thermals**

Figure 45-259. Thermals Page

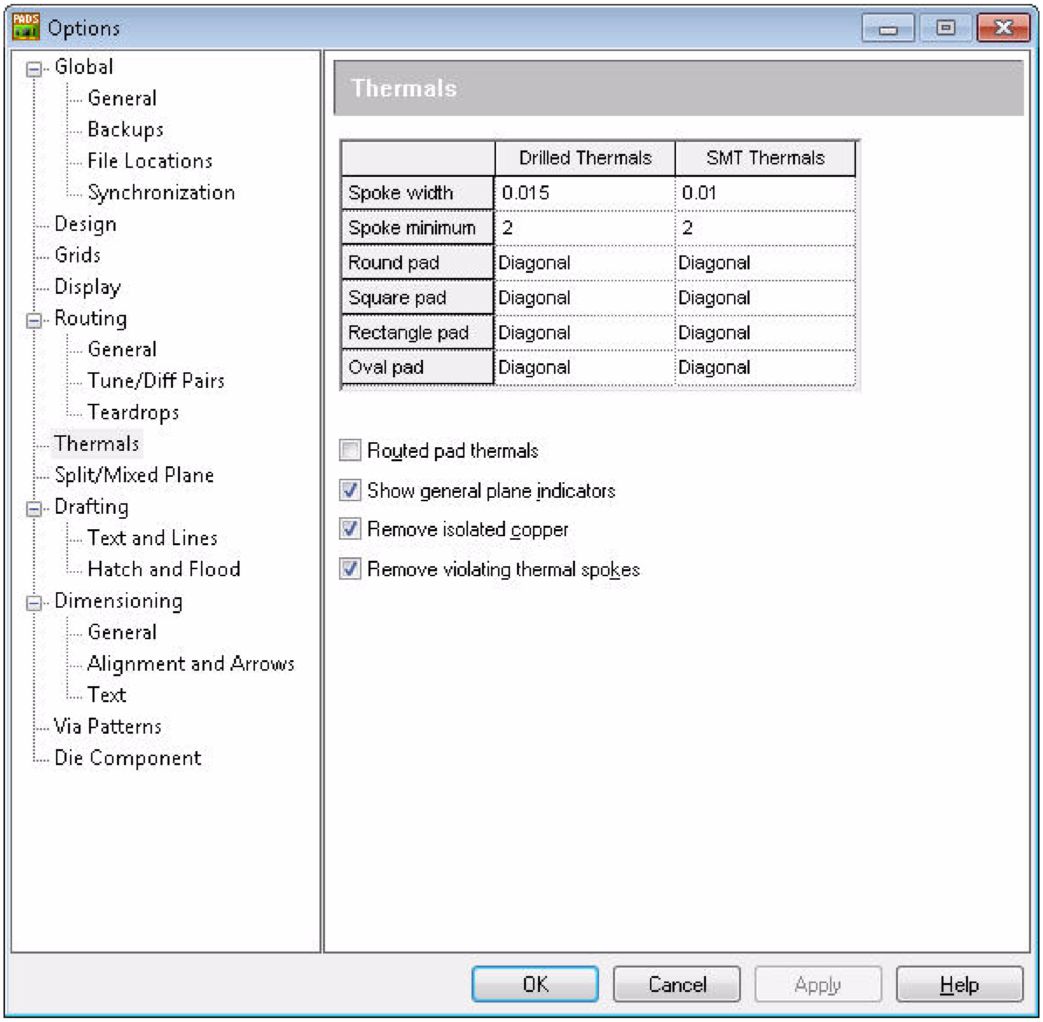


Table 45-240. Thermals page Contents

Name	Description
Thermal Shape/Size Table - Drilled thermal settings are for through hole pad stacks and SMT thermals are for surface mount or any non-drilled thermal.	
Width	Specifies the line width for the thermal relief in current design units.

Table 45-240. Thermals page Contents (cont.)

Name	Description
Min. Spoke	Specifies the minimum number (1-4) of spokes. **Tip:** If the pad intersects the boundary of a copper pour, it may not be possible to create all four spokes. A warning appears when thermals are created with fewer spokes than the minimum.
Pad shapes - Round, Square, Rectangle, Oval	Specifies the shape of the thermal relief: Round, Square, Rectangle, or Oval
Relief Shape	• **Orthogonal**—Creates orthogonal-shaped thermal reliefs. • **Diagonal**—Creates diagonal-shaped thermal reliefs. • **Flood Over**—Creates thermal reliefs that flood over the pad. You can also configure the copper pour or plane area to flood over vias. **See also:** Flood and Hatch Options dialog box • **No Connect**—Creates no thermal reliefs.
Routed pad thermals	Specifies that you can place thermals on routed pads or connections. Usually thermals are placed only on unrouted connections. **Restriction:** This Option applies only to copper pours and not to split planes. **Tip:** Small left-over trace segments attached to a pad prevents a pad from receiving thermals if the Routed Pad Thermals option is not enabled. Set the Selection Filter to Traces, Corners, and Tacks to select and remove trace segments attached to the pad.
Show general plane indicators	Specifies to display a general plane thermal indicator if a connection to a CAM or split/mixed plane exists somewhere within a pad stack. The physical appearance of this indicator resembles a small x in the center of the pad. Disable this option if you do not want to view indicators. **Tips:** • General plane indicators are independent of the current layer. For example, the actual indicator may not be on the current layer, but on one or more internal layers of the pad stack. • General plane indicators can sometimes be covered during editing. Redraw the screen to view all of the indicators.

Table 45-240. Thermals page Contents (cont.)

Name	Description
Remove isolated copper	Specifies to automatically remove isolated copper areas, which are areas not connected to nets, during the flood operation of regular copper pours **Tip:** This option also appears on the Split/Mixed tab. Changing this option here also changes the option on the Split/Mixed tab.
Remove violating thermal spokes	Specifies to automatically remove thermal spokes that cause clearance violations on non-plane layers. **Tip:** This option also appears on the Split/Mixed tab. Changing this option here also changes the option on the Split/Mixed tab.

Related Topics

Customizing Design Rule Thermals

Flooding Over Pads in a Copper Pour or Plane Area

Options Dialog Box, Via Patterns Page

Use the Via Patterns page of the Options dialog box to set options for the via shielding and via stitching operations.

Requirements:

- You must have a design open in order to set Via Patterns options.
- Set Via Patterns options before using the Add Via Shield or Via Stitch commands.

Accessing

- **Tools** menu > **Options** > **Via Patterns**

Figure 45-260. Via Patterns page

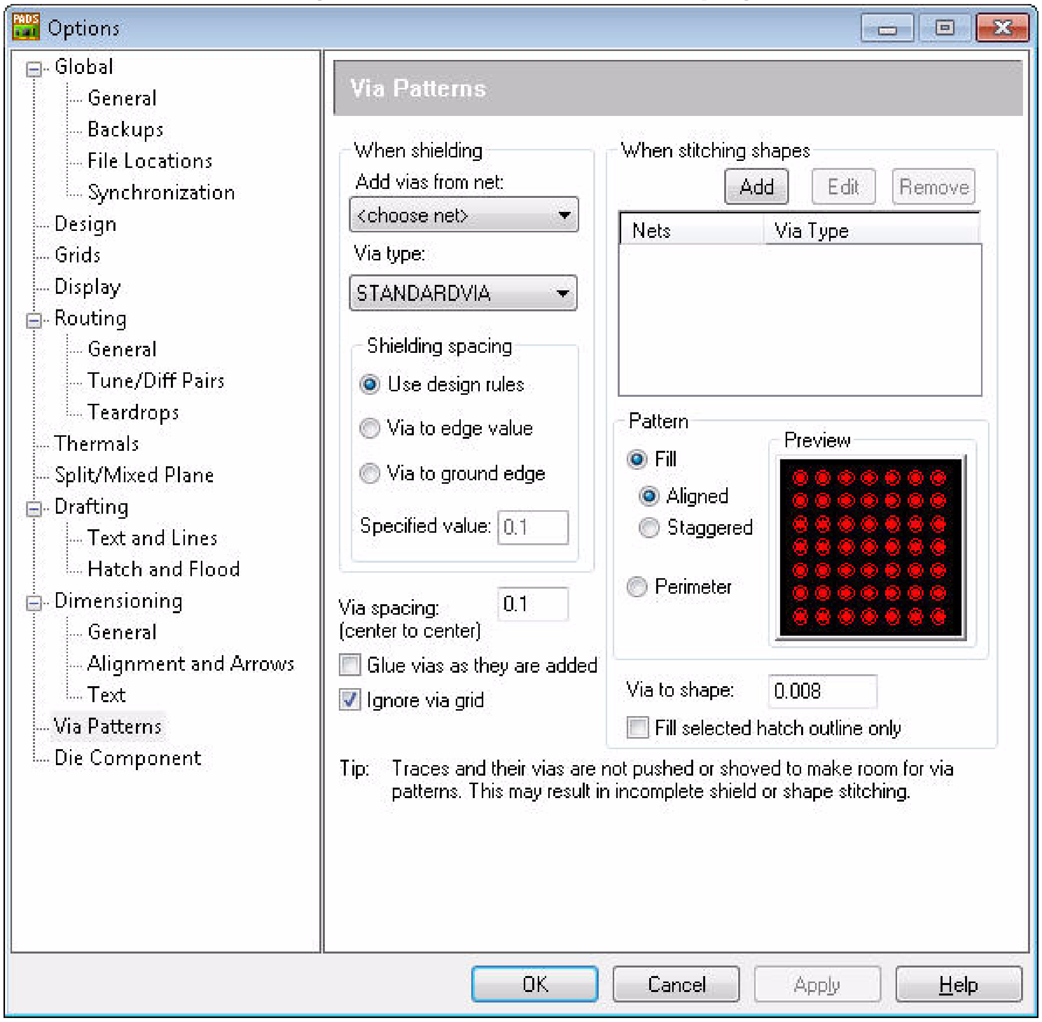


Table 45-241. Via Patterns page contents

Name	Description
When shielding area—Use these options to specify the via type for vias used for shielding.	
Add vias from net	Specifies the via type for vias used for shielding. Select the net associated with the vias used for shielding.
Via type	Specifies the via type for vias used for shielding. 1. In the Add vias from net list, select a net to display available via types. 2. In the Via type list, select the via type to use for shielding. (Design rules determine which via types are available for the selected net.)

Table 45-241. Via Patterns page contents (cont.)

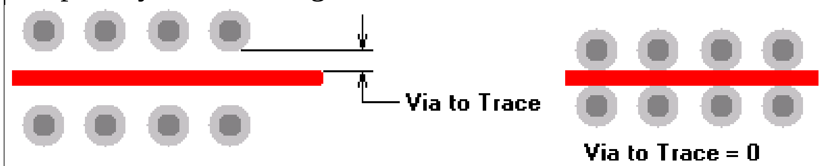
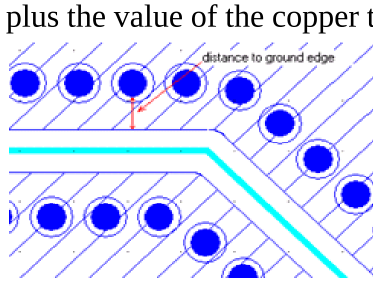
Name	Description
Shielding spacing area	Specify the distance from the shielding vias to the trace or shape they are shielding  **Requirement:** Select the **Ignore via grid** check box. Otherwise, vias snap to the via grid. Select one of the following: • **Use design rules**—Select this option (the default) to have design rules determine the distance from vias to the object being shielded. The Add Via Shield operation uses Via to Trace and Via to Copper clearances to place vias in relation to a selected trace or copper. **Tip:** Design rules are used to place vias in relation to the selected trace or copper; however, that placement can violate clearances with other objects—another trace, for example. To prevent addition of any via that causes clearance errors, set Design Rule Checking (DRC) to Prevent Errors. For more information, see DRC and the Via Stitching and Shielding Operations. • **Via to edge**—Select this option to specify via spacing different from the design rules for minimum Via to Trace and Via to Copper clearances. Select this option and type a value in the **Specified value** box. The value can be 0 to 1000 mils. A value of 0 causes the vias and the edge of the trace (or shape) to touch. **Tip:** To specify a value smaller than the minimum Via to Trace or Via to Copper clearances, set DRC to Off. If the DRC setting is Prevent Errors, an Add Via Shield operation fails and no vias are added. • **Via to ground edge**—Specifies the distance from the vias to the edge of the copper ground area (for example, a hatch outline). Select this option and type a value in the **Specified value** box. **Tip:** The via to trace distance equals the value of Via to ground edge plus the value of the copper to trace clearance. 

Table 45-241. Via Patterns page contents (cont.)

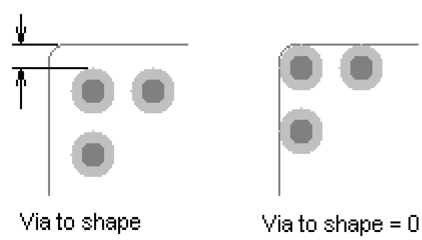
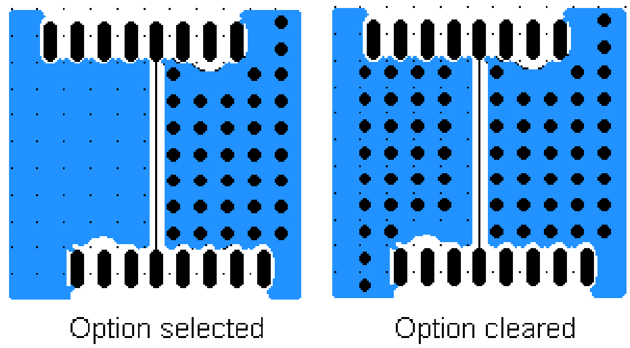
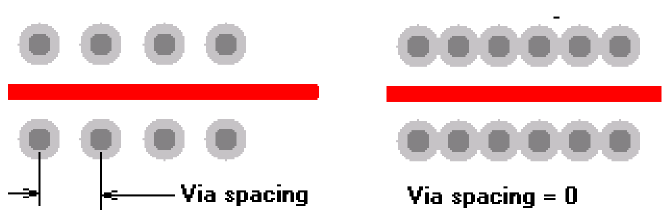
Name	Description
When Stitching Shapes area—Specifies the via type used in stitching a copper shape for a given net.	
Nets/Via Type list	Select the via type used per net for stitching shapes. By default, this list is empty. Use the following buttons to modify the list: • **Add—**Adds a row at the bottom of the When Stitching Shapes table. • **Edit—**Makes the selected cell available for editing. • **Remove—** Removes a line (Net/Via Type) from the list.
Pattern area	Specifies the stitching mode (Fill or Perimeter) for placing the pattern of vias in the shape. You have the following choices: • **Fill—**Fills the shape with the pattern (Aligned or Staggered). • **Perimeter—**Places vias inside the perimeter of the shape. **Tip:** To override the Pattern (Fill or Perimeter) setting for a selected shape, use the Via Stitch Mode command. **Restriction:** By default, the Via Stitch operation does not place vias around a void within a shape. For information, see Surrounding a Void with Vias.
Via to shape	Specifies the distance from the edge of the shape you are filling to the edge of the via pattern. The value can be 0 to 1000 mils. A value of 0 causes the edge of the via and the fill outline to touch. for example:  The default value is the same as the default Copper-to-Via clearance.

Table 45-241. Via Patterns page contents (cont.)

<table>
<tr><th>Name</th><th>Description</th></tr>
<tr><td>Fill selected hatch outline only</td><td>Use this setting when you have a single copper pour or split plane polygon that, when flooded, is separated into two or more distinct hatch outlines.

Select this option to fill only the selected hatch outline with the pattern of vias when you use the Via Stitch command. (This is the default setting.)
Clear the check box to fill the main pour or plane outline of which the selected hatch outline is a part.

For example:


</td></tr>
<tr><td colspan="2">Use the settings below for both the Add Via Shield and Via Stitch commands.</td></tr>
<tr><td>Via spacing</td><td>Specifies the distance between vias (center to center) added by the Add Via Shield and Via Stitch operations. Type a value of 0 to 1000 mils; the default is 100 mils.
If you specify a value of 0 or any value that is less than half of the diameter of the via, vias touch but do not overlap. For example:.


Tips:

To use this option, select the Ignore via grid check box.
To specify a value smaller than via diameter plus same net Via to Via clearance, set DRC to Off. If the DRC setting is Prevent Errors, an Add Via Shield or Via Stitching operation fails. (The operation adds some vias but skips those that create violations.)

</td></tr>
<tr><td>Glue vias as they are added</td><td>Specifies to glue each via added by the Add Via Shield and Via Stitch operations. Also sets each via’s property to Glued and stores the setting in the design database.</td></tr>
</table>

Table 45-241. Via Patterns page contents (cont.)

Name	Description
Ignore via grid	Allows the Add Via Shield and Via Stitch operations to ignore the via grid setting. (Vias do not snap to the via grid.) Instead, the operations place vias according to the settings you specified in the Shield spacing area and the Via spacing field.

Related Topics

Setting Via Shielding Options

Setting Shape Stitching Options

DRC and the Via Stitching and Shielding Operations

Adding a Via Shield

Filling a Shape with a Pattern of Vias

Placing Vias Inside the Perimeter of a Shape

Output Window

Use the Output window for displaying reports and session logs, macro editing and debugging, and custom programming and debugging.

Accessing

- **Output Window** button

The Output window is located in the lower left section of the display window. You can dock or float the Output window. You can also open or close the Output window.

The Output window has three tabs:

- Status Tab - Displays information on the current session.
- Macro Tab - Allows you to run, edit and debug macro scripts.

Related Topics

- Interface

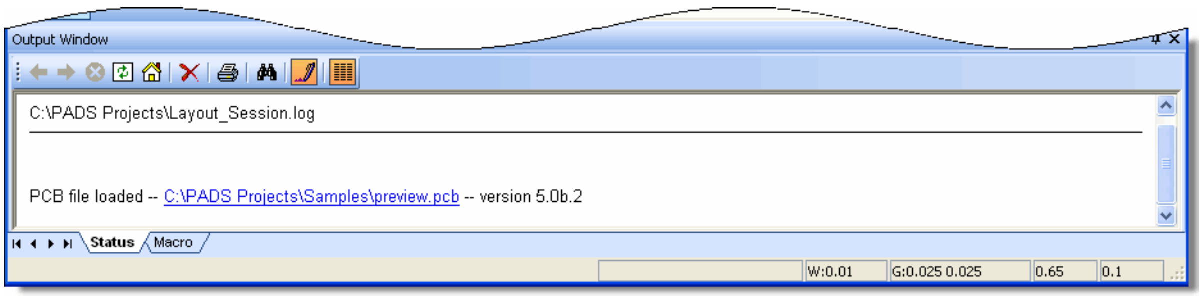
Status Tab

The Status tab displays information about the current session. It specifies the file name of the opened PCB file and the name of the test integrity file that is saved. It also reports routing statistics and messages when routing a board. If the Status tab is closed, and you get an error while autorouting - or performing other tasks - the Output window opens with the status tab active and the error appearing in red. The Output window reappears in its most recent state (floating or docked).

Accessing

- Click the Output Window button and then click the Status tab.

Figure 45-261. Status Tab



Related Topics

Managing Session Logs

Session Log in the *Routing Concepts Guide*

Session Log in the *PADS Layout Concepts Guide*

Macro Tab

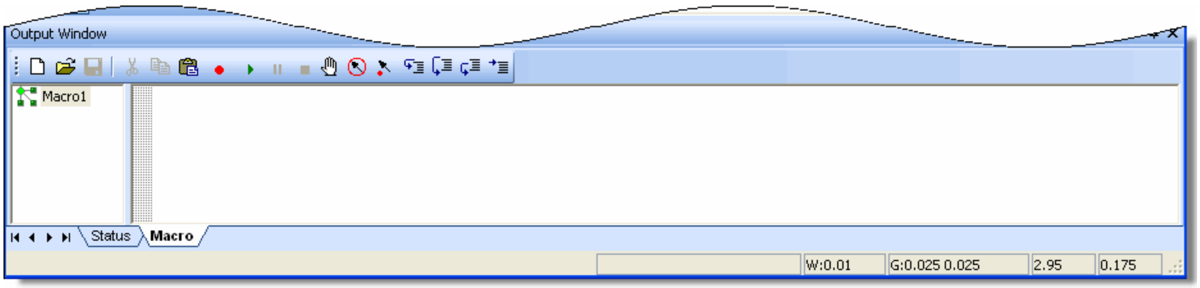
You can edit, run, and debug macro scripts in the Macro tab. You can open multiple macros and nest macros using the macro editor.

A macro is any combination of commands, keystrokes, and mouse clicks that you record to replay as a single action. You can record virtually any set of procedural steps for replay, thereby simplifying redundant activities, such as setting preferences and layer/display settings.

Accessing

- Click the Output Window button and then click the Macro tab.

Figure 45-262. Macro Tab



Related Topics

Creating Macros

Managing Macros

Playing Back Macros

Macros in the *Routing Concepts Guide*

Debugging Macro Scripts

Accessing Help on the Macro Language

Using Command Line Switches with Macros

Pad Entry Rules Dialog Box

Use the Pad Entry Rules dialog box to specify how traces enter and exit pads.

Restriction

While you can define these rules in either PADS Layout or PADS Router, the rules are only used in PADS Router.

Accessing

- **Setup** menu > **Design Rules** > **Default** button > **Pad Entry** button
-

Figure 45-263. Pad Entry Rules Dialog Box

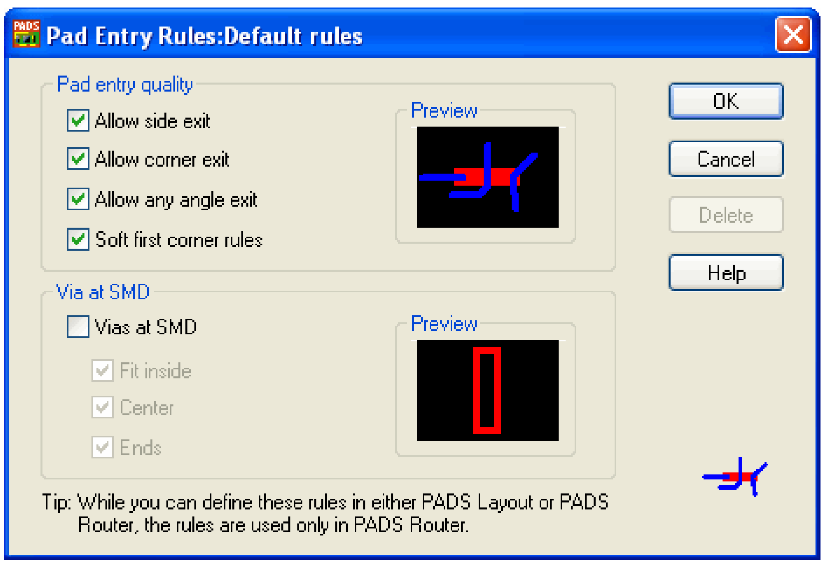


Table 45-242. Pad Entry Rules Dialog Box

Area	Description
Pad entry quality	Specify the options that PADS Router can use while routing from the pad: • **Allow side exit**—For rectangular pads only, enable routing to exit through the long side of the pad. • **Allow corner exit**—For rectangular pads only, enable routing to exit through the pad corner or arc. • **Allow any angle exit**—For rectangular and round pads, enable routing to exit at any angle, not just 45 or 90 degrees. • **Soft first corner rules**—For rectangular and round pads, enable routing to ignore first corner clearance rules and to exit at an angle less than 90 degrees. **Tip:** If you select this check box, acid traps may result. On the other hand if you clear this check box, lower completion rates and slower routing may result. Use DRC in PADS Layout to locate possible acid traps. You can ignore first corner clearance rules, because it is a soft rule.
Via at SMD	**Specify** to enable PADS Router to place vias on SMD pads while routing**.** **Restrictions:** For pins with associated copper, the via is not placed on the SMD pad. Only one via can be placed on an SMD pad. • **Vias at SMD**—Place vias on SMD pads while routing. **Tip:** Only one via can be placed on an SMD pad. • **Fit inside**—Fit the via entirely inside the SMD pad. • **Center**—Place the via at the geometric center (not pad origin) of the SMD pad. • **Ends**—For rectangular or oval pads, place the via at the end of the SMD pad. When using square pads, place the via at the midpoints of the sides of the pad. Round pads are ignored.
Delete button	Removes non-default pad entry rules from the current level of the rules hierarchy. **Restriction:** You cannot delete the Default pad entry rules.

Related Topics

Design Rule Hierarchy

Pad Entry Properties in PADS Router

Pad Stacks Properties Dialog Box

Use the Pad Stacks Properties dialog box to establish the size and shape of each pad and drilled hole in a given pad stack, including component and via pad stacks.

Accessing

- **Setup** menu > **Pad Stacks**

The Pad Stacks Properties dialog box controls change depending on what you have selected. The three major differences are:

- Figure 45-264: Pad Stacks Properties Dialog Box - Decal Pad Stack
- Figure 45-265: Pad Stacks Properties Dialog Box - Via Pad Stack
- Figure 45-266: Pad Stacks Properties Dialog Box - Thermal Pad Style

Figure 45-264. Pad Stacks Properties Dialog Box - Decal Pad Stack

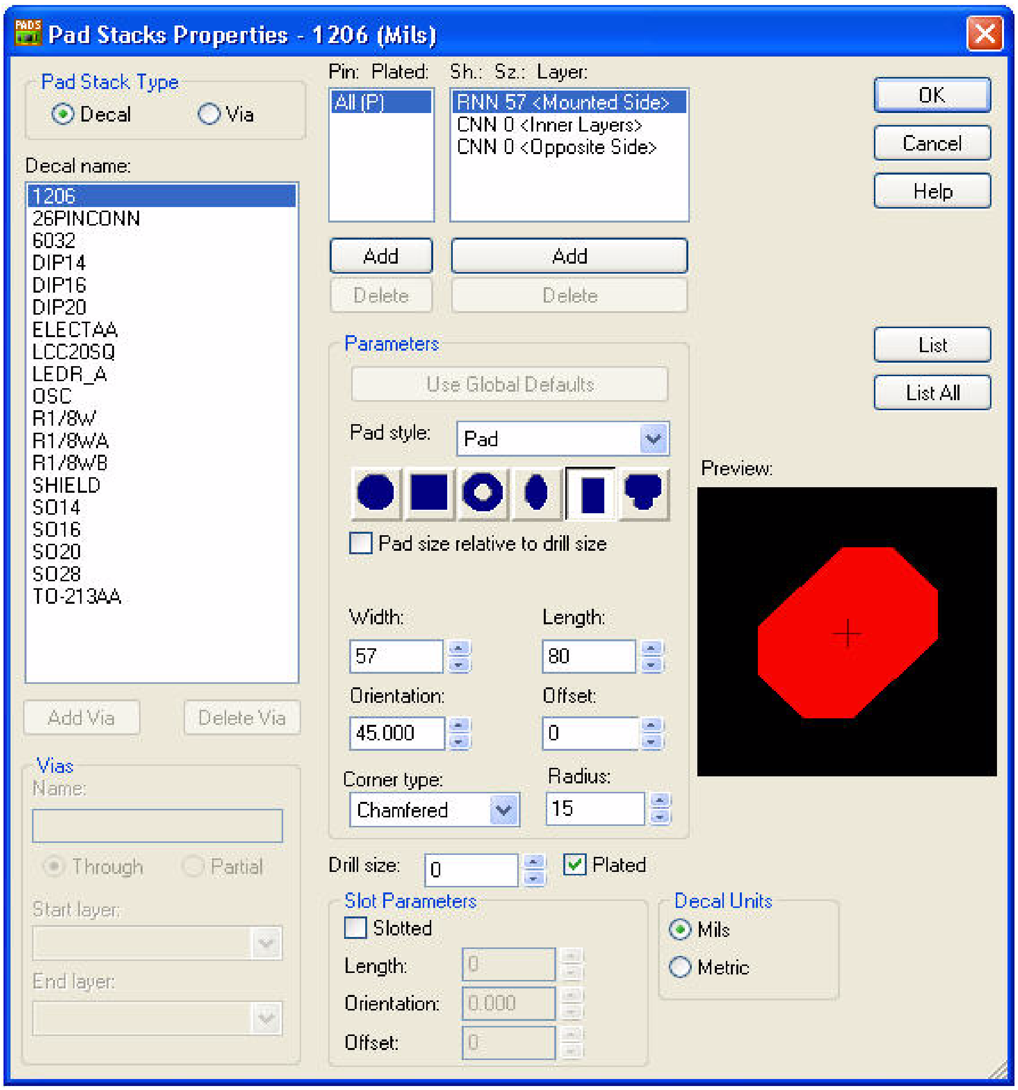


Figure 45-265. Pad Stacks Properties Dialog Box - Via Pad Stack

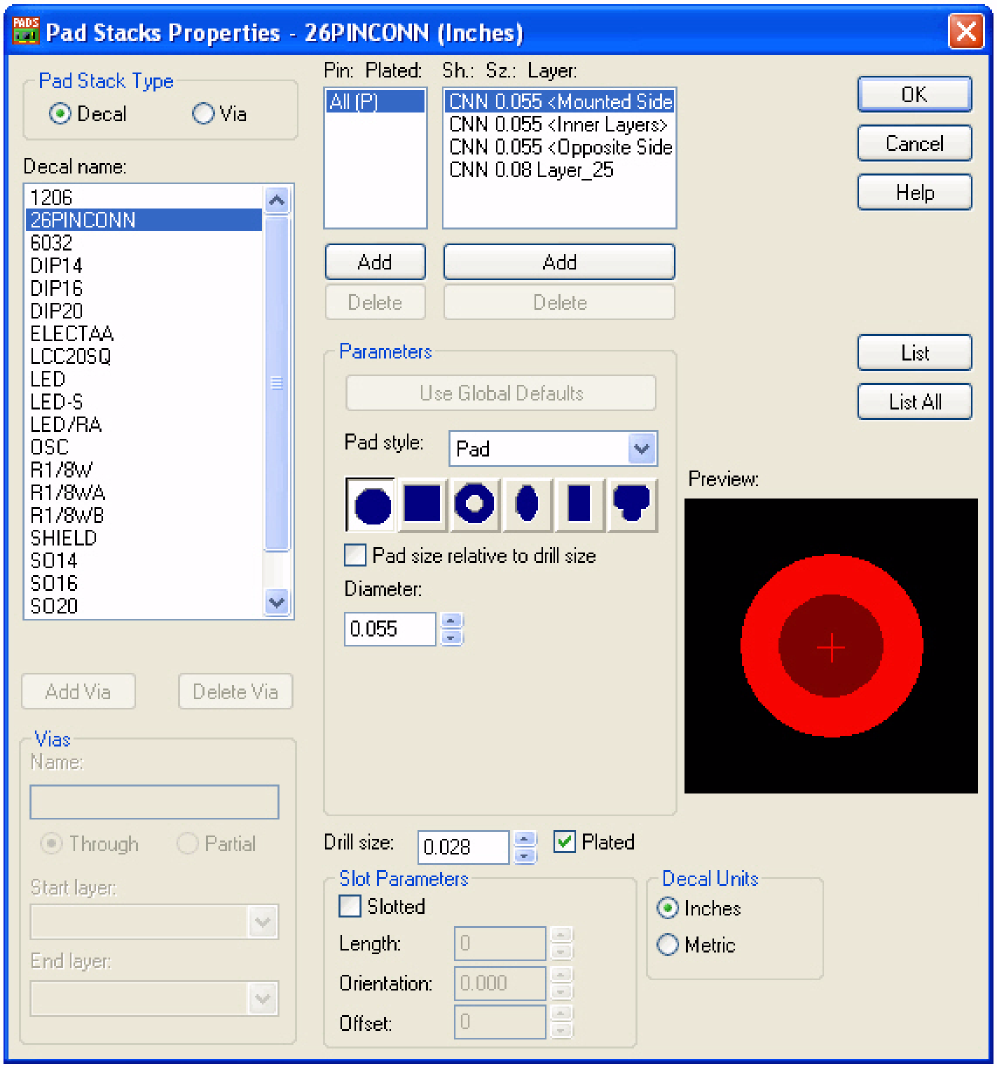


Figure 45-266. Pad Stacks Properties Dialog Box - Thermal Pad Style

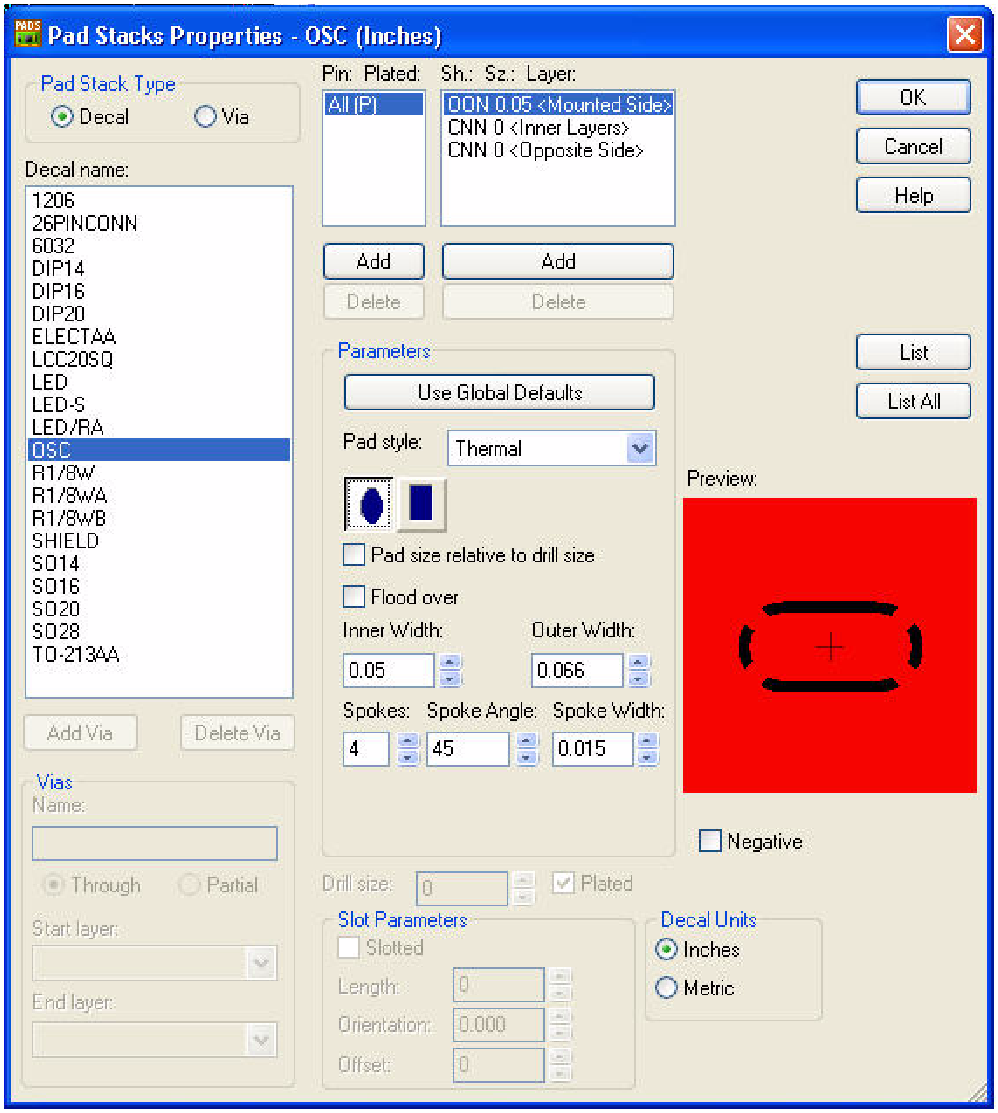


Table 45-243. Pad Stack Properties Dialog Box Contents

Name	Description
Pad Stack Type	**Decal**—Accesses component pad stack options. **Via**—Makes the Vias area of the dialog box available so you can define a via pad stack. Options related to decal or pin pad stacks are now unavailable. **See also:** Via Setup
Decal Name	Lists the available decal names for decals or vias, whichever is selected in the Pad Stack Type area.

Table 45-243. Pad Stack Properties Dialog Box Contents (cont.)

Name	Description
Add Via	Adds a via to the Decal Name list. **Restriction:** Available only when Via is selected in the Pad Stack Type area. **See also:** Creating a Through-hole Via, Creating a Partial Via
Delete Via	Deletes the selected via from the Decal Name list. **Restrictions:** • Available only when Via is selected in the Pad Stack Type area. • You cannot delete a via if it is already used in the design.
Vias Name	Assigns a name for a new via. **Restriction:** Available only when Via is selected in the Pad Stack Type area.
Through/Partial	**Through**—Sets the current via as a through via, through all layers of the board. **Partial**—Sets the current via as a partial via, through certain layers of the board. Click Partial and click a Start Layer and an End Layer. **Restriction:** Available only when Via is selected in the Pad Stack Type area.
Start Layer	Sets the start layer for a partial via. **Restriction:** Available only when Via is selected in the Pad Stack Type area, and Partial is the type of via.
End Layer	Sets the end layer for a partial via. **Restriction:** Available only when Via is selected in the Pad Stack Type area, and Partial is the type of via.
Pin Number and Plated	Specifies the pins to which to apply the pad stack edits. Use the Pin, Plated column to select the pin you want to edit. The Pin, Plated setup does not apply to via pad stacks; all vias are considered plated. You can select one or more pins and customize the pad stacks for your selection. **Restriction:** Available only when Decal is selected in the Pad Stack Type area. **Add**—Opens the Add Pin dialog box. **Delete**—Removes the selected Pin.

Table 45-243. Pad Stack Properties Dialog Box Contents (cont.)

Name	Description
Shape, Size, and Layer	Sets the size and shape of a pad on selected layers. Inner layers are drawn with a slightly larger pad size because they are often used for plane layers, where the pad size becomes an insulation area when output as a plane plot. **Add**—Opens the Add Layer dialog box. **Delete**—Removes the selected shape. In the shape column, the first character is the pad, the second is the thermal and the third is the antipad. “N” indicates that none has been defined. **See also:** Control of Solder Mask and Paste Mask
Use Global Defaults	Sets thermal and antipad shapes to those specified in the Thermals tab in the Options dialog box. **Restriction:** This option is available when the Pad Style list is set to Thermal or Antipad. **See also:** Design Rule Versus Pad Stack - Thermals and Antipads

Table 45-243. Pad Stack Properties Dialog Box Contents (cont.)

Name	Description
Pad Style	Specifies the style of pad: normal pad, thermal pad, or antipad. Thermal and Antipad pad styles control the size and shape of thermals and antipads used on split/mixed layers and CAM negative planes (for RS-274X output). **See also:** Design Rule Versus Pad Stack - Thermals and Antipads **Tips:** • Beginning with PADS 9.2, the size, shape and orientation of thermals for slotted pads are no longer derived from the length, drill size and orientation of the slotted hole, but are inherited from the normal pad. • Set the Inner Diam and Outer Diam values to be the same for a solid connection to the plane (flood over). The current pad diameter is used as the inner diameter with the outer diameter set at the default same-net pad to corner rule. For antipads, diameters are initially set to follow the current default pad to copper design rule. If you select Use Design Rules for Thermals and Antipads in the Split/Mixed Plane page of the Options dialog box, the outer diameter is ignored and the clearance rule is used instead, except when the outer diameter is less than the inner diameter. Inner and outer diameter options always, however, control flood over. The length of an antipad with a non-zero drill size equals the slot length minus the drill size, plus the width. **Restriction:** You cannot create antipads using the Mounted Side or Opposite side layers. You must add the actual layer to the list. When you select a Mounted or Opposite Side layer in the *Sh: Sz: Layer:* list, Antipad is unavailable.

Table 45-243. Pad Stack Properties Dialog Box Contents (cont.)

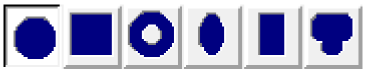
Name	Description
Shape buttons 	Assigns a pad shape to the layer selected in the *Sh: Sz: Layers:* list. You can assign pad shapes as round, square, annular, oval or rectangular, odd. • **Annular** lets you specify an inner pad diameter, bringing the inner diameter of the pad inside the drill outline. A pilot dimple at the center of the copper pad appears as an aid to hand fabrication of some prototypes. • Odd is useful for drawn items such as moiré pads or registration marks. It is a circular outline only, like an antipad, and is usually used for marking areas for airgap checking. This setting should not be confused with trying to create a custom shaped (odd shaped) pad. See Creating a Custom Pad Shape for more information.
Pad size relative to drill size	Displays inner and outer pad sizes relative to the drill size.
Flood over	Specifies that the pad requires no thermal relief and should be flooded over, irrespective of any default flooding options for a normal pad of this shape. **Restriction:** Available only when **Pad style** is Thermal and a pad shape button is selected.

Table 45-243. Pad Stack Properties Dialog Box Contents (cont.)

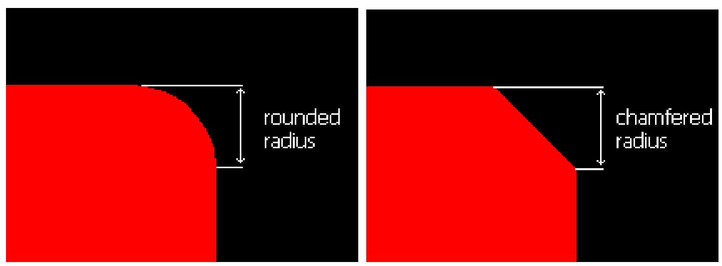
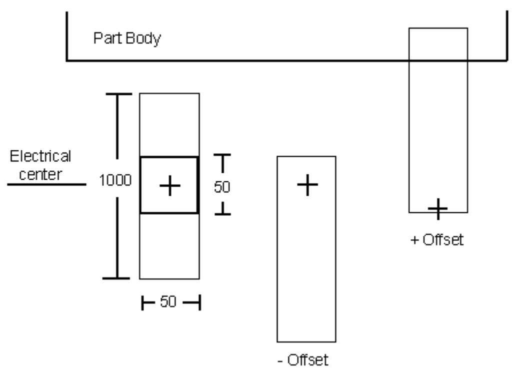
Name	Description
Parameters area	The options for sizing the pad shape vary according to the shape you choose. **Round**—Diameter (if hole is not slotted), Width (if hole is slotted). If the pad style is thermal: Inner Diam, Outer Diam, Spokes, Spoke Angle, Spoke Width. **Square**—Size (if hole is not slotted), Width (if hole is slotted), Corner type and Radius. If the pad style is thermal: Inner Size, Outer Size, Spokes, Spoke Angle, Spoke Width. **Annular**—Diameter and Inner Diameter **Oval**—Width, Length, Orientation, and Offset **Rectangular**—Width, Length, Orientation, Offset, Corner type and Radius **Odd**—Diameter **Figure 45-267. Radius Examples** 

Table 45-243. Pad Stack Properties Dialog Box Contents (cont.)

<table>
<tr><th>Name</th><th>Description</th></tr>
<tr><td>Orientation/Offset</td><td>Sets the orientation and offset for pads. Used for SMD decals and edge finger connectors. Available when you click the oval or rectangular shaped button.
For orientation, 90 degrees is perpendicular to the part body and 0 is parallel to the part body. You can type any degree of rotation.
By specifying the offset, you can move the rectangular or oval pad slightly off its electrical center for two purposes:
• Pads that are concentric on the display, as they are in edge fingers, are difficult to select and work with during routing. Offsetting one of the layers makes it easier to identify the top and bottom layers.
• To extend the pad out from the component body, leaving the electrical centers unmoved.
The maximum amount of offset you can assign is one half of the pad's length. If you exceed this limit, the pad fails to appear in the Preview dialog box for CAM.

Figure 45-268. Offset Example


Maximum offset = ½ length or (1000/2) = 500</td></tr>
<tr><td>Drill size</td><td>Specifies the hole size. If the hole is plated, this value remains the same. It is the finished hole size.
Drill holes are normally oversized by the fabricator in order to achieve the specified drill size value after the hole is plated.
See the Drill Oversize setting in the Options Dialog Box, Design page.</td></tr>
</table>

Table 45-243. Pad Stack Properties Dialog Box Contents (cont.)

Name	Description
Plated	Sets whether the drill hole is plated with copper. Normally, pad stacks with a hole are plated. To create a nonplated hole with no copper, such as a mounting hole, clear this Plated check box. Nonplated holes are drilled to true drill diameter, without oversize, and any drill oversize does not apply to them. **Tips:** • The batch clearance checking functions consider the added (plated hole) drill oversize value when flagging errors. See the Drill Oversize setting in the Options Dialog Box, Design page. • If the board is marked as Single-sided in the Layers Setup dialog box, this check box is ignored.
Slotted	Enables a slotted hole for the selected pad stack or pin. **See also:** Using Slotted Holes
Slot Length	Sets the length of the slotted hole. This option becomes available when you select Slotted.
Slot Orientation	Sets the orientation of the slotted hole. This option becomes available when you select Slotted. **Tip:** A custom thermal or antipad for a slotted hole has the same orientation as the slot.
Slot Offset	Sets the slotted hole offset. Slot offset moves the center of the slotted hole relative to the electrical center of the pin–always in the opposite direction of the pad offset. This option becomes available when you select Slotted. The maximum amount of offset you can assign is one half of the slot's length. If you exceed this limit, the slotted hole display is suppressed in the Preview dialog box in CAM. **Tip:** A custom thermal or antipad for a slotted hole has the same offset as the slot.
Decal Units	Shows and sets the current units of the selected decal. **Restriction:** This area is unavailable if you select the Via in the Pad Stack Type area or when you use this dialog box in the PCB Decal Editor. **Tip:** Switching the decal units can cause errors if rounding off occurs.
List	Produces a report describing the selected pad stack. Pad stack reports show pad stack, slotted hole, and unit information. Vias are output in design units.

Table 45-243. Pad Stack Properties Dialog Box Contents (cont.)

Name	Description
List All	Produces a report describing all pad stacks in all decals in the database. Pad stack reports show pad stack, slotted hole, and unit information. Vias are output in design units.
Preview	Shows pad shape and size for the current options.
Negative	Changes the preview area to a negative view. **Restriction:** Thermal Pad Style only.

Related Topics

Customizing Pad Stacks of Decal Pins

Flooding Over Pads in a Copper Pour or Plane Area

Flooding Over Vias in a Copper Pour or Plane Area

Pad Stack Report

Pad Stacks

Slotted Hole Offset Versus Pad Offset

Pad Stack Properties for Pin Dialog Box

Use the Pad Stack Properties for Pin dialog box to modify the size and shape of one or more selected terminals or slotted holes while in the PCB Decal Editor.

Accessing

- **PCB Decal Editor** > **select a terminal** > right-click > **Pad Stacks**

Figure 45-269. Pad Stack Properties for Pin Dialog Box

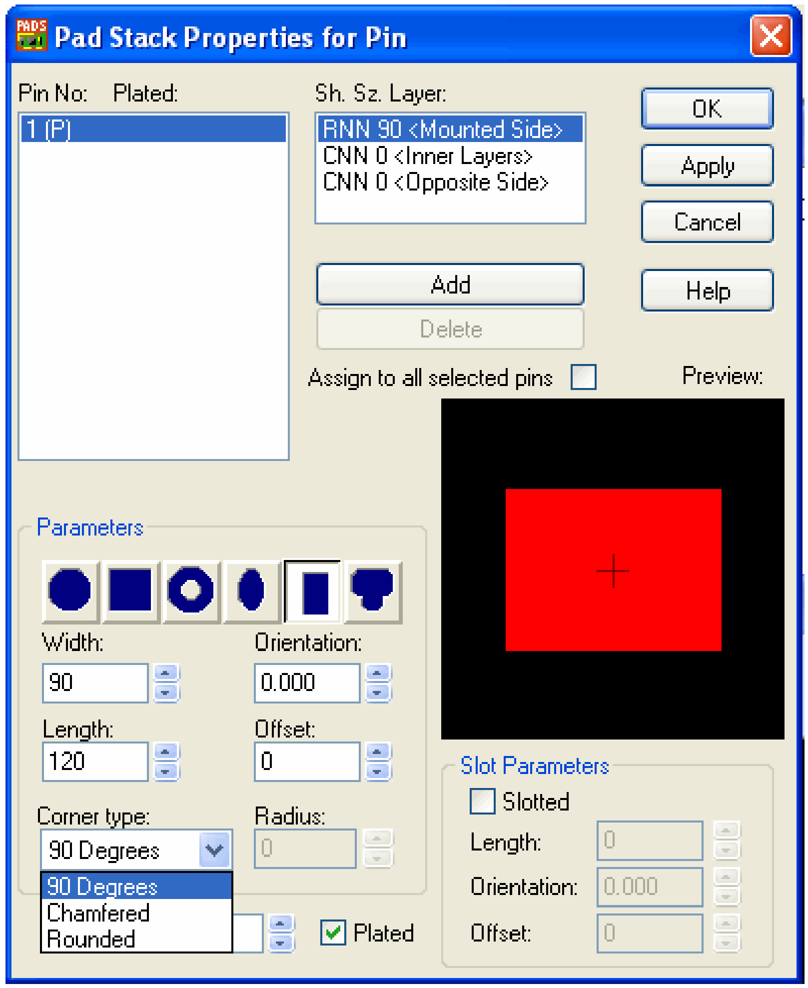


Table 45-244. Pad Stack Properties for Pin Dialog Box Contents

Name	Description
Pin Number and Plated	For Decal pad stacks, you need to specify the pins to which to apply the pad stack edits. Use the Pin, Plated column to select the pin you want to edit. The Pin, Plated setup does not apply to via pad stacks; all vias are considered plated.
Shape, Size, and Layer	For both via and component pad stacks, you can set the size and shape of a pad on selected layers. Inner layers are drawn with a slightly larger pad size because they are often used for plane layers, where the pad size becomes an insulation area when output as a plane plot. **See also:** Pad Stacks

Table 45-244. Pad Stack Properties for Pin Dialog Box Contents (cont.)

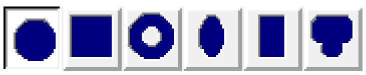
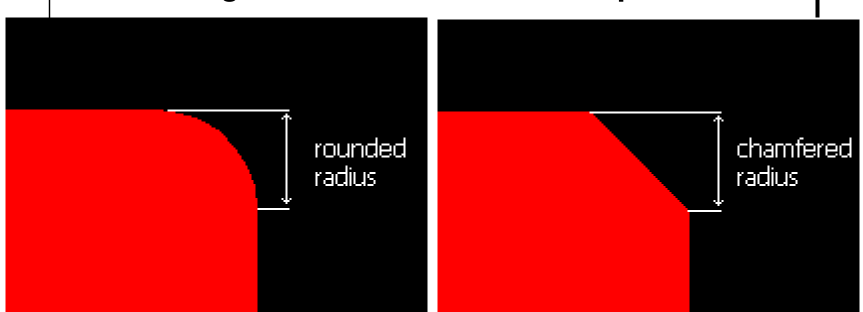
Name	Description
Add	Opens the Add Layer dialog box.
Delete	Removes the selected shape
Assign to all selected pins	Modifies the terminals you select in the Pin Name, Plated list to match the terminal selected in the workspace.
Shape buttons 	Assigns a pad shape to the layer selected in the Size, Shape, Layers list. You can set pads as round, square, annular, oval or rectangular, odd. • Annular lets you specify an inner pad diameter, bringing the inner diameter of the pad inside the drill outline. A pilot dimple at the center of the copper pad appears as an aid to hand fabrication of some prototypes. • Odd is useful for drawn items such as moiré pads or registration marks. It is a circular outline only, and is usually used for marking areas for airgap checking. This setting should not be confused with trying to create a custom shaped (odd shaped) pad. See Creating a Custom Pad Shape for more information.
Parameters area	The options for sizing the pad shape vary according to the shape you choose. **Round**—Diameter (if hole is not slotted), Width (if hole is slotted) **Square**—Size (if hole is not slotted), Width (if hole is slotted), Corner type and Radius **Annular**—Diameter and Inner Diameter **Oval**—Width, Length, Orientation, and Offset **Rectangular**—Width, Length, Orientation, Offset, Corner Type and Radius **Odd**—Diameter **Figure 45-270. Radius Examples** 

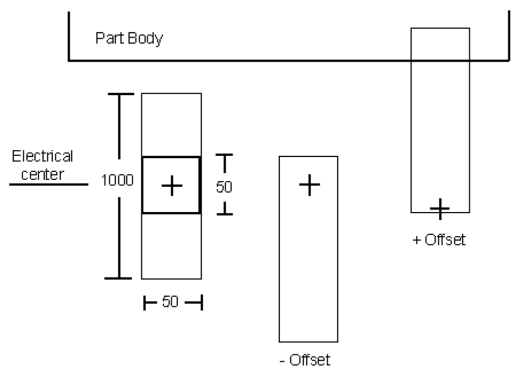
Table 45-244. Pad Stack Properties for Pin Dialog Box Contents (cont.)

<table>
<tr><th>Name</th><th>Description</th></tr>
<tr><td>Orientation/Offset</td><td>
Sets the orientation and offset for pads. Used for SMD decals and edge finger connectors. Available when you click the oval or rectangular shaped button.
For orientation, 90 degrees is perpendicular to the part body and 0 is parallel to the part body. You can type any degree of rotation.
By specifying the offset, you can move the rectangular or oval pad slightly off its electrical center for two purposes:

- Pads that are concentric on the display, as they are in edge fingers, are difficult to select and work with during routing. Offsetting one of the layers makes it easier to identify the top and bottom layers.
- To extend the pad out from the component body, leaving the electrical centers unmoved.

The maximum amount of offset you can assign is one half of the pad's length. If you exceed this limit, the pad fails to appear in the Preview dialog box for CAM.

Figure 45-271. Offset Example



Maximum offset = ½ length or (1000/2) = 500
</td></tr>
<tr><td>Drill size</td><td>Sets the drill size of the pad.</td></tr>
</table>

Table 45-244. Pad Stack Properties for Pin Dialog Box Contents (cont.)

Name	Description
Plated	Sets whether the pad is plated with copper. Normally, pads with a hole are plated. To create a nonplated hole with no copper, such as a mounting hole, click to clear Plated. Nonplated holes are drilled to true drill diameter, without oversize, and any drill oversize does not apply to them. **Tip:** The batch clearance checking functions consider the added drill oversize value when flagging errors. When Plating is cleared, batch clearance checking applies to the true drill value.
Slotted	Enables a slotted hole for the selected pad stack or pin. **See also:** Using Slotted Holes
Slot Length	Sets the length of the slotted hole. This option becomes available when you select Slotted.
Slot Orientation	Sets the orientation of the slotted hole. This option becomes available when you select Slotted. **Tip:** A custom thermal or antipad for a slotted hole has the same orientation as the slot.
Slot Offset	Sets the slotted hole offset. Slot offset moves the center of the slotted hole relative to the electrical center of the pin–always in the opposite direction of the pad offset. This option becomes available when you select Slotted. The maximum amount of offset you can assign is one half of the slot's length. If you exceed this limit, the slotted hole display is suppressed in the Preview dialog box in CAM. **Tip:** A custom thermal or antipad for a slotted hole has the same offset as the slot.
Preview	Shows pad shape and size for the current options.

Related Topics

Customizing Pad Stacks of Decal Pins

Pad Stacks

Pads for Die Pin Dialog Box

Use the Pads for Die Pin dialog box to specify which label to use for the die part's substrate bond pad.

Restriction: This information applies to only the BGA toolkit.

Accessing

- **BGA Toolbar** button > **Select a BGA Pin** > double-click in a BGA Pad cell > **Ellipsis** button

Figure 45-272. Pads for Die Pin Dialog Box

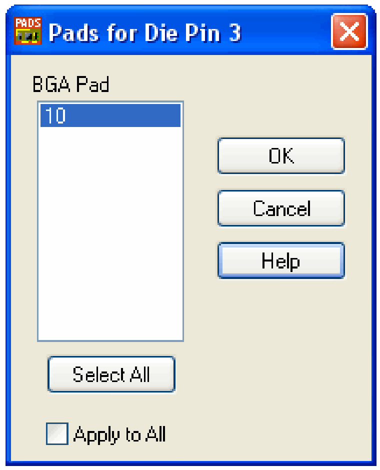


Table 45-245. Pads for Die Pin Dialog Box contents

Name	Description
BGA Pad	Lists all BGA pin pad labels that are connected to the die part pin selected in the Add BGA Pin Labels dialog box. Select the pin pad labels to use in the substrate bond pad's label.
Select All button	Selects all BGA pin pad labels in the BGA Pad list.
Apply to All	Applies the current die pin number and BGA pad setup to all die pin numbers that have multiple BGA pin pads.

Related Topics

To Display Die Pins

To List All BGA Pin Labels

To List Specific BGA Pin Labels

PADS Router Link Dialog Box

The depends on your Synchronization mode settings. When you are not in Synchronization mode, the PADS Router Link sets up autorouting options for PADS Router and passes information from PADS Layout to PADS Router. Data is stored in the design file. Using the link you can either run PADS Router and automatically open the current design file in the foreground, so you can view the autorouter's progress, or you can run PADS Router in the background. When you are in Synchronization mode, the PADS Router link sets up autorouting options for PADS Router and switch to PADS Router. Using the link you can either run PADS Router and automatically open the current design file in the foreground, so you can view the autorouter's progress, or you can run PADS Router in the background.

Accessing

- **Tools** menu > **PADS Router**

Figure 45-273. PADS Router Link Dialog Box

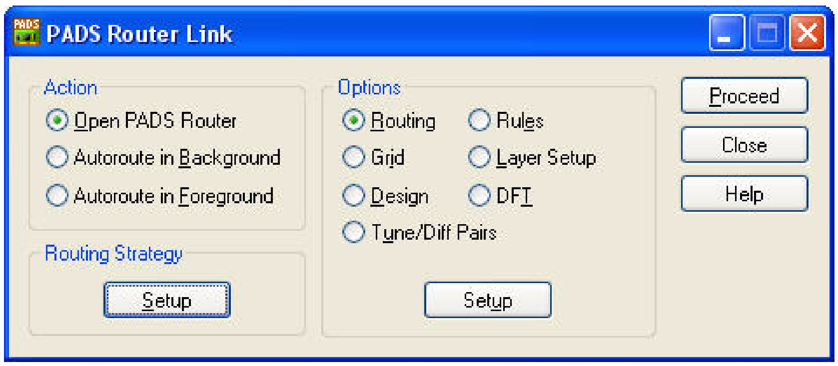


Figure 45-274. PADS Router Link Dialog Box - Synchronization Mode

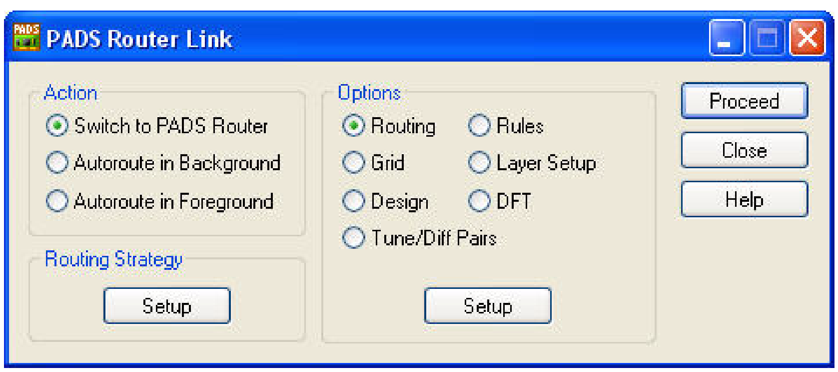


Table 45-246. PADS Router Link Dialog Box contents

Name	Description
Action	Specifies the action you want to perform. The options are: • **Open PADS Router**—Opens PADS Router and loads the design file into it. You can make changes to the design file in PADS Router (rather than PADS Layout) before routing. **Restriction:** Non-Synchronization mode only. • **Switch to PADS Router**—Links PADS Router and PADS Layout. Any changes you make to the design file in PADS Router are reflected in PADS Layout. **Restriction:** Synchronization mode only. • **Autoroute in Background**—Opens PADS Router and PADS Router Monitor, but runs PADS Router in the background. Layout commands are disabled and a wait cursor shown until autorouting is completed or the Stop button is selected in the Router Monitor. • **Autoroute in Foreground**—If you are not in Synchronization mode, opens PADS Router and PADS Router Monitor, and runs PADS Router in the foreground making it the active program. If you are in Synchronization mode, opens PADS Router and runs it in the foreground making it the active program.
Setup button	Opens the Routing Strategy Dialog Box.
Options	Specifies the Options dialog box tab you want to open when you click the Setup button.
Setup button	Opens the Options dialog box to the tab you specified above.
Proceed	Autoroutes the design with the settings you specified.

Related Topics

Autorouting Your Design

Synchronization Mode

PADS Router Monitor Dialog Box, Routing Tab

PADS Router Monitor provides dynamic information as the PADS Layout design is translated for autorouting.

Accessing

The Routing tab on the PADS Router Monitor dialog box appears when you click Proceed on the PADS Router Link dialog box.

Tip: If PADS Router Monitor does not appear on the desktop, look for it on the Windows taskbar.

Figure 45-275. Routing Tab

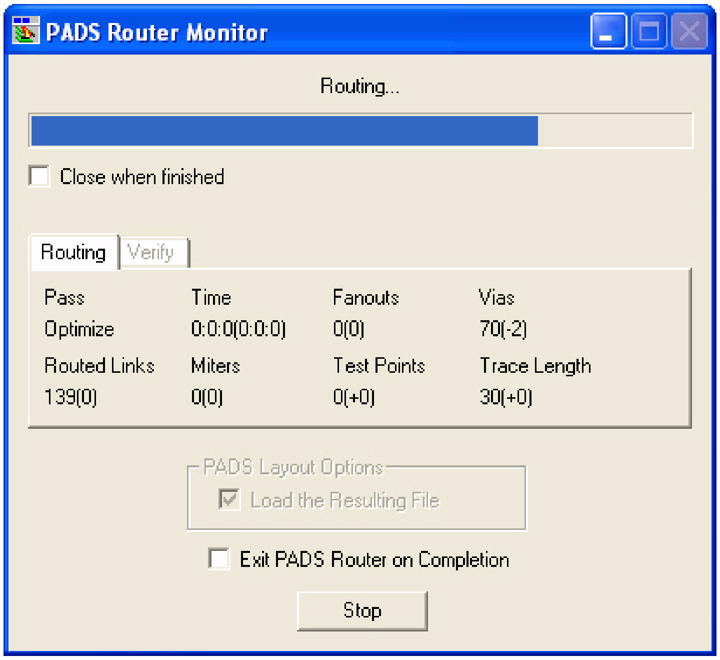


Table 45-247. Routing tab contents

Name	Description
Routing Progress Indicator	Displays the autorouting progress.
Close When Finished	Specifies to close PADS Router Monitor when autorouting completes.

Table 45-247. Routing tab contents (cont.)

Name	Description
Routing tab	Displays the following information about the current routing pass: • **Pass**—The name of the current routing pass. • **Time**—The total time spent in previous passes followed by the elapsed time for the current pass. • **Fanouts**—The total number of fanouts created in previous passes followed by the number of fanouts created during the current pass. • **Vias**—The total number of vias routed in previous passes followed by the number of vias routed during the current pass. • **Routed Links**—The total number of successfully routed links in previous passes followed by the number of links routed during the current pass. • **Miters**—The total number of miters created in previous passes followed by the number of miters created during the current pass. • **Test Points**—The total number of test points created in previous passes followed by the number of test points created during the current pass. • **Trace Length**—The total length of all successfully routed traces in previous passes followed by the total length of all traces routed during the current pass.
Load the Resulting File	Specifies to open the routed design in a new instance of PADS Layout when autorouting completes. **Restriction:** This option is available only when you are routing in the foreground. When you are routing in the background, this check box is checked and unavailable, and the routed file is automatically opened in the current Layout window.
Exit PADS Router on Completion	Specifies to close PADS Router automatically if Close When Finished is also checked. If Exit PADS Router on Completion is checked, and Close when Finished is cleared, both the Monitor and PADS Router will remain open until you click the Monitor Stop button.
Stop button	Stops the autorouting process and closes the PADS Router Monitor. Also closes PADS Router if you clicked Exit PADS Router on Completion.

PADS Router Monitor Dialog Box, Verify Tab

PADS Router Monitor provides dynamic information as the PADS Layout design is translated for autorouting.

Accessing

The Verify tab on the PADS Router Monitor dialog box appears when you click Start on the Verify Design dialog box.

Tip: If PADS Router Monitor does not appear on the desktop, look for it on the Windows taskbar.

Figure 45-276. Verify Tab

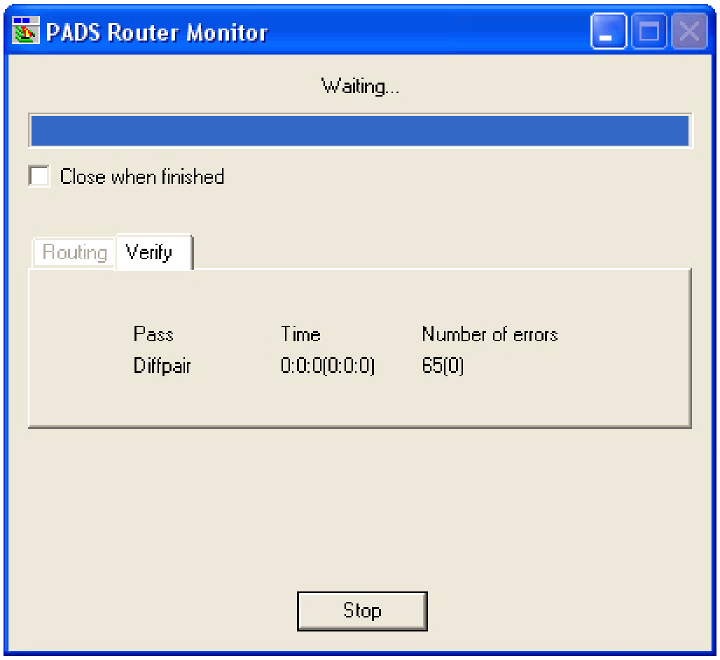


Table 45-248. Verify tab contents

Name	Description
Routing Progress Indicator	Displays the autorouting progress.
Close When Finished	Specifies to close PADS Router Monitor when autorouting completes.
Verify tab	Displays the following information about the current routing pass: • **Pass**—The name of the current routing pass. • **Time**—The total time spent in previous passes followed by the elapsed time for the current pass. • **Errors**—The total number of errors in the previous passes followed by the number of errors during the current pass.

Table 45-248. Verify tab contents (cont.)

Name	Description
Stop button	Stops the autorouting process and closes the PADS Router Monitor. Also closes PADS Router if you clicked Exit PADS Router on Completion.

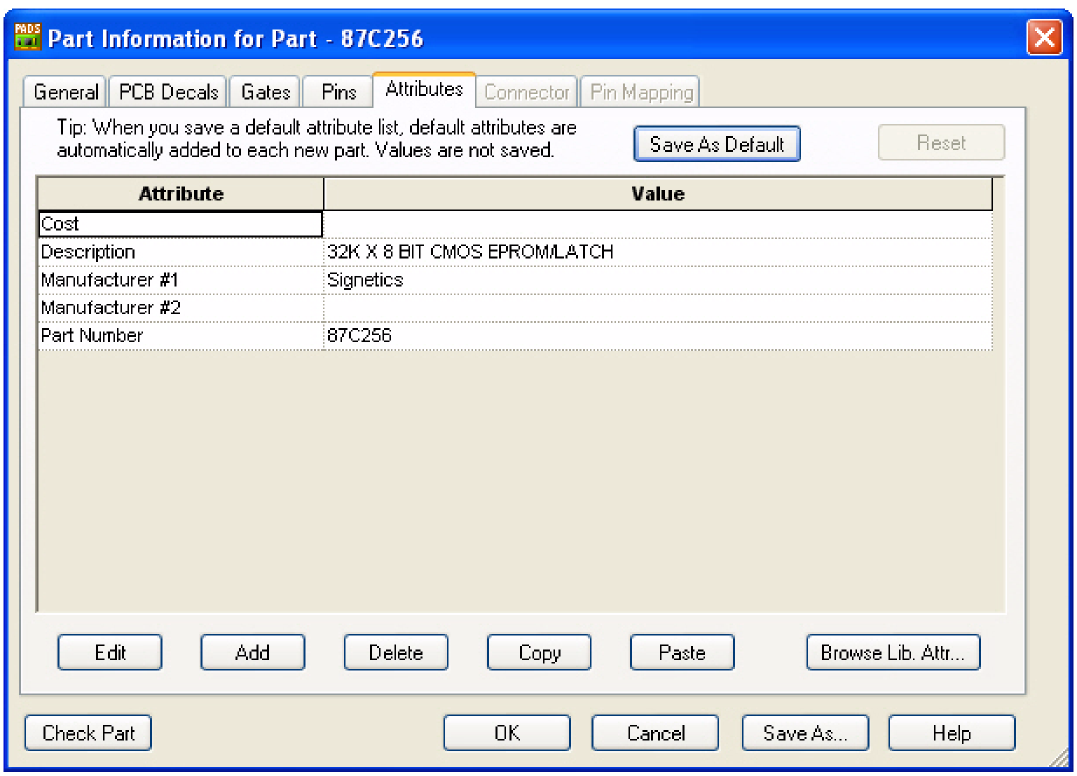
Part Information Dialog Box, Attributes Tab

Use the Attributes tab in the Part Information dialog box to manage attributes for the selected part, and to define default attributes for new parts.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **Attributes** tab

Figure 45-277. Attributes Tab



PADS Suite Configuration Dialog Box

Use the PADS Suite Configuration dialog box to manage PADS Suite (composite) licenses.

Accessing

- **Help** menu > **Installed Options** > **Suite Configuration** button

  Or

- **Optional:** Opens on program startup

Figure 45-278. PADS Suite Configuration dialog box

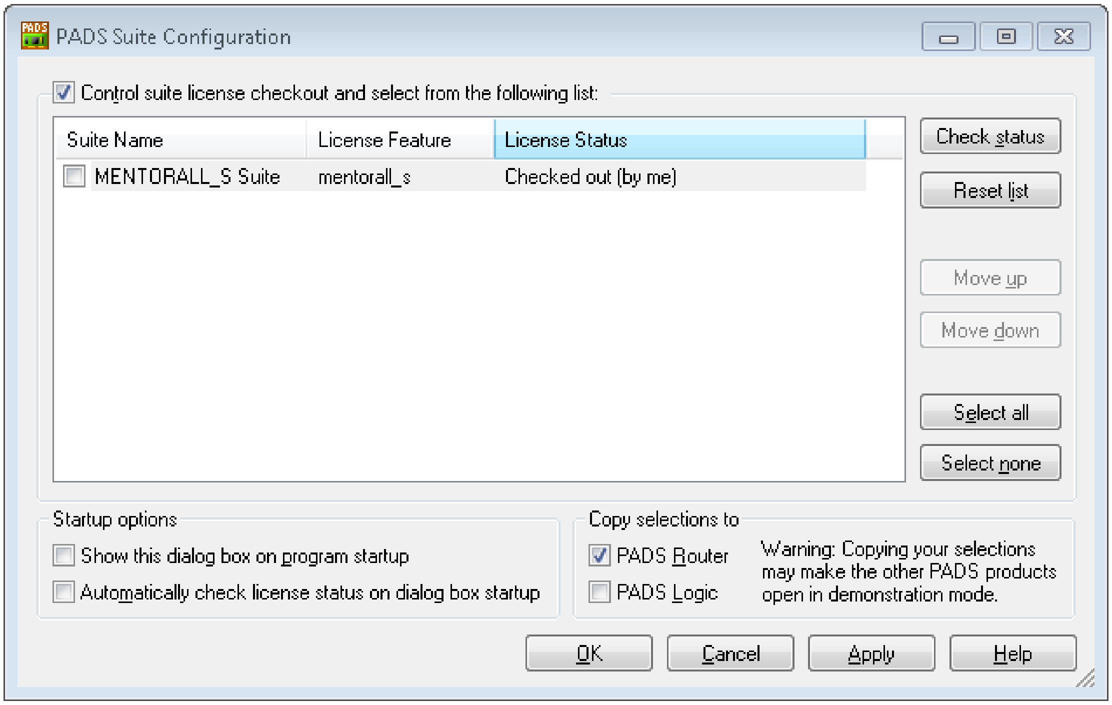


Table 45-249. PADS Suite Configuration dialog box Contents

Name	Description
Control suite license checkout and select from the following list	Enables the Suite License table for you to control checkouts.
Suite Name column	Lists the name of the suite for which the license works.
License Feature column	Lists the specific features available for each license.
License status column	Lists the status of the licenses when you click the Check status button.
Check Status button	Specifies to check the status of all licenses listed in the table and displays the status in the License status column.
Reset List button	Specifies to reset the list of suite licenses to only those detected in your licensing environment.

Table 45-249. PADS Suite Configuration dialog box Contents (cont.)

Name	Description
Move up button	Moves the selected license up one row.
Move down button	Moves the selected license down one row.
Select all button	Selects all of the listed licenses.
Select none button	Deselects all of the listed licenses.
Show this dialog box on program startup	Specifies to open the PADS Suite Configuration dialog box when PADS Layout starts.
Automatically check license status on dialog box startup	Specifies to check the status of the licenses when you open the PADS Suite Configuration dialog box.
Copy Selections to area	Specifies to copy what you've done to PADS Router and/or PADS Logic. Warning: Copying your selections may make the other PADS products open in demonstration mode.

Related Topics

Installed Options Dialog Box

Checking Out Suite Licenses

Part Information Dialog Box, Connector Tab

Use the Connector tab in the Part Information dialog box to assign one or more pin decals, or Special Symbols, to a connector. This allows you to use any of the special symbols as alternate pins of the connector. Instead of being displayed as a single symbol, the connector is broken up into individual pin symbols in the schematic. You can place the pins all together, or wherever you like on a single page or even across multiple pages.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **Connector** tab

  Restriction: This tab is unavailable when the Connector check box on the General tab is cleared, or when a gate has been assigned to the part on the Gates tab.

Figure 45-279. Connector Tab

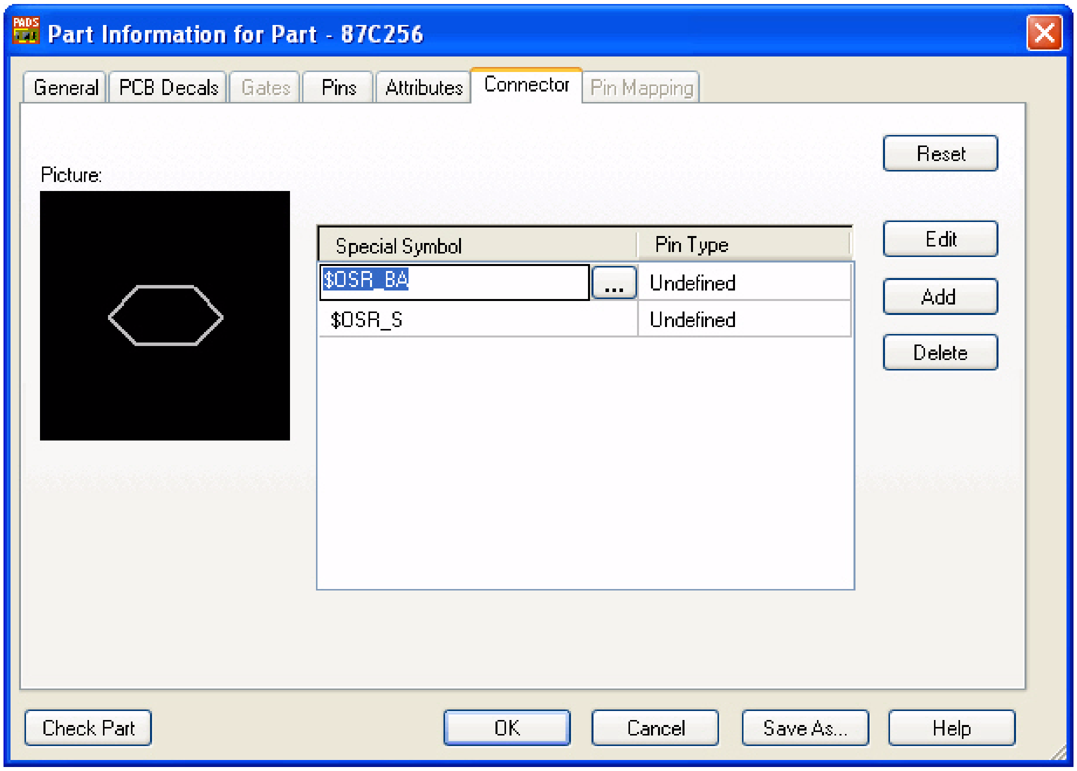


Table 45-250. Connector Tab Contents

Name	Description
Picture	Displays a picture of the selected Special Symbol.
Attribute table	• **Special Symbol**—The name of a connector pin decal for use in the schematic. • **Pin Type**—The function of the special symbol.
...	Opens the Browse for Special Symbols Dialog Box where you can browse for a pin decal.
Reset	Undoes all changes you made in the Connector tab.
Edit	Makes the selected cell available for editing. **Tip:** You can also double-click the cell to edit the contents.
Add	Adds a new row at the bottom of the table.
Delete	Removes the selected row.

Table 45-250. Connector Tab Contents (cont.)

Name	Description
Check Part	Checks for missing or inconsistent information. **Tip:** Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.
Save As	Opens the Save Part Type to Library dialog box.

Related Topics

Creating a Connector Part Type

Creating a New Part Type

Part Information Dialog Box, Gates Tab

Use the Gates tab in the Part Information dialog box to assign gate information, such as CAE decals and gate swap options to a part.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **Gates** tab

Figure 45-280. Gates Tab

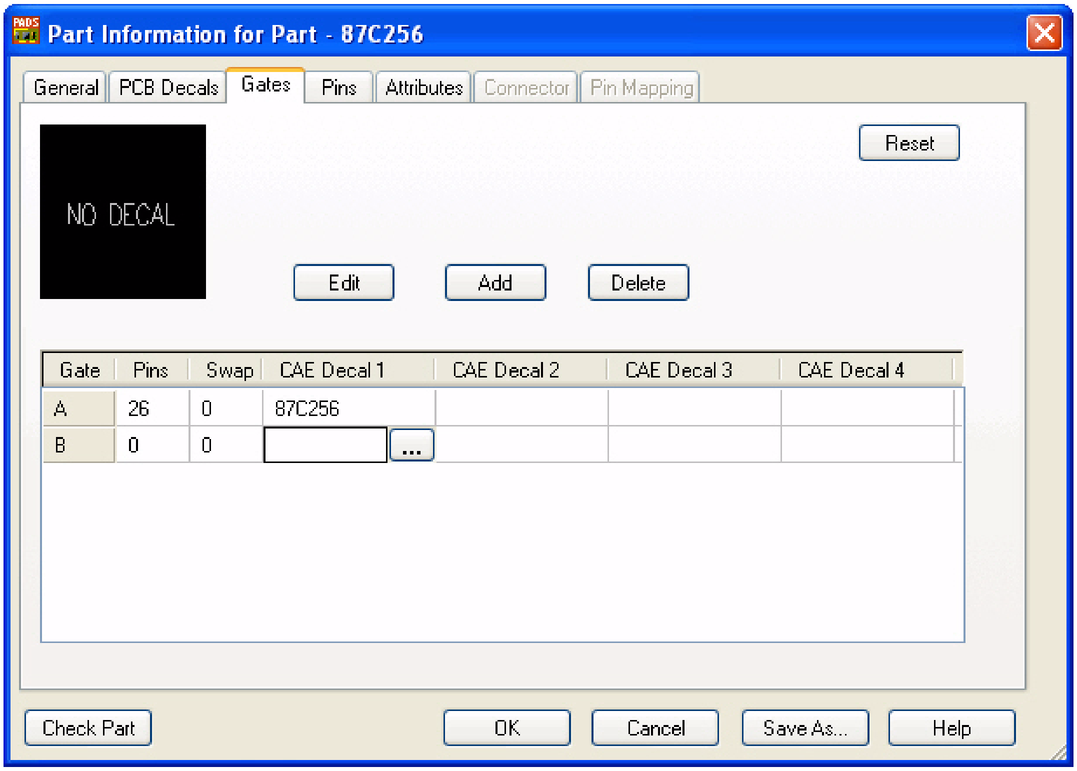


Table 45-251. Gates Tab Contents

Name	Description
Preview area	Shows the item selected in the Decal cell.
Reset	Undoes all changes you made in the Gates tab.
Edit	Makes the selected cell available for editing. Also displays the Browse button.
	Opens the Assign Decal to Gate Dialog Box. **Tip:** This button is available only in the CAE Decal columns, and only when the cell is available for editing.
Add	Adds a new row with the next Gate letter at the bottom of the Gate table.
Delete	Removes the selected row from the Gate table.

Table 45-251. Gates Tab Contents (cont.)

Name	Description
Gate table	• **Gate column**—Displays the letter of the gate. • **Pins column**—Displays the number of pins for the gate. Gate pins are added on the Pins tab. • **Swap column**—Displays the swap ID from 0 to 100. To uncross connections and facilitate routing, gates with the same swap ID (except for 0) can be swapped within a part or with another part of the same type. **Tip:** Type 0 to disable swapping. • **CAE Decal *N* column**—Displays the CAE Decal name. The decal listed for CAE Decal 1 is the default decal and is used when you add the part to the schematic. Additional decals are alternates. You can assign up to four CAE decals to a part. **Tip:** Double-click to Type a decal name or click the "..." (Browse) button to search for a decal from a library
Check Part	Checks for missing or inconsistent information. **Tip:** Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.
Save As	Opens the Save Part Type to Library dialog box.

Related Topics

Creating a New Part Type

Part Information Dialog Box, General Tab

Use the General tab in the Part Information dialog box to specify pin count, logic family, and various options for a part.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button

Figure 45-281. General Tab

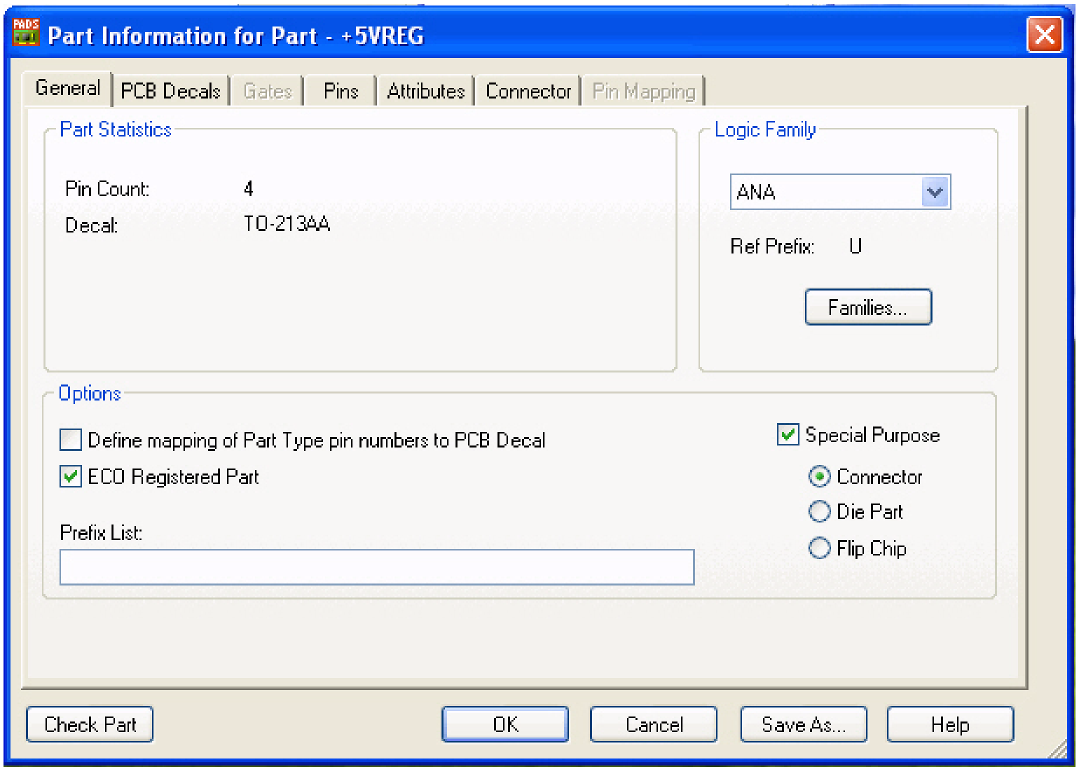


Table 45-252. General Tab Contents

Name	Description
Part Statistics	All the statistics in the Part Statistics area are read-only. Pin Count lists the total number of pins - gate pins, signal pins, and unused pins. If multiple decals are assigned with different pin counts, a range of smallest to largest decal pin counts is shown.
Logic Family list	Specifies the Logic Family (reference designator prefix) to use for the part. You can also create a new logic family or edit the existing reference designator prefix designations by clicking the families button. **Note:** Beginning with PADS 9.0, die parts and flip chips are no longer identified by their family designations (DIE or FLP), but instead by the Special Purpose settings on this tab. With this change, you can assign any reference designator (logic family) to a die part or flip chip without losing the special properties of these parts (such as the ability to move the part's substrate bond pads in the design).

Table 45-252. General Tab Contents (cont.)

Name	Description
Ref Prefix	Displays the prefix for the selected Logic family.
Families	Opens the Logic Families Dialog Box, where you can add, edit, or delete a logic family.
Define mapping of Part Type pin numbers to PCB Decal	Activates the Pin Mapping tab, where you map the numerical physical pins in the decal to the alphanumeric logical pins in the part type. **Restrictions:** • The check box is unavailable once you add one or more alphanumeric decals to the part type. Remove the assigned alphanumeric decals to make the check box available. • You must assign a numeric decal to use the Pin Mapping tab. • Only decals with sequential numerical pin numbers can be used with pin mapping.
Special Purpose	Check this box, then select the appropriate radio button: **Connector**—Select to identify the part as a connector and make the Connector tab available. Connectors do not require a prefix list or gate definitions. **Restrictions:** • This check box is automatically selected when you create or modify connectors. It is unavailable if you open a part other than a connector. • The Gate Decals tab is unavailable when the Connector check box is selected. • Some Pins tab controls not applicable to connector parts are disabled. **Die Part**—Select to identify the part as a die part. This also allows you to move the substrate bond pads in the design. **Flip Chip**—Select Flip Chip to identify the part as a flip chip. This also allows you to move the substrate bond pads in the design. **Note:** Beginning with PADS 9.0, die parts and flip chips are identified by these Special Purpose settings instead of by their family designation (DIE or FLP). With this change, you can assign any reference designator (logic family) to a die part or flip chip.

Table 45-252. General Tab Contents (cont.)

Name	Description
ECO registered part	Identifies the part as eligible for transfer between the design file and the schematic file during forward annotation or backward annotation. You can override this setting when creating the ECO file. You can specify it when generating an .eco file using either the ECO Toolbar, or the Compare/ECO dialog box. **Tip:** Typically you do not select this check box for non-electrical parts. For example, if you create a mounting hole to add to your design, you would not need the part (mounting hole) to pass back to the schematic software when you perform a backward annotation of the design.
Prefix list	Applies the part information to other parts in the library. Use the prefix with wildcards to identify the parts you want. **Examples:** • Question mark ? in a prefix acts as a wildcard for one character. The prefix “?4” is the equivalent of “54” or “74”. • If you type “\02” as the suffix, the edits are applied to all parts ending in 02. **Warning:** The contents of the Prefix List box are applied when you click OK or Save As on other tabs in the Part Information dialog box.
Check Part	Checks for missing or inconsistent information. **Tip:** Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.
Save As	Opens the Save Part Type to Library dialog box.

Related Topics

Creating a New Part Type

Part Information Dialog Box, PCB Decals Tab

Use the PCB Decals tab in the Part Information dialog box to specify the decal, or footprint, for a part. The decal determines the number of pins in the part.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **PCB Decals** tab

Figure 45-282. PCB Decals Tab

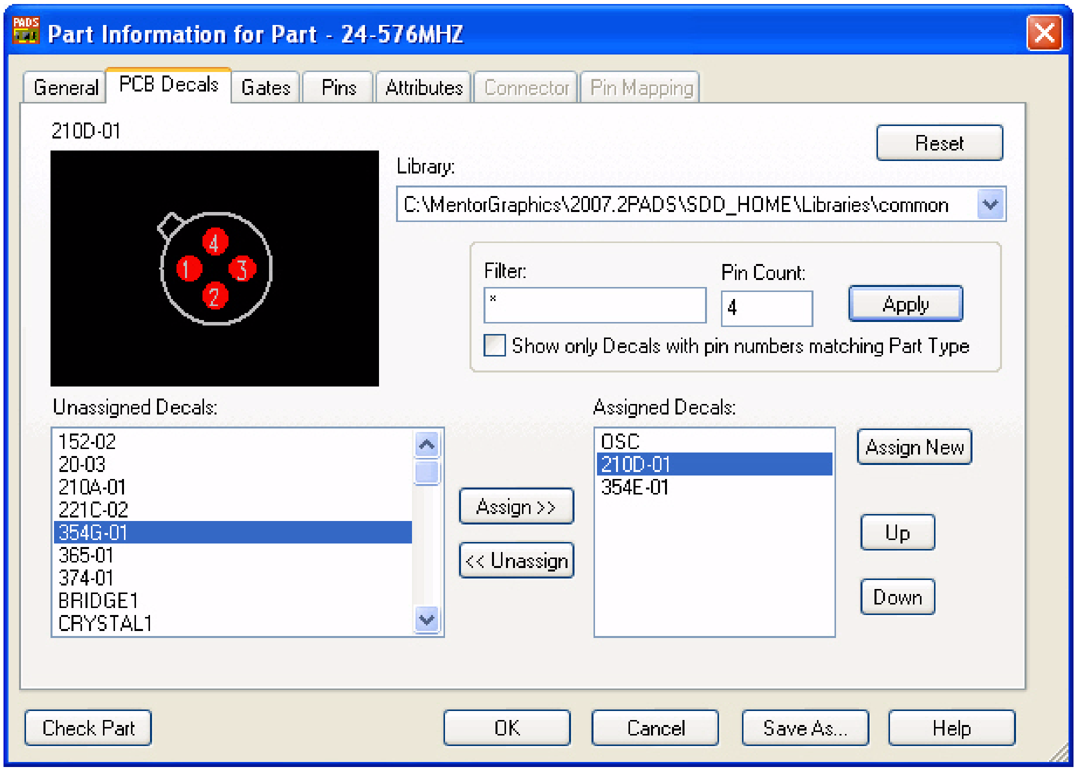


Table 45-253. PCB Decals Tab Contents

Name	Description
Preview area	Shows the item selected in the Assigned Decals list.
Reset	Undoes all changes you made in the PCB Decals tab.
Library list	Lists all your available libraries. Filters the Unassigned Decals list to only the selected library.
Filter	Narrows down your unassigned decals list. **Tips:** • You can use wildcards in this box. • Type * (asterisk) in the Filter box to display all decals.
Pin Count	Narrows down your unassigned decals list by displaying only the decals with the specified number of pins. **Tip:** Delete all numbers in the Pin Count box to display all decals. This box is always available as a filter to allow decals of differing pin counts to be assigned.
Apply	Executes the filter arguments.

Table 45-253. PCB Decals Tab Contents (cont.)

Name	Description
Show only Decals with pin numbers matching Part Type	Filters out decals that do not have pin numbers matching existing gate and signal pins on the Pins tab, or the physical pin numbers on the Pin Mapping tab.
Unassigned Decals list	Lists all unassigned decals available to assign from the selected library. **Tip:** Double-click a decal to assign it without needing to click the Assign button.
Assign >>	Moves the selected decal from the Unassigned Decals list to the Assigned Decals list. Assigned PCB decals can have a different number of pins, but you must assign a decal with enough pins for all the defined gate pins and signal pins on the Pins tab. **Restriction:** Only decals with sequential numerical pin numbers can be used with pin mapping.
<< Unassign	Moves the selected decal from the Assigned Decals list to the Unassigned Decals list.

Table 45-253. PCB Decals Tab Contents (cont.)

Name	Description
Assigned Decals list	Lists all assigned decals. Assigned PCB decals can have a different number of pins, but you must assign a decal with enough pins for all the defined gate pins and signal pins on the Pins tab. You can assign up to 16 PCB decals to a part. **Caution:** • Decals are switched to alternates using the Component Properties dialog box and can be changed outside of ECO mode. An .eco file created by the ECO toolbar will not contain decal changes to alternates unless you select the Output Decal Changes check box in the ECO Options. If you haven't specified to record decal changes in the .eco file written by the ECO Toolbar, you can use the Compare/ECO dialog box to create an .eco file that lists changes to alternate decals. • If you update a part in the library to have alternate decals and you use the Update from Library tool, but you don't select PCB decals to be updated, the updated alternates won't be listed in the Decal list for selection in the Component Properties dialog box. **Tips:** • The decal at the top of the list is the default decal and is used when you add the part to the design. • When you assign a decal, the pin numbers from the decal are automatically populated into the Pins tab table. PCB Decal pin numbers can be alphanumeric or numeric and the pin numbers in the PCB Decal must match the pin numbers listed in the Pins tab table.
Assign New	Opens the Assign New PCB Decal Dialog Box where you can type the name for a decal that does not yet exist in a library. In order to use this part in a design, you must either acquire or create this decal. **Restriction:** When you assign a decal that exists, it pre-populates the Pins tab with pin numbers. When you assign a new PCB decal, you must enter the pin numbers manually. See Adding a Series of Pins to the Pins Table for a procedure to quickly add a series of pins.
Up/Down buttons	Moves the selected Decal up or down. **Tip:** The decal at the top of the list is the default decal and is used when you add the part to the design.

Table 45-253. PCB Decals Tab Contents (cont.)

Name	Description
Check Part	Checks for missing or inconsistent information. **Tip:** Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.
Save As	Opens the Save Part Type to Library dialog box.

Related Topics

Creating a New Part Type

Part Information Dialog Box, Pins Tab

Use the Pins tab in the Part Information dialog box to assign gate pins, signal pins, and unused pins to the part. Pin numbers added to the Pins tab must match those of the PCB Decal. Use the Pin Mapping tab to overlay logical (schematic) alphanumeric pin numbers onto the physical numeric PCB decal.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **Pins** tab

Figure 45-283. Pins Tab

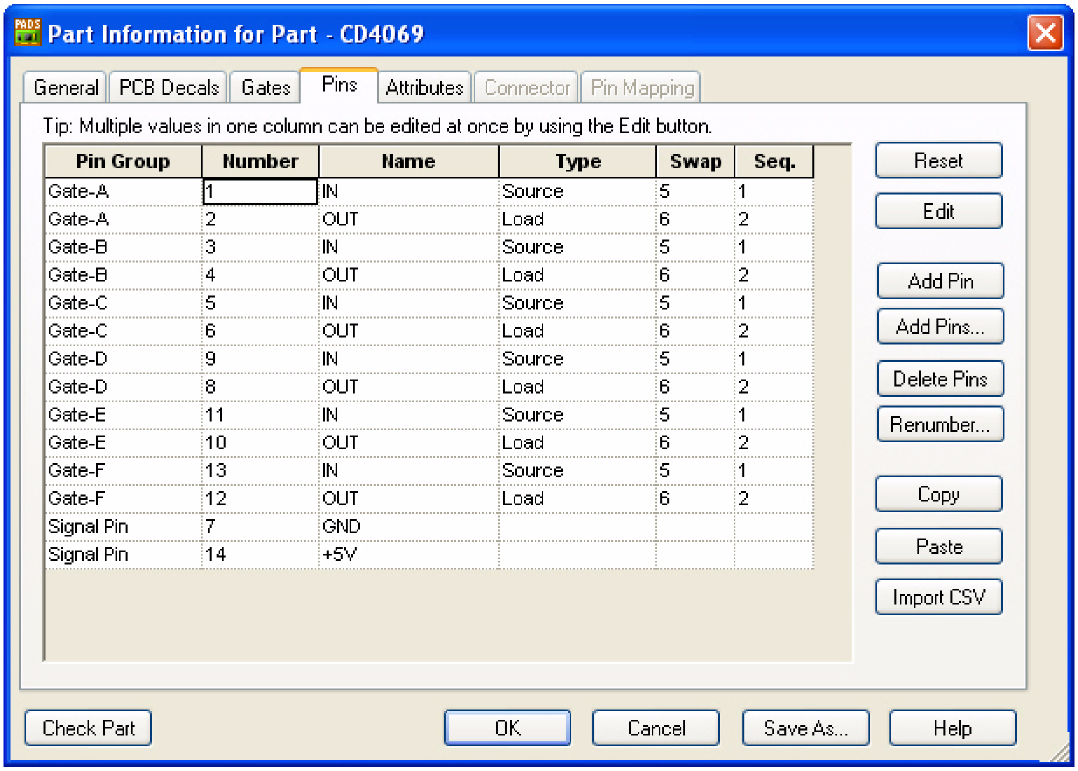


Table 45-254. Pins Tab Contents

Name	Description
Pins Table	Double-click a column to sort the column by ascending order. **Tip:** Sorting by pin Sequence number or Pin Group has the same effect, it sorts by Pin Group and then by sequence number within each gate.

Table 45-254. Pins Tab Contents (cont.)

Name	Description
Pin Group column	Select from Gate, Signal Pin, Connector Pin or Unused Pin. • **Gate Pins—**Assign to gates you've added to the part on the Gates tab. • **Signal Pins**—Assign to implicit pins (pins which are not displayed on any gate in the schematic). Typically, ground and power pins are the only implicit pins. You are not required to use Signal Pins. Instead, you can add power and ground pins to a gate or create a separate gate for power and/or ground pins. For the parts in the libraries shipped with PADS Logic, the standard ground signal name is GND. The standard power signal name is +5V. • **Connector Pins**—Assign to connector pins instead of using Gate pins. With a connector, every pin is its own gate in order to spread each connector pin throughout the schematic as needed, instead of having to create gates for each pin. You designate a part type as a connector on the General tab. • **Unused Pins—**You can assign a pin to be an unused pin. An unused pin is a pin that is defined in a PCB decal but has no electrical function in the part type. The unused pin information is not saved in the part type, but is derived automatically based on the number of assigned gate and signal pins to the number of pins in the assigned PCB decal.
Number column	Specifies the pin number for the pin. **Requirement:** The pin number must match the PCB decal. For example, alphanumeric to alphanumeric.
Name column	Specifies the pin signal or function name of the pin. **Requirement:** The pin must have a name to be valid.
Type column	Specifies the type of pin. **Tip:** This column is used with gate pins only.
Swap column	Specifies an identical number between pins that can be swapped. **Tip:** Type 0 to disable swapping.

Table 45-254. Pins Tab Contents (cont.)

Name	Description
Seq. column	Specifies the sequence number. **Tip:** The sequence number determines the mapping of CAE gate pins to PCB decal pins. The sequence is automatically shared with alternate CAE decals. For example, it shows how pin numbers appear on the CAE gate decal; therefore, in Gate A, sequence number 1 could be pin 1, but in Gate B, sequence number 1 would be pin 4.
Reset button	Undoes all changes you made in the Pins tab.
Edit button	Makes the selected cell available for editing. Press Ctrl or Shift and click to select multiple cells from the same column, then click Edit to make the same changes to the selected cells. The action of the edit button depends on the cells you select: • **Pin Group column**—Opens the Update Pin Gate dialog box. • **Number column**—Opens the Renumber Pins dialog box. • **Name column**—Opens the Update Pin Name dialog box. • **Type column**—Opens the Update Pin Type dialog box. • **Swap column**—Opens the Update Pin Swap dialog box. • **Seq. column**—Not available for multiple cell edits.
Add Pin button	Adds a new row below the selected row. If it's the first pin to be added it takes the default of belonging to Gate-A. If pins already exist, the new pin takes the Pin Group of the currently selected pin. **Requirement:** You must add a pin number to make the pin valid. **Tips:** • You can add all pins automatically by adding a decal. • To add a series of pins, click the Add Pins button. • To import pins using a comma separated value (.csv) file, click the Import CSV button.

Table 45-254. Pins Tab Contents (cont.)

Name	Description
Add Pins button	Opens the Add Terminals Dialog Box. **Restriction:** Total pins for the part can not exceed 32,767 pins. **Tips:** • You can add all pins automatically by adding a decal. • To add a single pin, click the Add Pin button. • To import pins using a comma separated value (.csv) file, click the Import CSV button.
Delete Pins button	Removes the selected row.
Renumber button	Opens the Renumber Pins Dialog Box.
Copy button	Places the selected cell information in the paste buffer. **Tip:** You can also copy from Microsoft Excel.
Paste button	Pastes the information from the paste buffer. The selected cell in the table is the paste origin. Data is pasted below and to the right of the paste origin. **Restriction:** When the pasted data includes either Pin Group or Pin Number data, extra pin rows are added automatically, otherwise the paste will fail if the number of rows and columns in the pasted data does not match those available in the table below and to the right of the paste origin.
Import CSV button	Opens the Library Import File dialog box where you select the .csv file to import. **Tips:** • The entire contents of the Pins tab table is replaced with the data of the CSV file. • CSV field names must correspond to the column headers in the Pins tab table. Only the first two characters of the header must match. For example, "Pi" for the Pin Group column. Gate or "Ga" are acceptable alternatives to the Pin Group header. • The sample Part_Pins_Template.csv file is located in your \PADS Projects\Samples folder.
Check Part button	Checks for missing or inconsistent information. **Tip:** Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.
Save As button	Opens the Save Part Type to Library dialog box where you choose your library and type a name for the part.

Related Topics

Adding a Series of Pins to the Pins Table

Editing Pins Table Data

Renumbering Pins in the Pins Table

Creating a New Part Type

Part Information Dialog Box, Pin Mapping Tab

Use the Pin Mapping tab in the Part Information dialog box to overlay alphanumeric pin numbers onto numeric PCB decal pins. Prior to PADS 2007, alphanumeric pin numbers could not be saved in PCB decals.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **Pin Mapping** tab

  Requirements:

  - On the General tab, select the **Define mapping of Part Type pin numbers to PCB Decal** check box to make the Pin Mapping tab available.
  - On the PCB Decals tab, assign a decal with sequential numerical pin numbers to use the Pin Mapping tab. The decal determines the number of pins in the part.

Figure 45-284. Pin Mapping Tab

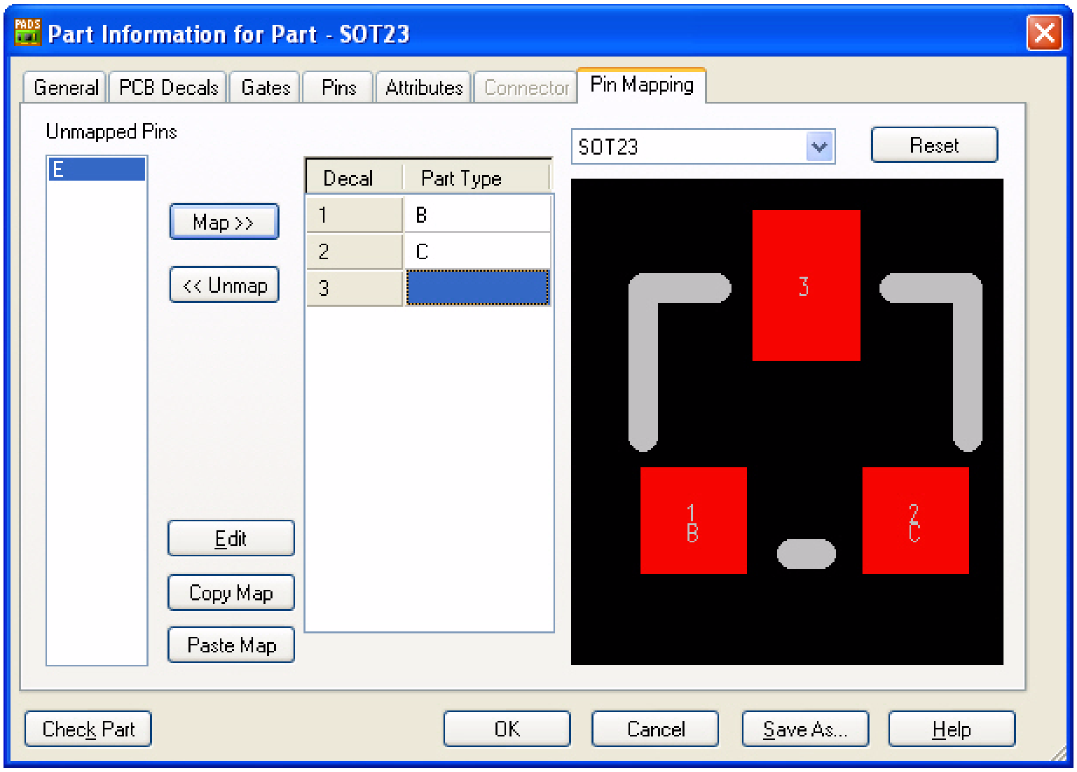


Table 45-255. Pin Mapping Tab Contents

Name	Description
Decal list	Lists the decals available to you for which you can map alphanumeric pins.
Reset	Undoes all changes you made in the Pin Mapping tab.
Unmapped Pins list	Lists all unmapped pins available to map in the Mapping table.
Map >>	Moves the selected pin from the Unmapped pins list to the selected cell in the Mapping table.
<< Unmap	Moves the selected decal number from the Mapping table to the Unmapped Pins list.
Mapping table	• **Decal column**—The number of the Decal. • **Part Type column**—The value of the Attribute.
Edit	Makes the selected cell available for editing. **Tip:** You can also double-click the cell to edit the contents.

Table 45-255. Pin Mapping Tab Contents (cont.)

Name	Description
Copy Map	Places the map information into the paste buffer to paste into Microsoft Excel where you can make mass edits. **Restriction:** Copy Map only works with the whole pin mapping table and not selective rows.
Paste Map	Pastes the map information from the paste buffer. **Restriction:** Paste Map only works with the whole pin mapping table and not selective rows.
Preview area	Shows the item selected in the Decal list. You can assign unmapped pins to decal pins by selecting the pins in the preview window. Select an alphanumeric in the Unmapped Pins list and double-click the pin in the decal preview window to map the alphanumeric to the pin. The next row in the Unmapped Pins list becomes the next selected alphanumeric for mapping. In the preview window, you can click and drag to define a zoom box, or use Shift+click or Shift+right-click to zoom in or out by a factor of two. You can zoom in up to 16X the original scale. The preview window will only zoom out to fit the decal entirely in the view.
Check Part	Checks for missing or inconsistent information. **Tip:** Even if you don't click the Check Part button, when you exit the tab, the assigned decals are checked to ensure that they contain physical pin numbers for all the gate and signal pins defined in the Pins tab.
Save As	Opens the Save Part Type to Library dialog box.

Related Topics

Creating a Part Type with Different Schematic and Layout Pin Numbering

Mapping Alphanumeric Pin Numbers to Numeric Decals

Creating a New Part Type

Part Label Properties Dialog Box

Use the Part Label properties dialog box to modify a label and to change the attribute the label displays.

Tip: If you select multiple labels, settings in this dialog box apply to all selected labels.

Accessing

- **Select a part label > Right-click > Properties**

Figure 45-285. Part Label Properties Dialog Box

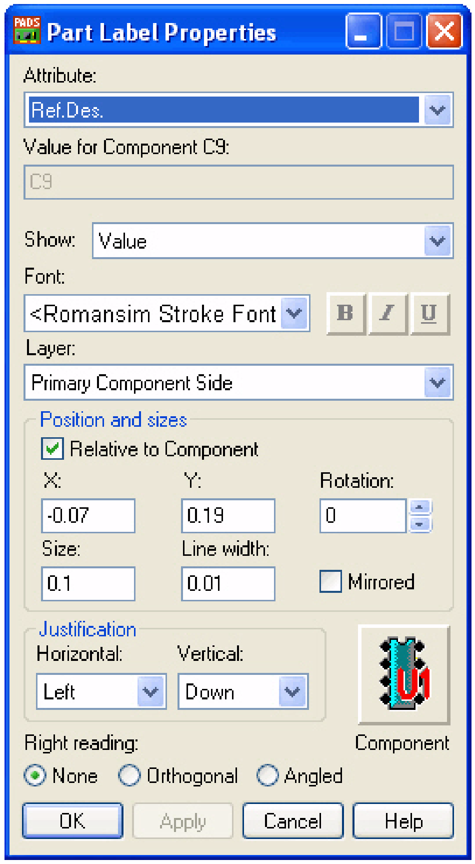


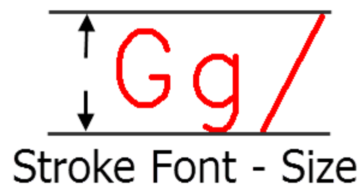
Table 45-256. Part Label Properties Dialog Box Contents

Name	Description
Attribute	The attributes available to you. If you are creating labels for jumpers, Reference Designator is the only available attribute. **Tip:** Hidden attributes do not appear in the Attribute list unless the attribute was selected for the label before it was set as a hidden attribute.

Table 45-256. Part Label Properties Dialog Box Contents (cont.)

Name	Description
Value for	The value of the selected attribute. **Tips:** • Unavailable if you selected Reference Designator or Part Type from the Attribute list, if the attribute is read-only, or if you open the Properties dialog box for labels of different attribute types. However, if the labels you select belong to attributes of the same type, you can edit this box. • If the attributes have different values, the box is blank. You can type a new value in the box to apply it to all of the selected attribute labels and their parent objects. • Value is also unavailable if the attribute is ECO-registered and PADS Layout is not in ECO mode.
Show	Controls the visibility of the label. • **None**—Turns visibility off. • **Value**— Displays only the label value. • **Name and Value**—Displays the name and value. • **Full Name and Value**—When labeling a structured attribute, displays the full structured name and value. **Tip:** Labels are invisible regardless of this setting unless you use the Display Colors Setup dialog box to change the color of labels to a color different from that of the background.
Font	The fonts available to you. **Tips:** • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.
Layer	The layers available to you.
Relative to	Places the label at the X and Y location relative to the component or the jumper. If you clear this check box, the label is placed at the X and Y location relative to the design origin.
X,Y	Places the decal label in a specified location.
Rotation	Specifies the rotation angle of the label.

Table 45-256. Part Label Properties Dialog Box Contents (cont.)

Name	Description
Size	Specifies the size of the font. **Size (pts):** This is font size in points and appears for system fonts **Size (mils):** This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters. 
Line Width	Specifies the line width for stroke fonts only. 
Mirrored	Flips the label - text is considered readable from the bottom side of the board.
Horizontal, Vertical	Sets the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change. **Tips:** • For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**. • Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.
Right reading	Controls whether labels are readable (from left to right, or from bottom to top if the label is rotated). Click the **None**, **Orthogonal**, or **Angled** button to indicate the direction of reading you want.
Component button	Opens the Component Properties dialog box.

Related Topics

Modifying Part Label Properties

Part Type List for Decal Dialog Box

Use the Part Type List for Decal dialog box to check your part for errors. You can check to ensure the pins match the pins listed in the Part Type > Pins tab table, or when pin mapping exists - on the Part Type > Pin Mapping tab.

Accessing

- Opens automatically when you open a decal in the Decal Editor
- **Tools** menu > **PCB Decal Editor** > **Tools** menu > **Part Types**

Figure 45-286. Part Type List for Decal Dialog Box

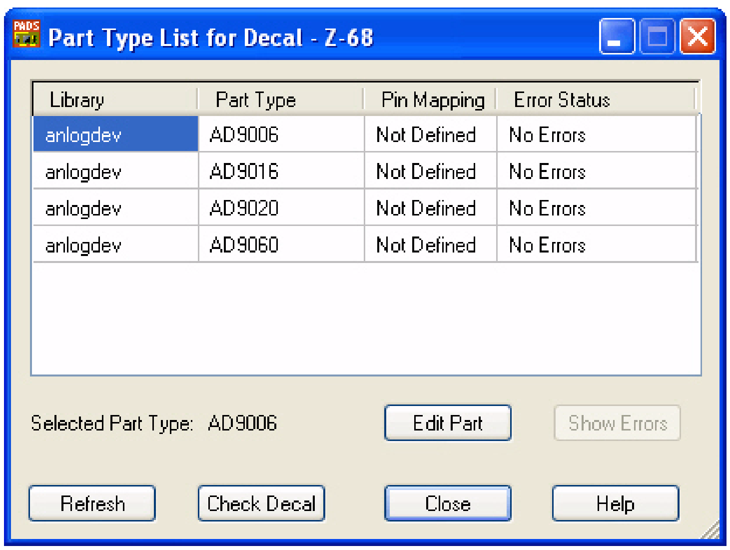


Table 45-257. Part Type List for Decal Dialog Box Contents

Name	Description
Library column	Displays the associated part type library.
Part Type column	Displays the part type name.
Pin Mapping column	Displays whether a pin mapping is defined.
Error Status column	• **No Errors**—No errors exist within the part type. • **Logical Errors**—Errors exist within the part type. • **Mismatched Pins**—The decal does not contain all the pin numbers defined in the part type.

Table 45-257. Part Type List for Decal Dialog Box Contents (cont.)

Name	Description
Edit Part	Opens the Part Information dialog box.
Show Errors	Create and opens a report with details of the errors.
Refresh	Updates the dialog box with any fixes you have made.
Check Decal	Checks the decal against all associated part types to show all mismatched pin number errors.

Related Topics

Checking for Errors Between Part Types and Assigned PCB Decals

PCB Decal Editor

PADS Layout uses components from the parts libraries. Every part in the library has a decal associated with a part type in a parts library. Use the PCB Decal Editor to create or edit these decals. Many of the drafting operations in the PCB Decal Editor are identical to those in the Layout Editor.

When the PCB Decal Editor opens, the open design is stored and the PCB Decal Editor interface replaces the Layout Editor. Use the File commands to save information and exit the PCB Decal Editor as you would a stand-alone program. When you exit the PCB Decal Editor, you return to the open design in PADS Layout.

If during a PCB Decal Editor session you try to change the decal to increased layer mode and the current design is in default layer mode, the message “You will not be able to apply decal changes to the design. Continue?” appears. Either click Cancel to return to the Layers Setup Dialog box without changing the decal to increased layer mode, or press OK to proceed to the Increase Maximum Layer Number dialog box. You can save the decal changes to library and choose to exit the PCB Decal Editor without applying changes. Then switch the current design to increased layer mode and update the decal.

Tips:

- PADS Layout supports 16 alternate decals per part type. The PCB Decal Editor supports up to 65,536 components.
- You can use Dimensioning within the PCB Decal Editor; however, dimensions are converted to 2D lines and text when you save the decal.

Recommendation: Place free text and attribute values on the Silkscreen Top layer to avoid DRC violations or shorts.

Accessing

- **Tools** menu > PCB **Decal Editor**

Related Topics

Creating a New Decal

PDF Configuration Dialog Box

Use this dialog box to save one or more layers of your design as images in a PDF document. You can specify the contents of the document, its format, and its behavior, including the kinds of design items readers can search for, and how they are displayed.

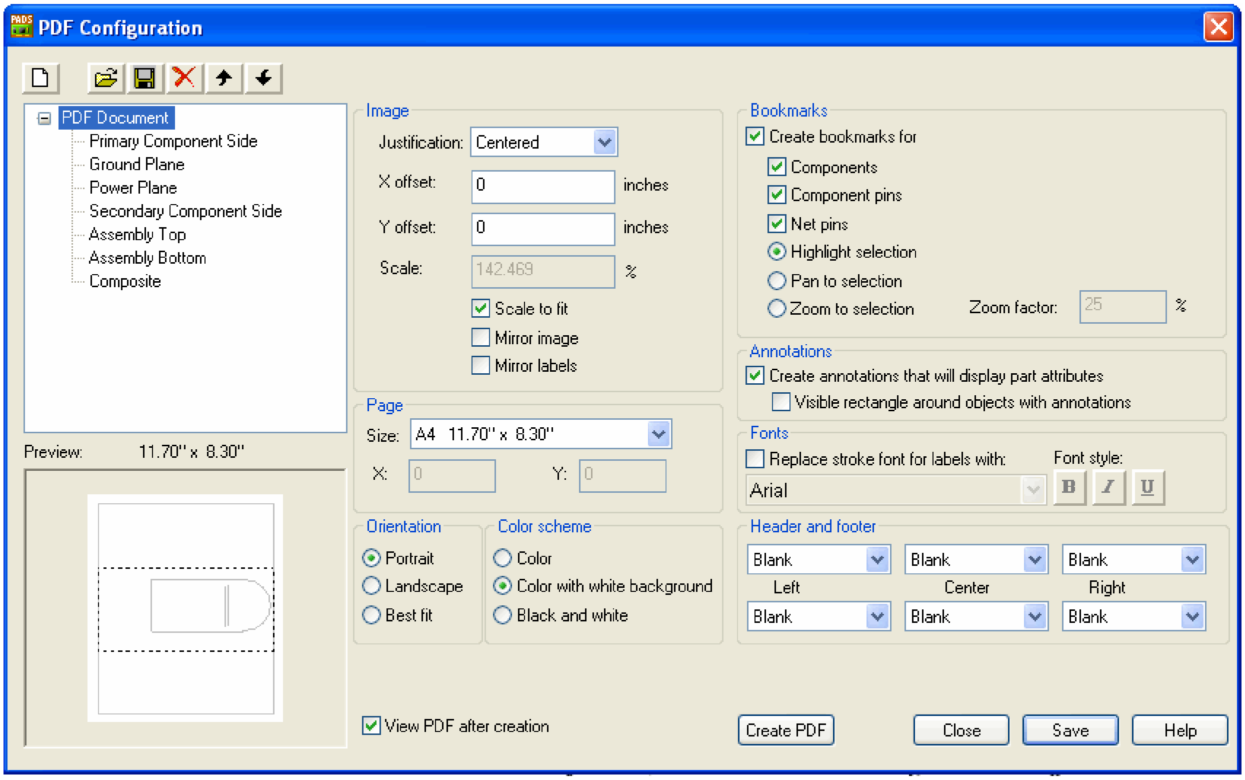
Accessing

- **File menu > Create PDF**

This dialog box displays its controls in two views--a "document" view and a "page" view.

The document view appears when the root item (a PDF configuration) is selected in the page list at the top left of the dialog box. Settings made in the Document view affect all pages in the document.

Figure 45-287. PDF Configuration Dialog Box, Document View



The page view appears when a single page is selected in the page list. Its controls affect only the selected page.

Figure 45-288. PDF Configuration Dialog Box, Page View

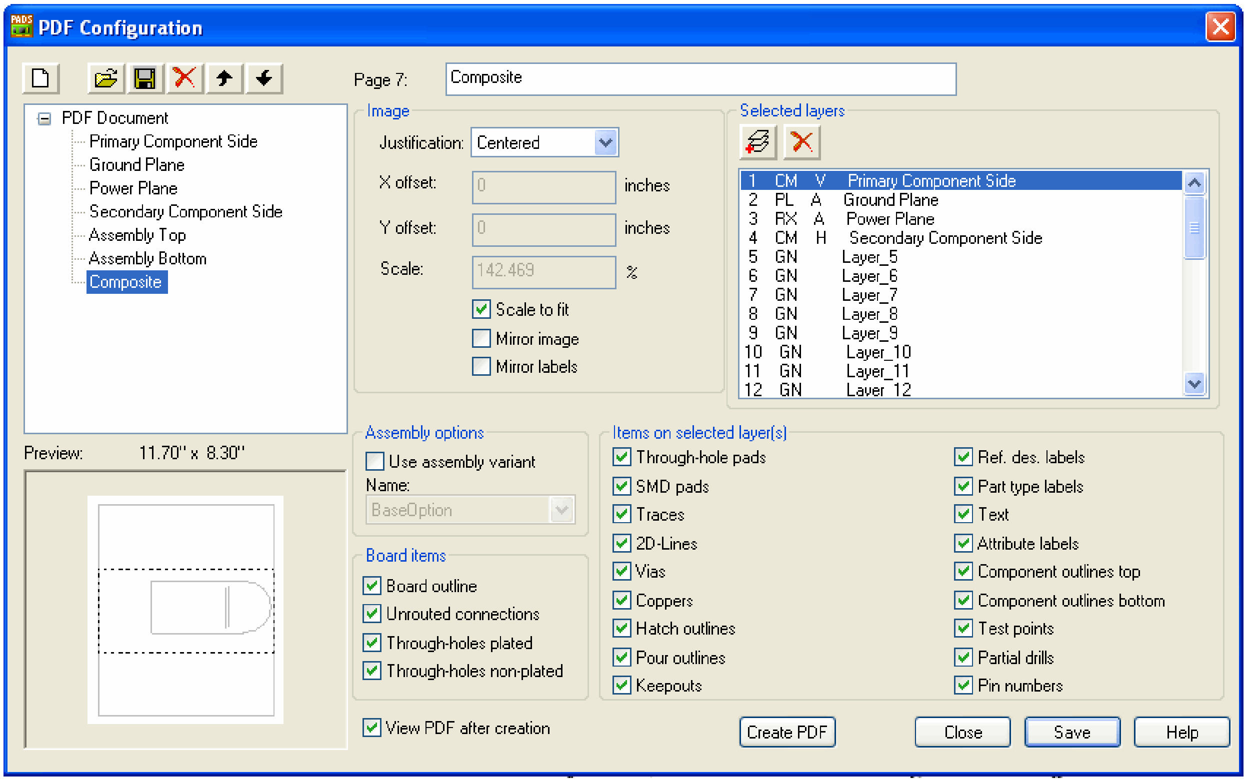


Table 45-258. PDF Configuration Dialog Box Controls

Control/Area	Description
Page-list Toolbar (Both views)	
Add Page button	Adds a page at the bottom of the Page list.
Import Configuration button	Imports a PDF configuration from a .pdc file.
Export Configuration button	Exports the current dialog box settings (that is, the current configuration) to a new or existing configuration (.pdc) file. **Tip:** Use this button to save the current dialog box settings and reuse them for other designs.
Delete Page button	Deletes the selected page from the page list.
Move Page Up button	Moves the selected page up in the Page list.
Move Page Down button	Moves the selected page down in the Page list.
Page-list area (Both views)	
	Select the top of the Page list (the PDF configuration name) to display the Document view, where you specify options for the PDF document as a whole. Select any individual page to display the Page view, where you specify options for the selected page. **Tip:** The Composite page, by default, includes all the selected layers. You can modify the layer assignments for this page (like all the other pages) with the Add layer and Remove layer buttons in the Selected layers area.
Preview area (Both views)	
	Shows the position of the selected design layer image(s) on the PDF page, as follows: • The grey outer rectangle shows the available image area inside the margins and the header and footer. • The dashed-line box shows the image boundary (scaled, if Scale to fit is selected). • Inside the image boundary, the board outline is shown (scaled, if Scale to fit is selected).
Page <page_number>: edit box	
	Displays the name of the page selected in the page list. You can edit the page name here.
Image area (Both views)	
Justification	Specifies the position of the design layer image(s) on the PDF page. Select Bottom Left, Bottom Right, Centered, Top Left, Top Right, or Use offset.

Table 45-258. PDF Configuration Dialog Box Controls

Control/Area	Description
X offset	Specifies the distance of the lower left corner of the image boundary from the left side of the PDF page boundary.
Y offset	Specifies the distance of the lower left corner of the image boundary from the bottom of the PDF page boundary. **Tip:** When the PDF Configuration is selected, this checkbox may display a greyed-out check mark or a small square; these indicate that the settings of the individual pages vary.
Scale	Specifies the size of the PDF image as a percentage of the actual design size. For example: • 100%—The image is displayed actual size. • 50%—The image is displayed half-size. • 200%—The image is displayed double-size.
Scale to fit	Specifies that the PDF image should be enlarged or reduced to fit within the margins of the PDF sheet. **Tip:** When the PDF Configuration is selected, this checkbox may display a greyed-out check mark or a small square; these indicate that the settings of the individual pages vary.
Mirror image	Flips the PDF image left/right. **Tip:** When the PDF Configuration is selected, this checkbox may display a greyed-out check mark or a small square; these indicate that the settings of the individual pages vary.
Mirror labels	Flips standard and attribute labels left/right. **Tip:** When the PDF Configuration is selected, this checkbox may display a greyed-out check mark or a small square; these indicate that the settings of the individual pages vary.
Page area (Document view only)	
Size	Select a PDF page size, or select “Custom” to specify a non-standard sheet size.
X	Specifies the width of the custom sheet *in current design units*.
Y	Specifies the height of the custom sheet *in current design units*.
Orientation area (Document view only)	

Table 45-258. PDF Configuration Dialog Box Controls

Control/Area	Description
	Specifies the orientation of the design image on the PDF sheet(s): Portrait, Landscape, or Best fit.
Color Scheme area (Document view only)	
	Specifies the color scheme for the PDF: Color (with design background color), Color with white background, or Black and White.
Bookmarks area (Document view only)	
Create bookmarks for	Create PDF bookmarks (Table of Contents entries) for the items selected (Components, Component pins, Net pins)
Highlight selection	Specifies that the Acrobat PDF viewer should: • Highlight an item when it is selected. • If the item is located outside the PDF document window, pan to move it to the center of the window.
Pan to selection	Specifies that the Acrobat PDF viewer should pan to an item when it is selected. The item will appear in the upper left corner of the PDF document window.
Zoom to selection	Specifies that the PDF viewer should pan and zoom to an item when it is selected, using the zoom factor set in the Zoom factor field. The item will appear in the center of the PDF document window.
Zoom factor	Sets the factor to be used when zooming to a selected item: • 100% is the maximum; the item will occupy the entire window. • 10% is the minimum; the item will occupy 10% of the window area.
Annotations area (Document view only)	
Create annotations that will display part attributes	Creates popup notes that display an item’s attributes when the reader left-clicks on the item.
Visible rectangle around object with annotations	Draws a dotted line rectangle around each annotated item.
Fonts area (Document view only)	
Replace stroke font for labels with:	Uses the font selected in the box to display labels that use Stroke font in the design.
B, I, and U buttons	Applies any or all of Bold, Italic and Underline styles to labels converted from Stroke font.

Table 45-258. PDF Configuration Dialog Box Controls

Control/Area	Description
Header and Footer area (Document view only)	
	Specifies optional text for the left, right and center of the header and footer. At the bottom of the list, select Custom to open the Custom String dialog box and type your own custom text.
Assembly Options area (Page view only)	
Use assembly variant	Includes in the PDF document only components belonging to the variant selected in the Name: box. The visibility of the following design items is affected: • Component outlines top • Component outlines bottom • RefDes labels • Part type labels • Component attribute labels
Board Items area (Page view only)	
	Includes in the PDF document PCB design items not belonging to a specific layer. Select any or all of Board outline, Unrouted connections, Through-holes plated, and Through-holes non-plated.
Selected Layers area (Page view only)	
List box	Lists the layers that will be shown on this page of the PDF document.
Add Layer button	Opens the Layer Selection dialog, where you can edit the list of layers assigned to this PDF page.
Remove Layer button	Removes the selected layer(s) from the list.
Items on selected layer(s) area (Page view only)	
	Includes items in this page of the PDF document. Select any or all of Through-hole pads, SMD pads, Traces, 2D-Lines, Vias, Coppers, Hatch outlines, Pour outlines, Keepouts, Ref. des. labels, Part type labels, Text, Attribute labels, Component outlines top, Component outlines bottom, Test points, Partial drills, and Pin numbers.

Pen Plotter Advanced Setup Dialog Box

Use the Pen Plotter Advanced Setup dialog box to add a new pen plotter to the Device list of available plotters.

Restriction: You cannot modify the PADS-supplied plotter advanced settings - you only edit the data for an existing plotter if it was created originally from this dialog box. The new plotter must support one of the two supported interface languages: HPGL or HGML. Other plotter languages or formats are not yet supported.

Accessing

- **File** menu > **CAM** > **Add** button > **Pen** button > **Device Setup** button > **Advanced** button

  or

- **File** menu > **CAM** > select a document name > **Edit** button > **Pen button** > **Device Setup** button > **Advanced** button

Figure 45-289. Pen Plotter Advanced Setup Dialog Box

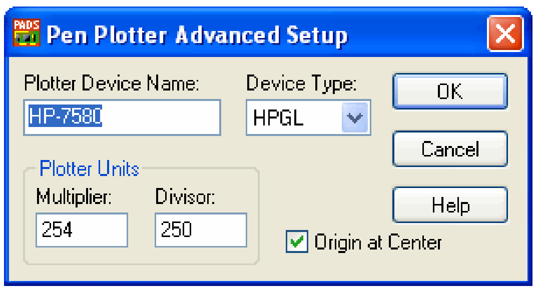


Table 45-259. Pen Plotter Advanced Setup Dialog Box contents

Name	Description
Plotter Device Name	Specifies the name of a different pen plotter you want to use. **Restriction:** You cannot reuse one of the existing, supplied device names.
Device Type	Specifies the interface language the plotter uses: HPGL or HGML.
Plotter Units area	Sets the plotter resolution by providing a scaling ratio. The ratio defined is the scale factor to convert from mils (0.001 in) to plotter units. **Example:** Most Hewlett-Packard plotters have a resolution of 0.025 mm or 1/40 mm. This means that a distance of one inch (1000 mils) is 1016 plotter units (25.4 X 40). So a ratio of 1016 to 1000 would be defined. The ratio actually used is 254 to 250 which is the same as 1016 to 1000.
Origin at Center	Specifies that the origin of the plotter is at the center of the paper. Clear this check box if the origin is in the lower left corner or other location.

Related Topics

Pen Plotter Setup Dialog Box

Creating CAM Outputs to Manufacture Your PCB

Pen Plotter Setup Dialog Box

Use the Pen Plotter Setup dialog box to set various pen plot options.

Accessing

- **File** menu > **CAM** > **Add** button > **Pen** button > **Device Setup** button

  or

- **File** menu > **CAM** > select a document name > **Edit** button > **Pen button** > **Device Setup** button

Figure 45-290. Pen Plotter Setup Dialog Box

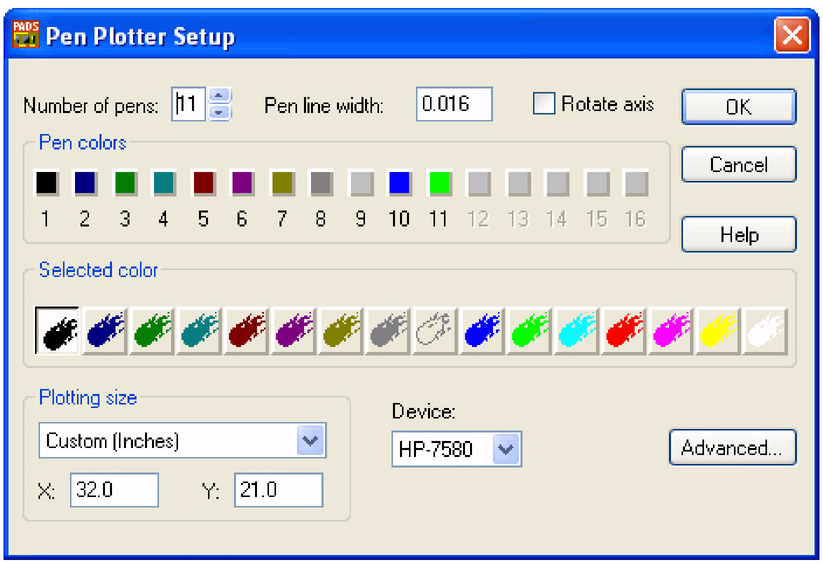


Table 45-260. Pen Plotter Setup Dialog Box contents

Name	Description
Number of pens	Specifies the number of pens in your device (1-16).
Pen line width	Specifies the line width of your pen in mils.
Rotate axis	Reverses the X and Y axes of the design

Table 45-260. Pen Plotter Setup Dialog Box contents (cont.)

Name	Description
Pen colors	Specifies the color of each pen. To assign colors, select a color tile from the Selected color area and then click a pen number. Tip: The assigned colors are used to customize CAM document objects through the Select Items dialog box.
Selected color	Specifies the colors available for your Pen colors.
Plotting Size area	Specifies the plotting size you want to use. **Tip:** To define a custom size, select Custom (mm) or Custom (inches), and then type the X and Y dimensions to use.
Device	Specifies the plotter device to use. **Tip:** If you can't find your device in the list, click Advanced to open the Pen Plotter Advanced Setup Dialog Box and create a new device listing.
Advanced	Opens the Pen Plotter Advanced Setup dialog box.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Photo Plotter Advanced Setup Dialog Box

Use the Photo Plotter Advanced Setup dialog box to customize the output of photo plot files.

Accessing

- **File** menu > **CAM** > **Add** button > **Photo** button > **Device Setup** button > **Advanced** button

  or

- **File** menu > **CAM** > select a document name > **Edit** button > **Photo** button > **Device Setup** button > **Advanced** button

Figure 45-291. Photo Plotter Advanced Setup Dialog Box

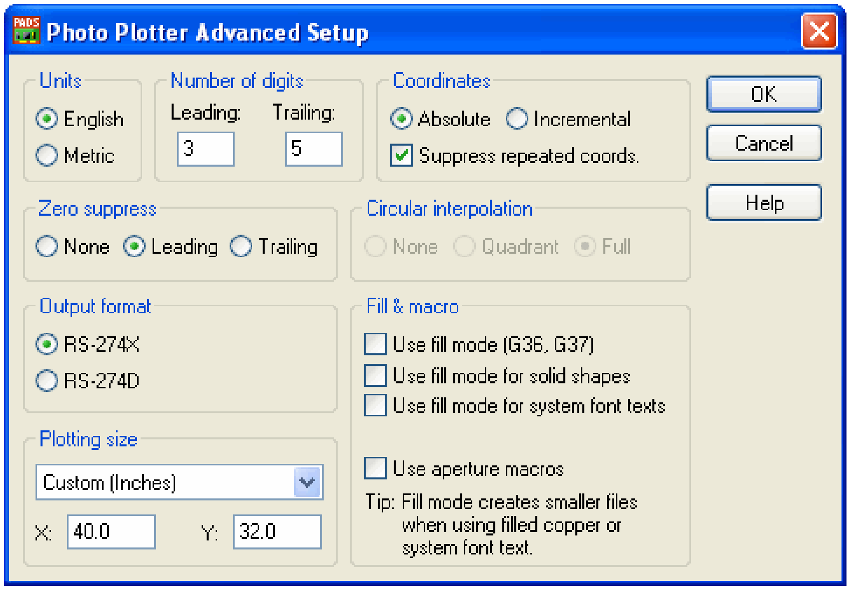


Table 45-261. Photo Plotter Advanced Setup Dialog Box contents

Name	Description
Units	Sets the CAM file measurement units. • **English**—mils • **Metric**—millimeters
Leading	Specifies the precision of the output file coordinates; the number of digits that should lead the decimal point.
Trailing	Specifies the precision of the output file coordinates; the number of digits that should trail the decimal point.
Coordinates	Sets the coordinates for the output file. • **Absolute**—referenced to the origin. • **Incremental**—measured relative to the previous coordinate.
Suppress repeated coords	Eliminates repeated coordinates from the output file.
Zero suppress	Defines how to handle zero suppression in the output file. • **None**—retains leading and training zeros. • **Leadin** —suppresses zeros before the decimal point. • **Trailing**—suppresses zeros after the decimal point.

Table 45-261. Photo Plotter Advanced Setup Dialog Box contents (cont.)

Name	Description
Circular interpolation	Defines how to draw arcs and circles: • **None—Specifies that** your photo plotter does not support circular interpolation. Arcs and circles are drawn as small straight-line segments. • **Quadrant—Specifies that** your photo plotter does not support full, 360-degree circular interpolation. • **Full—Specifies that** your photo plotter supports full, 360-degree circular interpolation. **Restriction:** these options are only available for the RS-274-D format. If you select RS-274-X format, these options are unavailable and is automatically set to Full. All devices that support RS274X support this setting. **Tip:** Use RS-274-X output to avoid generating a large number of hatch lines in the Gerber output. This also applies to copper chamfered paths.
Output format	Specifies the output format you want to use. The RS-274-X data format has more features and contains embedded aperture information while the RS-274-D format is older and uses a separate file for aperture information. **Restrictions:** When you select RS274X: • The Circular Interpolation options are unavailable and is automatically set to Full. All devices that support RS274X support this setting. • The Aperture Count in the Photo Plotter Setup Dialog box is unavailable and is automatically set to 989. All devices that support RS274X support this setting. **Tip:** The Output Format setting is saved in the devicesn.dat file with all other photo plotter data.
Plotting size list	Sets the plotting size, select a standard Plotting size from the list, or select **Custom (Inches)** or **Custom (mm)** and enter the X and Y dimensions to define a custom size.

Table 45-261. Photo Plotter Advanced Setup Dialog Box contents (cont.)

Name	Description
Use fill mode (G36, G37)	Uses fill mode to draw all filled regions (including solid shapes and system font texts) in the design. Fill mode avoids generating a large number of hatch lines in the Gerber output. This also applies to copper chamfered paths. **Warning:** When using fill mode, compare your final RS-274-X output (whether it is from PADS Layout or a Gerber viewer) against output from RS-274-D. Use a Gerber viewer to perform the comparison and check for overfilling or missing fills. **Tip:** When this option is selected, the **Use fill mode for solid shapes and Use fill mode for system font texts** options are selected and unavailable. **Note:** This option uses G36, G37 pairs to draw filled regions. All filled regions for which fill mode is not specified by one of these commands are plotted as hatched by strokes, as in PowerPCB 1.6 or earlier.
Use fill mode for solid shapes	Uses fill mode to draw shapes having the “solid copper” property. Fill mode avoids generating a large number of hatch lines in the Gerber output. This also applies to copper chamfered paths. **Warning:** When using fill mode, compare your final RS-274-X output (whether it is from PADS Layout or a Gerber viewer) against output from RS-274-D. Use a Gerber viewer to perform the comparison and check for overfilling or missing fills. **Restriction:** Unavailable when the **Use fill mode** is selected. **Note:** This option uses G36, G37 pairs to draw filled regions. All filled regions for which fill mode is not specified by one of these commands are plotted as hatched by strokes, as in PowerPCB 1.6 or earlier.
Use fill mode for system font texts	Uses fill mode to draw system font texts. Fill mode avoids generating a large number of hatch lines in the Gerber output. This also applies to copper chamfered paths. **Warning:** When using fill mode, compare your final RS-274-X output (whether it is from PADS Layout or a Gerber viewer) against output from RS-274-D. Use a Gerber viewer to perform the comparison and check for overfilling or missing fills. **Restriction:** Unavailable when the **Use fill mode** is selected. **Note:** This option uses G36, G37 pairs to draw filled regions. All filled regions for which fill mode is not specified by one of these commands are plotted as hatched by strokes, as in PowerPCB 1.6 or earlier.

Table 45-261. Photo Plotter Advanced Setup Dialog Box contents (cont.)

Name	Description
Use aperture macros	Defines aperture macros for associated pin copper in the RS-274-X Gerber files. When this option is off, associated coppers are hatched coppers, as in RS-274-D output. This prevents the reuse of D-codes that are not supported in some software.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

RS-274-X Format

Adding or Editing CAM Documents

Photo Plotter Setup Dialog Box

Use the Photo Plotter Setup dialog box to set up photo plotting and send your output to a plotter

Accessing

- **File** menu > **CAM** > **Add** button > **Photo** button > **Device Setup** button

  or

- **File** menu > **CAM** > select a document name > **Edit** button > **Photo** button > **Device Setup** button

Figure 45-292. Photo Plotter Setup Dialog Box

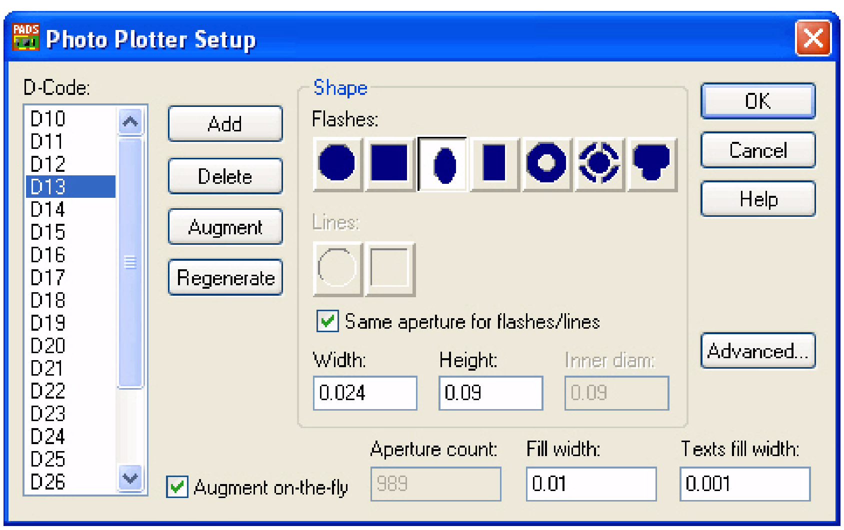


Table 45-262. Photo Plotter Setup Dialog Box contents

Name	Description
D-Code	Contains the apertures required for the Photo Plotter. The D-Code list can be created automatically or you can maintain it manually. The D-Code list contains all defined apertures. **Tip:** Before you generate your CAM files, regenerate your list of apertures.
Add button	Adds a new D-code to the D-Code list. **Tip:** Type a value excluding the “D” prefix.
Delete button	Removes the selected D-code from the D-Code list.
Augment button	Automatically generates D-codes for items in the design file that are not in the current list. **Tip:** Augment adds oval and rectangular flash apertures for orthogonal orientations only.
Regenerate button	Clears the current D-code list and automatically adds D-codes for all items in the design**.** **Tip:** Regenerate adds oval and rectangular flash apertures for orthogonal orientations only.
Flashes	Sets a flash aperture.
Lines	Sets a line aperture. **Restriction:** Unavailable if Same aperture for flashes/lines is selected.
Same aperture for flashes/lines	Specifies to draw lines and flashed items with the same aperture.
Width	Specifies a width for square, rectangle, and oval shapes or the outer diameter of round and thermal shapes. **Restriction:** This box is unavailable if a value is not appropriate for the specified shape.
Height	Specifies a height for oval and rectangular shapes. **Restriction:** This box is unavailable if a value is not appropriate for the specified shape.
Inner diam	Specifies an inner diameter for annular ring shapes **Restriction:** This box is unavailable if a value is not appropriate for the specified shape.
Augment on-the-fly	Automatically adds apertures as you add information to the design.

Table 45-262. Photo Plotter Setup Dialog Box contents (cont.)

Name	Description
Aperture count	Specifies the maximum aperture count. **Restriction:** When you click RS-274-X format in the Photo Plotter Advanced Setup dialog box, the aperture count is set to 989 and is unavailable.
Fill width	Specifies the width used for shapes to be filled. An example shape is a non-orthogonal pad. Larger widths decrease photo plot time but provide a less precise approximation of the shape.
Texts fill width	Specifies the width used for hatching of system texts. Larger widths decrease photo plot time but provide a less precise approximation of the text.
Advanced button	Opens the Photo Plotter Advanced Setup dialog box

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Setting Up the Photo Plotter Output

Adding or Editing CAM Documents

Pin Numbers Dialog Box

Use the Pin Numbers dialog box to interactively renumber the terminals in the design area. Selecting pin numbers in the dialog box selects the matching pins in the design area and selecting pins in the design area selects the matching pins in the dialog box.

Accessing

- **PCB Decal Editor** > **Setup** menu > **Pin Numbers**

Figure 45-293. Pin Numbers Dialog Box

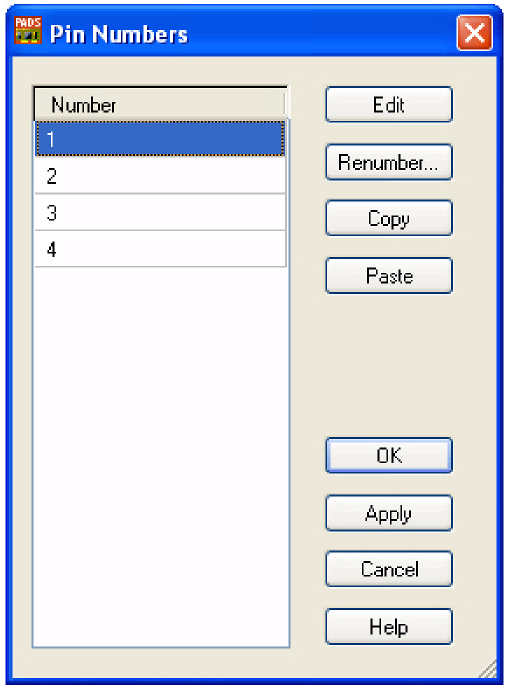


Table 45-263. Pin Numbers Dialog Box Contents

Name	Description
Number	The number of the pin available for renumbering.
Edit	Makes the selected pin number available for editing.
Renumber	Opens the Renumber Pins dialog box.
Copy	Places the selected pin numbers into the paste buffer.
Paste	Pastes the pin number information from the paste buffer. **Restriction:** Data only pastes into selected cells.

Related Topics

Renumbering Using the Pin Numbers Dialog Box

Pin Pair Properties Dialog Box

The Pin Pairs Properties dialog box displays the netname of which the pin pair is part, the pin-to-pin connection, routing information, and rules data.

The Pin Pair Properties dialog box remains open until you click **OK** or Cancel. If you select another pin pair while the dialog box is open, the information updates for the selected pin pair.

Tip: Several of the options in this dialog box are unavailable if the pin pair is part of a physical design reuse or contains protected routes.

Accessing

- **Select a pin pair** > **Right-click** > **Properties**

Figure 45-294. Pin Pair Properties Dialog Box

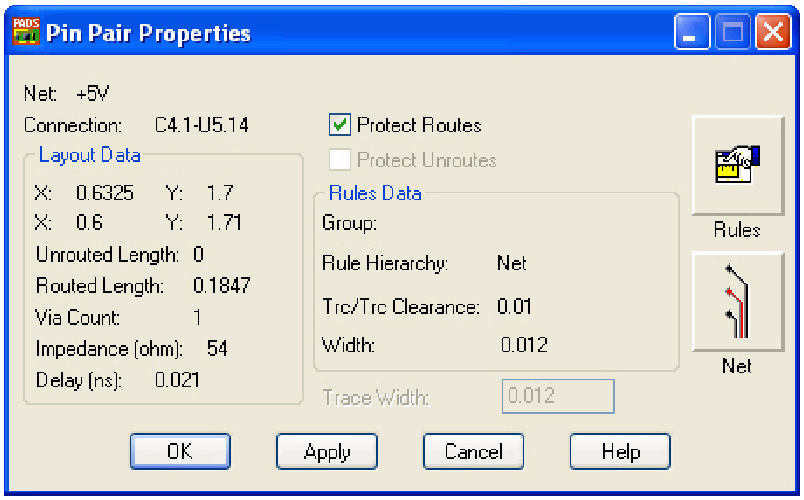


Table 45-264. Pin Pair Properties Dialog Box contents

Name	Description
Net	Displays the net name of the pin pair.
Connection	Displays the component pin connections that define the pin pair. For each connection point, the reference designator of the component is listed first, followed by a period, then the pin number.

Table 45-264. Pin Pair Properties Dialog Box contents (cont.)

Name	Description
Layout Data area	Displays all layout data about the selected pin pair. • **Coordinates**—Display the coordinates of pins that define the pin pair. • **Unrouted Length**—Displays the length of the pin pair that is unrouted. • **Routed Length**—Displays the length of the pin pair that is routed. • **Via Count**—Displays the number of vias that are used in the pin pair. • **Impedance (ohm)**—displays a baseline impedance value of the pin pair. This calculation uses the trace parameters, the board material values you set in the Layer Thickness Dialog Box, and require at least one adjacent plane to display values. A maximum of two adjacent planes are used in the calculation. If the plane layer is a split/mixed plane layer, you must have a plane shape on the layer. The location and outline of the shape and cutouts of the plane area are all ignored in the calculation. You can set a Minimum and Maximum Impedance rule in the HiSpeed Rules Dialog Box, but it is only verified when you run Tools > Verify Design after you set up a High Speed check to check Impedance. It is not verified by online DRC. • **Delay (ns)**—displays a baseline delay value of the pin pair. This calculation uses the trace parameters, the board material values you set in the Layer Thickness Dialog Box, and require at least one adjacent plane to display values. A maximum of two adjacent planes are used in the calculation. If the plane layer is a split/mixed plane layer, you must have a plane shape on the layer. The location and outline of the shape and cutouts of the plane area are all ignored in the calculation. You can set a Minimum and Maximum Delay rule in the HiSpeed Rules Dialog Box, but it is only verified when you run Tools > Verify Design after you set up a High Speed check to check Delay. It is not verified by online DRC.
Protect Routes	Protects selected routes or traces from being moved. Similar to gluing components but you can allow protected traces to be edited by PADS Router in the Routing Rules. **See also:** To Protect Routes
Protect Unroutes	Protects unrouted connections and the unrouted portions of partial routes. Protected unroutes can't be modified or routed. **See also:** To Protect Unroutes

Table 45-264. Pin Pair Properties Dialog Box contents (cont.)

Name	Description

Rules Data area	Displays some rule information for the selected pin pair.
	• **Group**—If the pin pair is included in a Group rule, the Group name is displayed.
	• **Rule Hierarchy**—Displays one of the following from lowest priority to highest priority:
	  • "Default" is shown if the pin pair is using the Default rules.
	  • "Class" is shown if the pin pair is getting its rules from a Class rule. Click the Net button to open the Net Properties and see the name of the Class to which the net belongs.
	  • "Net" is shown if the pin pair is getting its rules from the Net rules.
	  • "Group" is shown if the pin pair is getting its rules from a Group rule. The group name will be shown immediately above.
	  • Pin Pair" is shown if the pin pair has its own rules.
	• **Trc/Trc Clearance**—Displays the Trace to Trace clearance value from the Clearance Rules dialog box. See the Rule Hierarchy above to determine if the value is from the Default, Class, Net, Group or Pin Pair level rules. Conditional rules are not shown.
	Width—Displays the Recommended Trace Width from the Clearance Rules dialog box. See the Rule Hierarchy above to determine if the value is from the Default, Class, Net, Group or Pin Pair level rules.

Trace Width	Modifies the trace width. Type a new value in the box. You are restricted to the range of Trace widths you have specified in the Clearance Rules dialog box. This option is unavailable if the Protect Routes checkbox is selected.
Rules button	Opens the Pin Pair Rules dialog box with the pin pair preselected in order to apply Pin Pair level rules.
Net button	Opens the Net Properties dialog box of the net to which the pin pair belongs.

Related Topics

Modifying Pin Pair Properties

Pin Pair Rules Dialog Box

Use the Pin Pair Rules dialog box to define design rules that apply to pin pairs.

Accessing

- **Setup** menu > **Design Rules** > **Pin Pairs** button

Figure 45-295. Pin Pair Rules Dialog Box

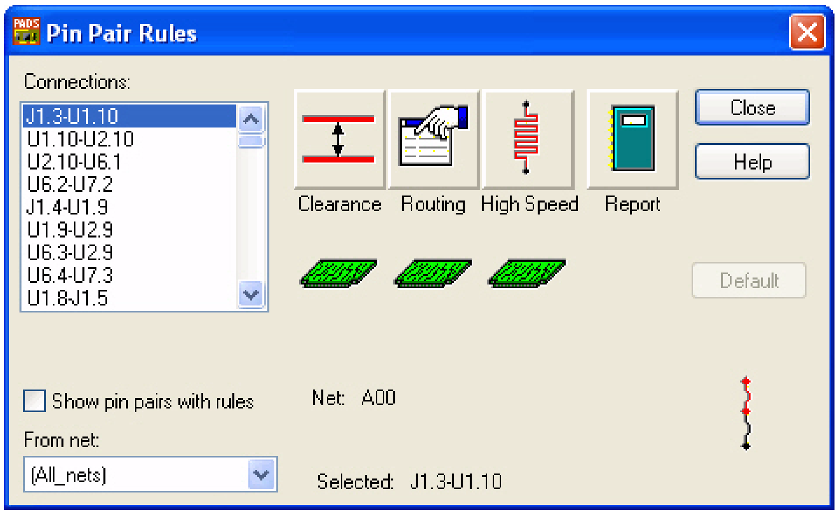


Table 45-265. Pin Pair Rules Dialog Box

Name	Description
Connections list	Lists all connections in the design.
Show pin pairs with rules	Specifies to show only pin pairs that have rules.
From net	Specifies to display pin pairs for a specific net. **Tip:** Select (All_nets) to display all pin pairs.
Clearance	Opens the Clearance Rules Dialog Box.
Routing	Opens the Routing Rules Dialog Box.
HiSpeed	Opens the HiSpeed Rules Dialog Box.
Report	Opens the Rules Report Dialog Box.
Picture below rule button	The picture below each type of rule button identifies which rules hierarchy level is used for that rule type. The meaning of the picture corresponds to the button in the Hierarchy area of the Rules dialog box. For example, if you select a class in the Class list and a green polygon appears below the Clearance button, then the default values apply to the class.

Table 45-265. Pin Pair Rules Dialog Box (cont.)

Name	Description
Net:	Lists the net(s) associated with the pin pair(s) selected in the Connections list.
Selected:	Lists the pin pair(s) selected in the Connections list.
Default	Removes non-default rules from the selected connections, so that only default rules apply.

Related Topics

Creating Pin Pair Design Rules

Modifying Pin Pair Design Rules

Resetting Pin Pair Rules to Default Rules

Design Rule Hierarchy

Pin Properties Dialog Box

The Pin Properties dialog box displays the component's reference designator, the pin number, pin type, function (pin name), swap identification, netname, and coordinates of the selected pin.

If you select another pin while the dialog box is open, the dialog box updates with information about the selected pin.

Accessing

- Select a pin > **Right-click** > **Properties**

Figure 45-296. Pin Properties Dialog Box

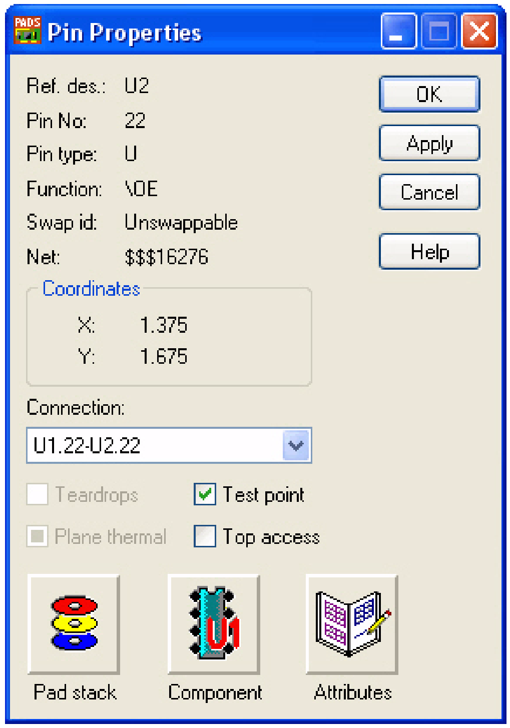


Table 45-266. Pin Properties Dialog Box contents

Name	Description
Ref. Des.	Displays the reference designator name.
Pin No	Displays the numeric or alphanumeric pin number. This is the logical pin number if a pin mapping exists in the part type, otherwise the physical pin number is displayed.
Decal Pin	Displays the decal pin number when pin mapping exists in the part type. The physical pin number is shown since the Pin No item only displays the logical pin number when there is pin mapping.
Pin type	Displays the pin type. For example, S is shown for a source pin, P is shown for a Power pin.
Function	Displays the pin name.
Connection	Lists the connection of which the pin pair is a part.
Teardrops	Turn off the Teardrops check box to remove teardrops from the selected pin. If Generate Teardrops is not turned on in the Routing / General page of the Options dialog box, this option is unavailable.

Table 45-266. Pin Properties Dialog Box contents (cont.)

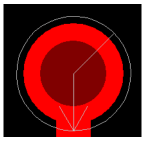
Name	Description
Plane Thermal	Determines whether the pin or via is eligible to receive a thermal. **Restriction:** This check box is unavailable if the pin is a no-connect. The status of a thermal is set individually by pin and via, and the thermal indicators will appear on plane layers only. Click to clear this check box if you do not want the via or pin to connect to any plane. Once a pin or via is eligible, it is not automatically assigned a thermal attribute. **See also:** Connecting a Net with a Plane, Setting Pins and Vias as Thermals
Test Point	Makes the via or pin a test point. **See also:** Performing a Test Point Audit This is a three-state check box that depends on the state of the selected objects. If all of the selected vias or pins are a test point, then it is on. If none of the selected vias or pins are a test point, then it is off. If some of the selected vias or pins are a test point and some are not, then it is undefined. You can make all selected vias or pins test points by turning Test Point on and choosing Apply. You can remove the test point from all selected vias or pins by turning Test Point off and choosing Apply. When you click Apply the pad stack is automatically checked to see if the via or pin can be a test point; for example, you cannot make buried vias test points because a probe cannot access a buried via. **Tip:** When the via or pin is flagged as a test point, and Show Test Points is checked on the Routing / General page of the Options dialog box, an arrow is drawn on it in the design: 

Table 45-266. Pin Properties Dialog Box contents (cont.)

Name	Description
Top Access	Attempts to probe the test point from the top and bottom in DFT Audit. The default is bottom; so with Top Access off, DFT Audit automatically tries to probe the test point from the bottom. **See also:** Performing a Test Point Audit When you click Apply, the pad stack is automatically checked to see if top access is valid; for example, you must assign Top Access to partial vias with only top access if you want to use the vias as test points. You can only set the Top Access option if the via or pin is a test point (Test Point is on).
Pad Stacks button	Opens the Pad Stacks Properties dialog box where you can modify the pad stack. If you make a change to the pad stack and the pin or via is locked, the Warning: Test Point Locked dialog box opens.
Component button	Opens the Component Properties dialog box where you can edit the component location.
Attributes button	Opens the Object Attributes dialog box and displays attribute information for the selected objects. You can view and modify nail diameter and nail number pin attributes for component pins, vias, and jumper pins, including test point attributes.

Related Topics

Modifying Pin Properties

Place Clusters Setup Dialog Box

Use the Place Clusters Setup dialog box to place the clusters based on connectivity and topology. Place Clusters treats all top-level objects with glued members as glued. Place Clusters assumes that pin positions of top-level cluster members are located in the center of the top-level cluster regardless of the glued member positions.

Accessing

- **Tools** menu > **Cluster Placement** > **Place Clusters** button > **Setup** button

Figure 45-297. Place Clusters Setup Dialog Box

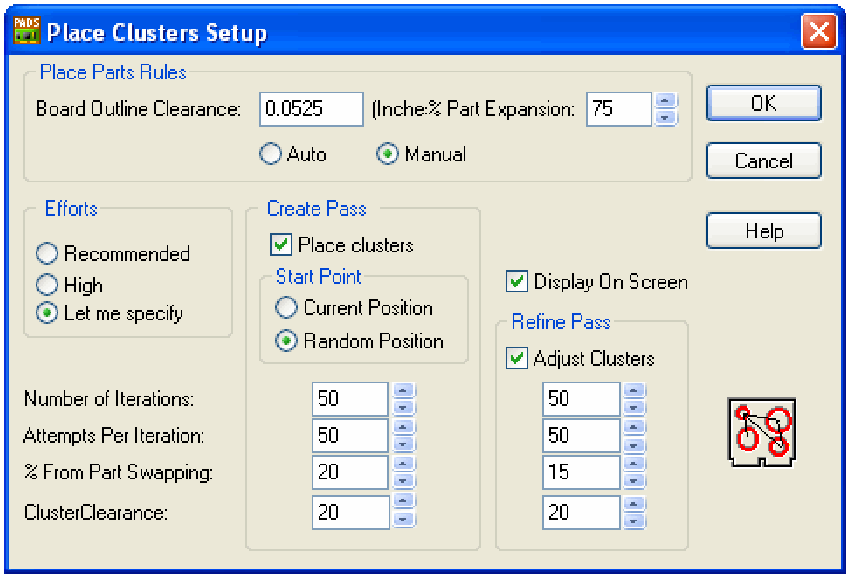


Table 45-267. Place Clusters Setup Dialog Box Contents

Name	Description
Board Outline Clearance	Determines how much clusters spread out within the board outline. This option works like Percent Part Expansion for positioning parts within the board outline. **Tip:** Place Part Rules are shared between Place Clusters Setup and Place Parts Setup; changing the value in one dialog box automatically updates the other.
Percent Part Expansion	Controls the amount of free space around parts relative to the total board area. A setting of 0% places parts as close together as possible. Set this value to 100% to place parts as far apart as possible. • **Auto**—Sets a default value of 75%. • **Manual**—Sets a user-defined value. **Tip:** Place Part Rules are shared between Place Clusters Setup and Place Parts Setup; changing the value in one dialog box automatically updates the other.
Efforts area	Adjusts the pass options in the lower half of this dialog box. • **Recommended**—Uses the default values for Number of Iterations and Attempts per Iteration. • **High**—Doubles the default values for Number of Iterations and Attempts per Iteration. • **Let Me Specify**—Enables the four Iteration and Swapping options so you can specify the values.

Table 45-267. Place Clusters Setup Dialog Box Contents (cont.)

Name	Description
Place Clusters	Enables placement operations. Clear this option to use only the Refine Pass portions of the routine for minor adjustments to already placed clusters.
Start Point area	• **Current Position**—Bases automatic placement of clusters on the part's current position. This is useful when clusters are already located in the board outline; because it tries to maintain the current position. • **Random Position**—Bases placement of clusters on a random position. This option advanced algorithms to place clusters that exist outside the board outline.
Display On Screen	Displays an outline of each part and its movement throughout the automatic placement process.
Adjust Clusters	Adjusts placed clusters using different values for number of iterations, attempts, swapping, and clearance values set in the Efforts area. To activate, click Let Me Specify.
Number of Iterations	Specifies the number of placement passes to make. The default value accommodates most average designs. Use a lower number for small designs and faster processing. Use a higher number for large or dense designs.
Attempts per Iteration	Within each iteration, attempts are made to position parts, minimize net lengths, reduce part overlaps, and keep parts inside the board outline. Increase this value to group parts closer together and closer to glued parts during an iteration.
Percent from Part Swapping	During each iteration, parts, clusters, and unions are either swapped with other parts or repositioned to improve placement. Use this feature to increase the amount of swapping that occurs instead of moving parts.
Component Clearance	Determines the distance that clusters spread out within the board outline. This option works like the Percent Part Expansion option for positioning parts within the board outline.

Related Topics

Using the Cluster Placement Dialog Box

Cluster Placement

Place Parts Setup Dialog Box

Use the Place Parts Setup dialog box to control how parts are handled during Automatic Placement.

Accessing

- **Tools** menu > **Cluster Placement** > **Place Clusters** button > **Setup** button

Figure 45-298. Place Parts Setup Dialog Box

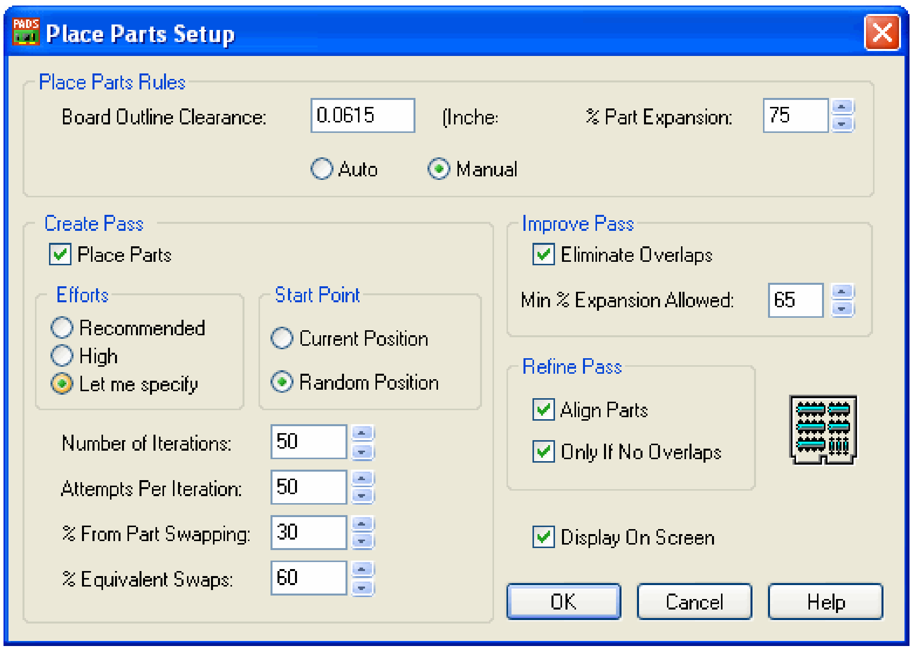


Table 45-268. Place Parts Setup Dialog Box Contents

Name	Description
Board Outline Clearance	Determines how much clusters spread out within the board outline. This option works like Percent Part Expansion for positioning parts within the board outline. **Tip:** Place Part Rules are shared between Place Clusters Setup and Place Parts Setup; changing the value in one dialog box automatically updates the other.

Table 45-268. Place Parts Setup Dialog Box Contents (cont.)

Name	Description
Percent Part Expansion	Controls the amount of free space around parts relative to the total board area. A setting of 0% places parts as close together as possible. Set this value to 100% to place parts as far apart as possible. • **Auto**—Sets a default value of 75%. • **Manual**—Sets a user-defined value. **Tip:** Place Part Rules are shared between Place Clusters Setup and Place Parts Setup; changing the value in one dialog box automatically updates the other.
Place Parts	Enables part placement operations. Clear this option to use only the Improve Pass portions of the routine for overlap and alignment operations.
Efforts area	Adjusts the pass options in the lower half of this dialog box. • **Recommended**—Uses the default values for Number of Iterations and Attempts per Iteration. • **High**—Doubles the default values for Number of Iterations and Attempts per Iteration. • **Let Me Specify**—Enables the four Iteration and Swapping options so you can specify the values.
Start Point area	• **Current Position**—Bases automatic placement of unglued parts on the part's current position. This is useful when parts are already located in the board outline; because the current position is maintained when possible. • **Random Position**—Bases placement of parts on a random position. This uses advanced placement algorithms to create new placements.
Number of Iterations	Specifies the number of placement passes to make. The default value accommodates most average designs. Use a lower number for small designs and faster processing. Use a higher number for large or dense designs.
Attempts per Iteration	Within each iteration, attempts are made to position parts, minimize net lengths, reduce part overlaps, and keep parts inside the board outline. Increase this value to group parts closer together and closer to glued parts during an iteration.
Percent from Part Swapping	During each iteration, parts, clusters, and unions are either swapped with other parts or repositioned to improve placement. Use this feature to increase the amount of swapping that occurs instead of moving parts.
Percent Equivalent Swaps	During part swapping, either equivalent or non equivalent swaps are made. An equivalent swap would be between two parts (or unions) that have the same extents. Setting this value to 100% to swap only equivalent parts.

Table 45-268. Place Parts Setup Dialog Box Contents (cont.)

Name	Description
Eliminate Overlaps	Moves overlapping parts without violating the Percentage Part Expansion setting. The Percentage Part Expansion setting is lowered if necessary to eliminate overlaps, but will not exceed the setting for Minimum Percentage Expansion Allowed.
Minimum Percentage Expansion Allowed	Sets the lowest Percentage Part Expansion setting allowed during the eliminate overlaps pass.
Align Parts	Enables an alignment pass that adjusts neighboring parts to improve routability.
Only if No Overlaps	Prohibits the alignment pass if overlaps are present.
Display On Screen	Displays an outline of each part and its movement throughout the automatic placement process.

Related Topics

Using the Cluster Placement Dialog Box

Cluster Placement

Plane Layer Nets Dialog Box

Use the Plane Layer Nets dialog box to assign or unassign nets with plane layers. CAM (Negative) planes should only have one net assigned. Split/Mixed may have multiple nets assigned. Plane layers contain large copper areas which components can access for ground and power. CAM output produces thermal relief connection pads for nets associated with the plane layer.

Accessing

- **Setup** menu > **Layer Definition** > select a plane layer **>** **Assign Nets** button

Figure 45-299. Plane Layer Nets Dialog Box

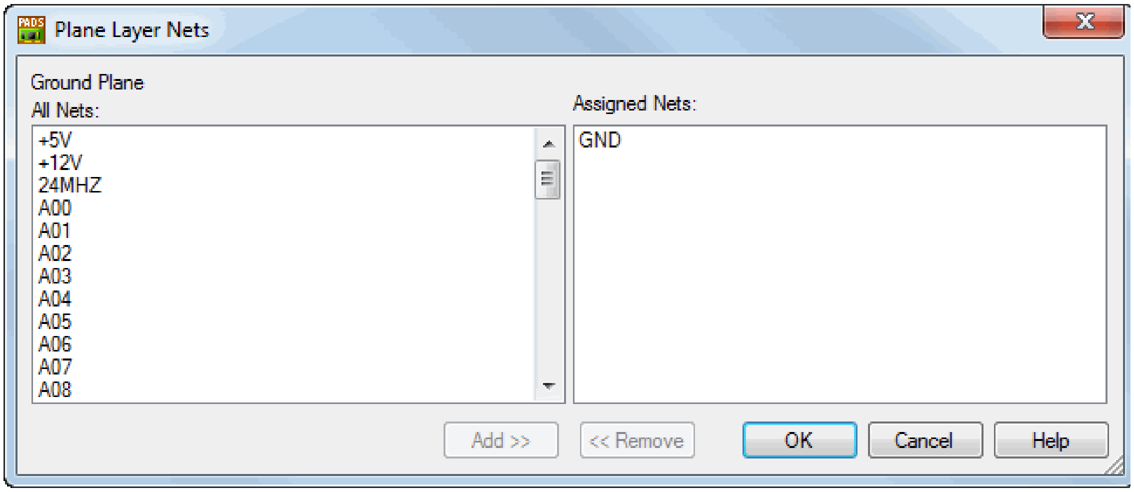


Table 45-269. Plane Layer Nets Dialog Box Contents

Name	Description
All Nets	Lists all nets in the design which you can associate to the plane area.
Assigned Nets	Lists the nets associated with the plane.
Add >>	Adds the net selected in the All Nets list to the Assigned Nets list, thus associating the net with the plane.
<< Remove	Removes the net selected in the Assigned Nets list to the All Nets list, thus breaking the association between the net and the plane.

Related Topics

Setting Up an Outer Layer

Setting Up an Inner Layer

Plot Options Dialog Box

Use the Plot Options dialog box to set plotting options. You can also use this dialog box to gain access to the Drill Drawing Options.

Use the Plot Positioning settings to create the best view of your design on the page that you print or pen plot. But more importantly, use the Plot Positioning settings to ensure that all the documents of the layers of your design align perfectly with one another. The most efficient way to correctly align your layers involves using board or global fiducials and using the two-adjacent-sides types of plot justification.

Use the Preview window to help determine the correct positioning with regard to your output - whether a file or a page.

Tip: Jumper pins of SMD jumpers are output on the paste mask layer, similar to SMD component output.

Accessing

- **File** menu > **CAM** > **Add** button > **Options** button

  or

- **File** menu > **CAM** > select a document name > **Edit** button > **Options** button

Exception: If you selected NC Drill as the Document Type, the NC Drill Options dialog box opens instead.

Figure 45-300. Plot Options Dialog Box

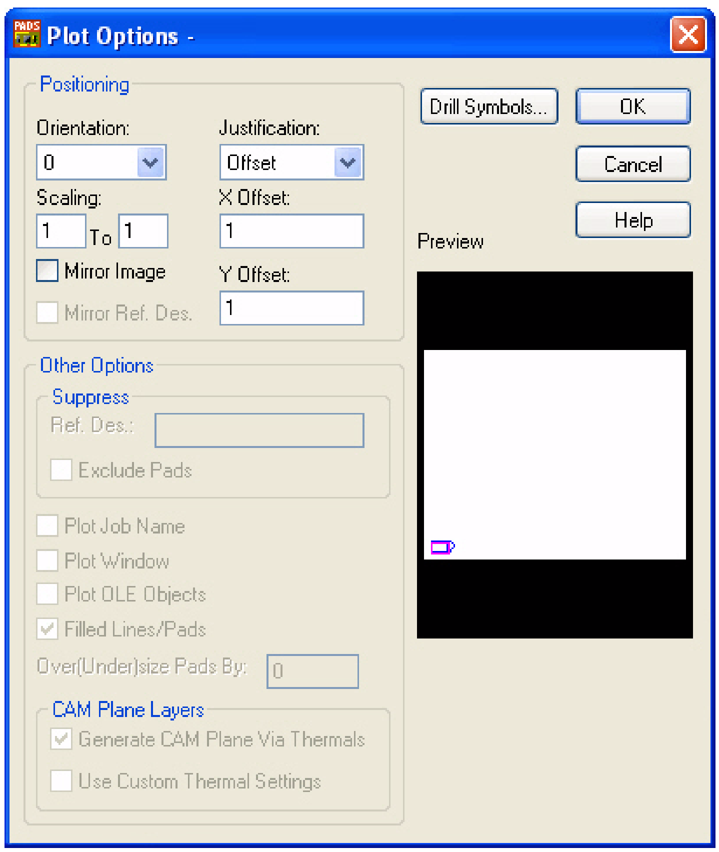


Table 45-270. Plot Options Dialog Box contents

Name	Description
Orientation list	Sets the orientation angle of the design. Choose from 0, 90, 180 or 270 degrees rotation.
Justification list	Specifies the justification for the plot. Edge style/corner justifications use the outermost object and are best used in conjunction with board/global fiducials. Choose from: • **Centered**—Centers the design within the output media boundaries. • **Bottom Left**—Aligns output to the bottom left corner of the output media. • **Bottom Right**—Aligns output to the bottom right corner of the output media. • **Top Left**—Aligns output to the top left corner of the output media. • **Top Right**—Align output to the top right corner of the output media. • **Offset**—Aligns output origin to the x and y offsets. • **Scale to Fit**—Scale to Fit automatically adjusts the scaling so that the entire output is visible within the output media boundaries.
Scaling	Defines the plot-size to actual- size ratio. **Tip:** Two to one (2:1) scaling results in a plot that is twice the actual size.
X/Y Offset	Defines margin offsets which lets you move the design within the output media boundaries. The values used for the offset place the origin of the design at that point. **Restriction:** Unavailable for Centered and Scale to Fit justifications.
Mirror Image	Specifies to mirror the image.
Mirror Ref. Des.	Specifies to mirror reference designators.
Suppress Ref. Des	Specifies the reference designators you want to suppress. Type the prefixes of all reference designators, a specific reference designator, or specify a range of reference designators, separated by commas. Use use a tilde to suppress a range of designators. Do not add spaces between listings. For example, J,U1,R2~5.
Exclude Pads	Specifies to exclude the parts represented by the reference designators specified in the Ref. Des. box from the CAM document. **Restriction:** This check box is only available for Solder Mask and Paste Mask outputs.

Table 45-270. Plot Options Dialog Box contents (cont.)

Name	Description
Plot Job Name	Specifies to plot the .pcb name, time, and date on the plot. **Restriction:** This option is unavailable if the output device is a photo plotter.
Plot Window	Specifies to plot only the current view in the Layout Editor. Clear this option to plot the entire design. **Tip:** Pad flashes only appear in the plot output if the flash center is within the layout window.
Plot OLE Objects	Specifies to plot OLE objects. This option is only available when you print the output. You cannot photoplot or pen plot OLE objects. **Restriction:** You cannot select this option unless the orientation is set to zero. **Tip:** OLE objects do not appear in the CAM Preview dialog box, but they are used in the calculation when the plot location is determined.
Filled Lines/Pads	Specifies to plot filled lines and pads. Drawing times are faster when lines and pads are unfilled.
Over(under)size Pads By	Creates global oversized or undersized pads and vias with positive or negative values. See also: Control of Solder Mask and Paste Mask **Restriction:** Beginning with PADS2007, the Over(Under)size value only applies to electrical layers. To apply the value to all layers, see Applying the Over(Under)size Value to All Layers. **Tips:** • In plane plots, this option also creates clearances. A positive value increases the plotted pad size; a negative value decreases the plotted pad size. • You can also use this option to oversize or undersize a pad or thermal shape on a plane layer. When defining pads or thermals using this option, you cannot define different undersize or oversize amounts per pad, but you gain design time because you do not have to set size or layer requirements in the library or pad stacks.
Generate CAM Plane Via Thermals	Creates thermals for vias associated with the plane net. If this option is cleared, thermals are not created, and solid connections to all plane net associated vias are output. This option is available only for CAM plane plots.

Table 45-270. Plot Options Dialog Box contents (cont.)

Name	Description
Use Custom Thermal Settings	Specifies to use thermal and aperture settings defined in the Pad Stacks Properties dialog box. If you disable this setting, CAM plane thermals are generated according to a set of parameters. See Interpreting CAM Plane Thermal Graphics for more information. **Requirement:** CAM planes using custom antipads defined in the Pad Stacks require that you select the Remove Unused Pads check box in the Split/Mixed Planes tab of Options. Without this setting enabled, default antipad settings will be used instead. This option is available only for CAM plane plots and only when RS-274-X is selected as the photo plotter output format. Custom thermals, by default, use the pad size plus the clearance to define thermal spokes, whereas default CAM plane thermals use the pad size for the outer width of the thermal spokes. When brought into CAM350, these shapes are too large. Modify these pads accordingly for CAM350. **Restriction:** Custom thermals for CAM plane layers do not work with the RS-274-D photo plotter output format selected.
Drill Symbols button	Opens the Drill Drawing Options dialog box where you can define the drill symbols you want to use and set up the drill chart. **Restriction:** The Drill Symbols button is only available when you select the Drill Drawing document type.
Preview area	Displays the document. The graphic in the Preview area updates as you make changes to the options. The **Preview** window displays the positioning of the design with regard to the plot area or printed page. The blue outline shows the board outline of your design with respect to the plot area. When creating a drill drawing, the preview area will also display the position of the Drill Chart (if enabled) with a magenta-colored rectangular outline. By default, the drill drawing will be positioned with a corner at the design origin. You may need to reposition the drill chart to keep from overlapping your design. See Creating a Drill Drawing with Drill Table for more information.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Adding or Editing CAM Documents

Pour Manager Dialog Box, Flood Tab

Use the Flood tab in the Pour Manager dialog box to fill the area inside copper pour outlines with copper, represented by a hatched grid. Flood creates isolation areas around copper, traces, and pads that are inside the copper pour outline but are not part of the same net. It also creates thermal relief connections around pins that belong to the same net.

Flood floods over instances where a drill size is larger than the pad to which it is assigned. By flooding over the drill and pads, the pad remains connected to the plane. So if you flood around pads (intending them to get a thermal relief) and these pads have a drill size that is larger than the pad size, the pad and drill are entirely flooded over. A connectivity check detects this condition.

Accessing

- **Tools** menu > **Pour Manager** > **Flood** tab

Figure 45-301. Flood tab

Table 45-271. Flood tab contents

Name	Description
Flood Mode Area	• **Flood All**—Floods all copper pour outlines. • **Fast Flood**—Floods copper pour that has not previously been flooded; in other words, only newly added outlines.
Confirm Flood Operations	Enables confirmations for flood operations. When this check box is on and you click Start, the “Ok to connect plane(s)?” message appears. This message lets you proceed with or cancel flood operations.
Start button	Starts the flood, hatch, or connection process.
Setup button	Opens the Thermals tab of the Options dialog box. Use this tab to set the size and shape of thermals.

Related Topics

Flooding a Copper Pour Area

Pour Manager Dialog Box, Hatch Tab

Pour Manager Dialog Box, Plane Connect Tab

Verify the Design

Pour Manager Dialog Box, Hatch Tab

Use the Hatch tab in the Pour Manager dialog box to fill in the hatch lines of previously flooded copper pour areas. You should perform a hatch after manually editing a previously flooded copper pour area. Because the hatch lines are not stored, you should also perform a hatch after opening a file.

Tip: Unlike the Flood tab, the Hatch command does not create new isolation areas or thermal connections. If you've moved objects within the pour area or changed thermal settings, you need to change the hatch outline to a pour outline and flood the area to generate the new hatch. Use the Flood tab instead.

Accessing

- **Tools** menu > **Pour Manager** > **Hatch** tab

Figure 45-302. Hatch tab

Table 45-272. Hatch tab contents

Name	Description
Hatch Mode Area	• **Hatch All**—Restores hatch to areas that are not already hatched, to hatch islands that have been edited since the last hatch operation, and to all areas when a design is opened. • **Fast Hatch**—Restores only the most recently effected hatch outline.

Table 45-272. Hatch tab contents (cont.)

Name	Description
Start button	Starts the flood, hatch, or connection process.
Setup button	Opens the Drafting / Hatch and Flood page of the Options dialog box. Use this page to set hatching and flooding options for drafting.

Related Topics

Hatching a Copper Pour Area

Pour Manager Dialog Box, Flood Tab

Pour Manager Dialog Box, Plane Connect Tab

Verify the Design

Pour Manager Dialog Box, Plane Connect Tab

Use the Plane Connect tab in the Pour Manager dialog box to create connections to a split/mixed plane.

Accessing

- **Tools** menu > **Pour Manager** > **Plane Connect** tab

Figure 45-303. Plane Connect tab

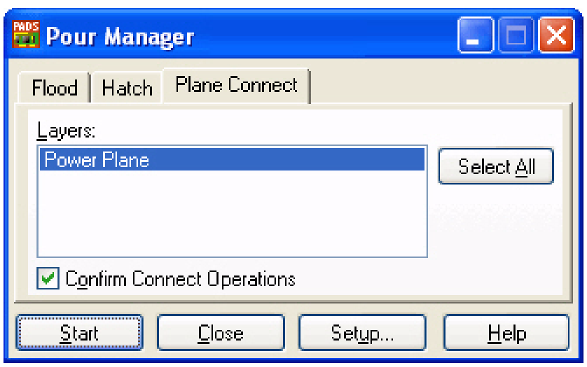


Table 45-273. Plane Connect tab contents

Name	Description
Layers list	Displays all layers identified as Split/Mixed in the Setup Layers dialog box. Click a layer to connect.
Select All button	Selects all split/mixed plane layers listed in the Layers list.

Table 45-273. Plane Connect tab contents (cont.)

Name	Description
Confirm Connect Operations	Enables confirmations for connect operations. When this check box in on and you click start, the "Ok to connect plane(s)?" message appears. This message lets you proceed with or cancel flood operations.
Start button	Starts the flood, hatch, or connection process.
Setup button	Opens the Split/Mixed Plane page of the Options dialog box. Use this tab to set the size and shape of thermals.

Related Topics

Flooding a Plane Area

Pour Manager Dialog Box, Hatch Tab

Pour Manager Dialog Box, Flood Tab

Process Indicator Dialog Box, Backward Annotation

Use the Process Indicator dialog box to monitor the annotation progress, display the reports, or cancel the annotation process being run by the Backward from PCB process of the DxDesigner Link.

Accessing

- Backward Annotation dialog box > click **OK**.

Figure 45-304. Process Indicator Dialog Box, Backward Annotation

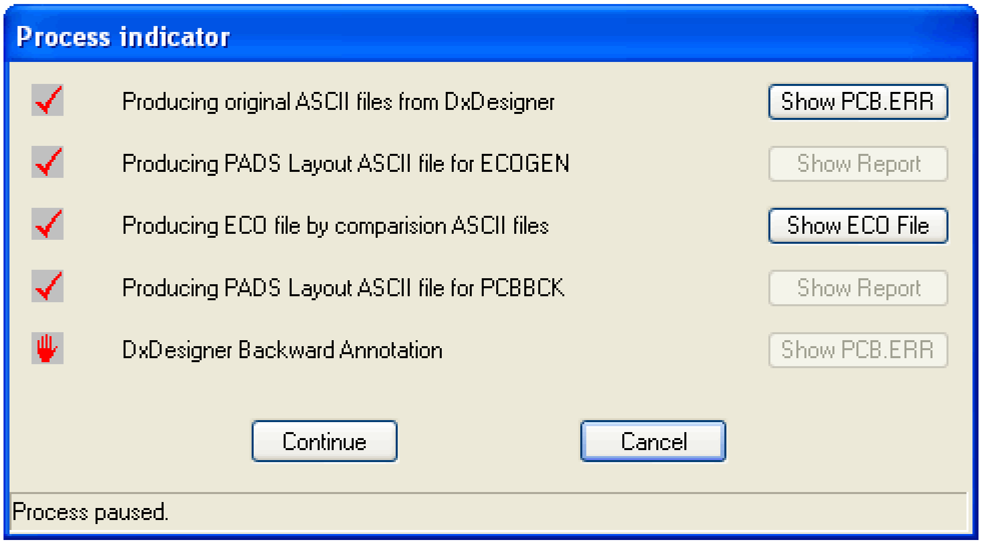


Table 45-274. Process Indicator Dialog Box, Backward Annotation Contents

Name	Description
Producing original ASCII files from DxDesigner	Displays the status of creating all the netlisting files from DxDesigner
Producing PADS Layout ASCII file for ECOGEN	Displays the status of creating the PADS Layout ASCII file used to compare with the schematic netlist to generate the .eco file.
Producing ECO file by comparing ASCII files	Displays the status of generating the .eco file from comparing the ASCII file from both DxDesigner and PADS Layout.
Producing PADS Layout ASCII file for PCBBCK	Displays the status of generating the .asc file required by the DxDesigner backward annotation.
DxDesigner Backward Annotation	Displays the status of backward annotating the .eco and .asc file into DxDesigner. **Restriction:** This Show Report button is only available after the process has run. If you've chosen to pause before this process, you will need to click Continue to finish the process.

Table 45-274. Process Indicator Dialog Box, Backward Annotation Contents

Name	Description
Close/Continue button	This is a multi-function button. Depending on the state of the annotation process, it displays either Close or Continue. It displays Close if you click Cancel during the annotation process, or if the annotation process is complete. It displays Continue if you've decided to pause before the final process - backward annotating the .eco and .asc file.

Process Indicator Dialog Box, Forward Annotation

Use the Process Indicator dialog box to monitor the annotation progress, display the reports, or cancel the annotation process being run by the Forward to PCB—Update PCB process of the DxDesigner Link.

Accessing

- Forward Annotation dialog box > click **OK**.

Figure 45-305. Process Indicator Dialog Box, Forward Annotation

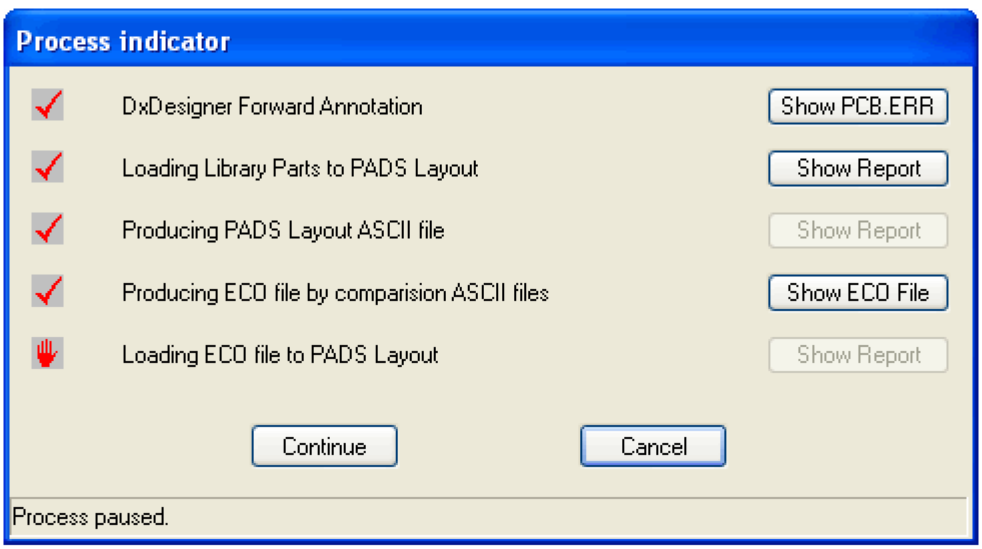


Table 45-275. Process Indicator Dialog Box, Forward Annotation Contents

Name	Description
DxDesigner Forward Annotation	Displays the status of creating all the netlisting files from DxDesigner
Loading Library Parts to PADS Layout	Displays the status of importing the .p file into the library.
Producing PADS Layout ASCII file	Displays the status of creating the PADS Layout ASCII file used to compare with the schematic netlist to generate the .eco file.
Producing ECO file by comparison ASCII files	Displays the status of generating the .eco file from comparing the ASCII file from both DxDesigner and PADS Layout.
Loading ECO file to PADS Layout	Displays the status of importing the .eco file into PADS Layout. **Restriction:** This Show Report button is only available after the process has run. If you've chosen to pause before this process, you will need to click Continue to finish the process.
Close/Continue button	This is a multi-function button. Depending on the state of the annotation process, it displays either Close or Continue. It displays Close if you click Cancel during the annotation process, or if the annotation process is complete. It displays Continue if you've decided to pause before the final process - importing the .eco file.

Process Indicator Dialog Box, Create PCB

Use the Process Indicator dialog box to monitor the annotation progress, display the reports, or cancel the annotation process being run by the Forward to PCB—Create PCB process of the DxDesigner Link.

Accessing

- Forward Annotation dialog box > click **OK**.

Figure 45-306. Process Indicator Dialog Box, Create PCB

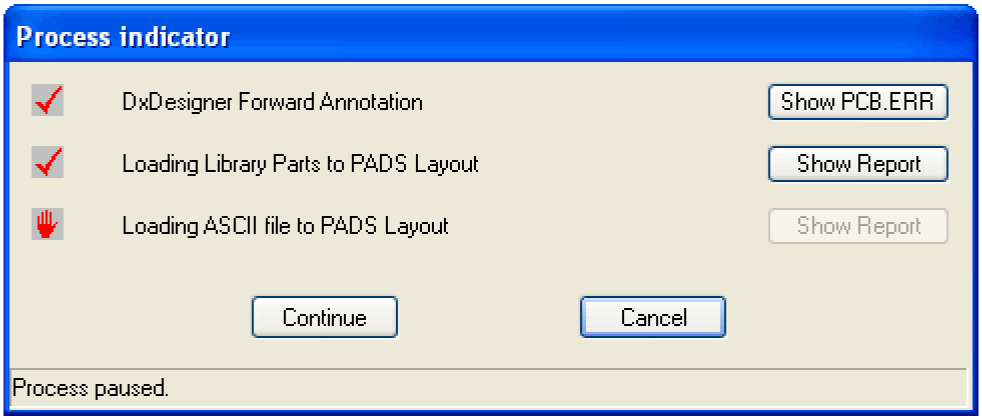


Table 45-276. Process Indicator Dialog Box, Create PCB Contents

Name	Description
DxDesigner Forward Annotation	Displays the status of creating all the netlisting files from DxDesigner.
Loading Library Parts to PADS Layout	Displays the status of importing the .p file into the library.
Loading ASCII file to PADS Layout	Displays the status of importing the .asc file into PADS Layout. **Restriction:** This Show Report button is only available after the process has run. If you've chosen to pause before this process, you will need to click Continue to finish the process.
Close/Continue button	This is a multi-function button. Depending on the state of the annotation process, it displays either Close or Continue. It displays Close if you click Cancel during the annotation process, or if the annotation process is complete. It displays Continue if you've decided to pause before the final process - importing the .asc file.

Process Status Dialog Box

Use the Process Status dialog box to monitor the .eco (and .asc) generation progress, display the reports, or cancel the process being run by the Compare/ECO process when creating a .eco file.

Accessing

- Compare/ECO dialog box > select Generate ECO File check box > click **Run**.

Figure 45-307. Process Status Dialog Box

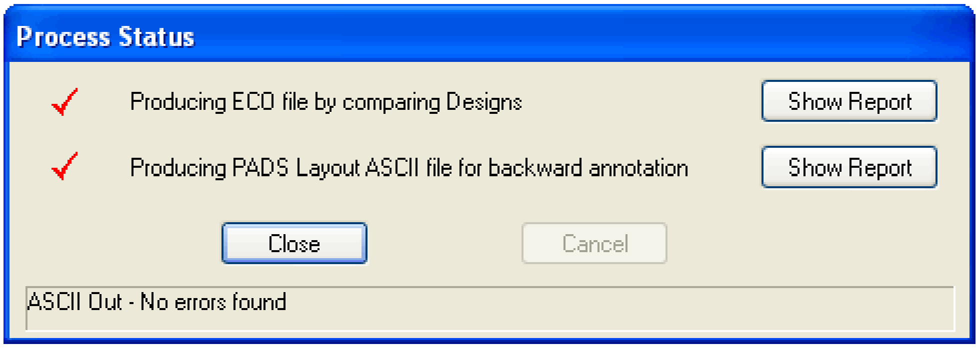


Table 45-277. Process Status Dialog Box Contents

Name	Description
Producing ECO file by comparing Designs	Displays the status of generating the .eco by comparing designs. Click Show Report to display the .eco file and the .log file.
Producing PADS Layout ASCII file for backward annotation	Displays the status of exporting the .asc file used in backward annotation to DxDesigner. **Restriction:** This item is only shown when performing back annotation and when you select the Generate ASCII File for Back Annotation to Schematic check box. Click Show Report to display the Layout.err file.

Project Explorer

The Project Explorer shows a hierarchical structure for the objects in your design. It provides access to objects and rules. When you update your design, the hierarchical structure is automatically updated to reflect the changes you make.

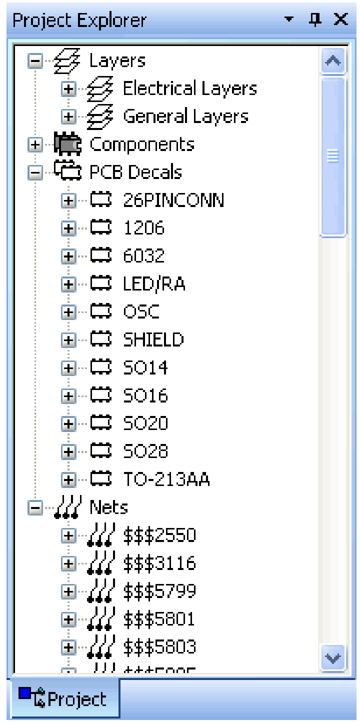
Accessing

- Click the **Project Explorer** button.

Tip: The Hierarchical structure is available only when a design is open.

Restriction: The Project Explorer is not available in the PCB Decal Editor.

Figure 45-308. Project Explorer



Object Types

Objects in the Project Explorer are placed in object groups. Object groups are of two types: primary and secondary.

Restrictions:

- You cannot remove or rename primary object groups.
- Modification of secondary group items is only available in PADS Router.

The following table lists and describes the primary and secondary object groups.

Table 45-278. Object Groups and Subgroups

Primary Group	Product Availability	Secondary Group	Description
schematic sheets	PADS Logic	Sheet names	Lists all parts on the sheet
Layers	PADS Layout PADS Router	Electrical layers	Lists all electrical layers, including plane layers and routing layers
		General layers	Lists all other layers except electrical

Table 45-278. Object Groups and Subgroups (cont.)

Primary Group	Product Availability	Secondary Group	Description
Components	PADS Logic PADS Layout PADS Router		Lists all components and pin pairs
Part decals/ PCB decals	PADS Logic PADS Layout PADS Router		Lists all part decals in the design or all components that use the selected part decal
Nets	PADS Logic PADS Layout		Lists all nets in the design
Net objects	PADS Router	Net classes	Lists all nets belonging to net classes
		Matched length net groups	Lists all matched length net groups
		Nets	Lists all nets in the design
		Matched length pin pair groups	Lists all matched length pin pair groups
		Pin pair groups	Lists all nets belonging to pin pair groups (containing pin pair rules)
		Conditional rules	Lists all nets with conditional rules
		Differential pairs	Lists all differential pairs
Via types	PADS Router		Lists the via types used in the design
CAE decals	PADS Logic		Lists the CAE decals used in the design
PCB decals	PADS Logic PADS Layout		Lists the PCB decals used in the design

Related Topics

Selecting Objects

Zooming to Selection

Project and Folder Properties Dialog Boxes

Use the Project Properties dialog box and the Folder Properties dialog box to view and edit the properties of an Archive Navigator project or folder.

Accessing

- In the **Vault** view, right-click a project or folder, and click **Properties.**

Figure 45-309. Project Properties Dialog Box

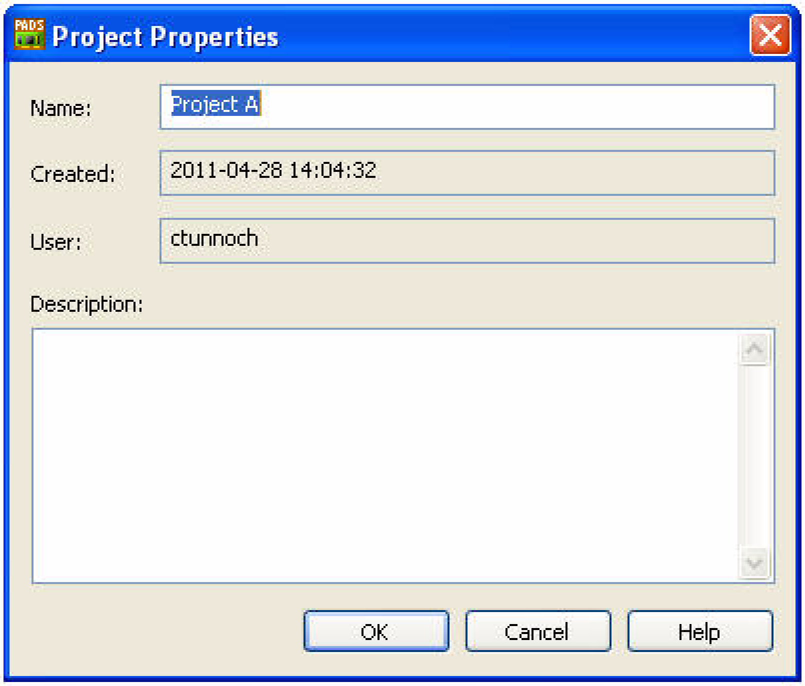


Table 45-279. Project/Folder Properties Dialog Box Contents

Name	Description
Name	Specifies the project/folder name. **Tip:** Create a name you can search for with the Find in Vault tool.
Created	The date/time when the project/folder was created
User	The user who created the project/folder
Description	Specifies a description attribute for the projec/folder. **Tip:** Create a name you can search for with the Find in Vault tool.

Related Topics

Adding a Project Container to the Vault

Radial Move Setup Dialog Box

Use the Radial Move Setup dialog box to set up the polar grid origin for radial moves, radial move options, and other information related to moving objects radially.

Accessing

- **Tools menu** > **Options** > **Grids** tab > **Radial Move Setup button**

Figure 45-310. Radial Move Setup Dialog Box

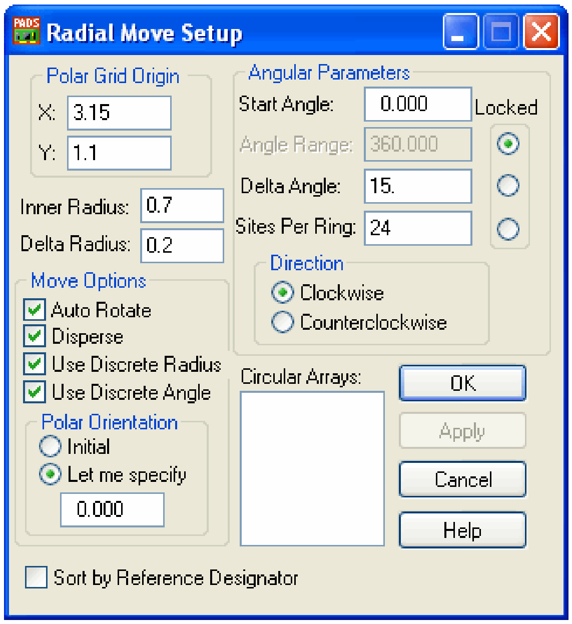


Table 45-280. Radial Move Setup Dialog Box

Name	Description
Polar Grid Origin Area	Sets the x-coordinate and y-coordinate for the polar grid origin in current design units.
Inner Radius	Sets the radius of the inner ring of the polar grid or circular array in current design units. You cannot use zero or negative values.
Delta Radius	Sets the radial distance between neighboring rings of the polar grid or circular array in current design units. For Radial Move, zero sets the Delta Radius equal to the Inner Radius. You cannot use negative values.
Auto Rotate	Automatically adjusts the orientation of the selected objects on the grid.

Table 45-280. Radial Move Setup Dialog Box (cont.)

Name	Description
Disperse	Arranges all of the selected objects on grid sites without overlapping. When Disperse is selected, the angular distance between neighboring objects is equal to or greater than the Delta Angle value. Disperse is useful for initial placing of a group of objects on the grid. Also, when Disperse is selected, text and reference designators are sorted alphabetically. In the Decal Editor, Disperse allows intersecting objects. Terminals are sorted by increasing pin number.
Use Discrete Radius	Sets the mode of radial displacement: smooth or discrete. Discrete snaps to points along the polar grid.
Use Discrete Angle	Sets the mode of angular displacement: smooth or discrete. Discrete snaps to points along the polar grid.
Polar Orientation area	Sets the orientation of all selected objects. • **Initial** — Retains the individual orientation values of the selected objects from their starting positions. • **Let Me Specify** — Assigns the same polar orientation to the selected objects. Type the orientation in the box.
Sort by Reference Designator	Sorts selected items by reference designator when creating or modifying an array or during a Radial Move. When this option is cleared, items are sorted by the order in which they were selected.
Start Angle	Sets the polar angle, in degrees, of the first grid or circular array site. You can type a value between 0.000 and 359.999.
Angle Range	Sets the range within which you want to place objects. 360 sets a full circle grid or array; smaller values set sector-shaped grids or arrays.
Delta Angle	Sets the angular distance between neighboring sites within a ring.
Sites Per Ring	Sets the number of sites for each ring of the grid or array. Type a value equal to or greater than 2. You cannot use zero or negative values.

Table 45-280. Radial Move Setup Dialog Box (cont.)

Name	Description
Locked	Controls the automatic adjustment of Angle Range, Delta Angle, and Sites Per Ring for Radial Move. Controls the automatic adjustment of Count, Angle, and Angle Range for Polar Step and Repeat. The three settings above, Angle Range, Delta Angle, and Sites per Ring, are interdependent; each value depends on the values in the other two. Set one of the values and lock it. Set one of the unlocked values; the other unlocked value automatically updates. For example, if you set Angle Range to 360 and Sites Per Ring to 36, Delta Angle updates to 10.
Direction	Sets how the sites are placed on the grid or circular array: Clockwise or Counterclockwise.
Circular Arrays	Lists all circular arrays and unions in the design. If you double-click an item from the list, the values in that array or union appear in the dialog box. Create and name circular arrays using Create Array. Create and name unions using Create Union. **Tip:** You can only use this option in the Layout Editor.

Related Topics

To Set Up a Polar Grid

To Use Radial Move

Defining Arrays

Reassign Electrical Layers Dialog Box

Use the Reassign Electrical Layers Dialog box to move data from one electrical layer to another.

Accessing

- **Setup** menu > **Layer Definition** > **Reassign** button

Figure 45-311. Reassign Electrical Layers Dialog Box

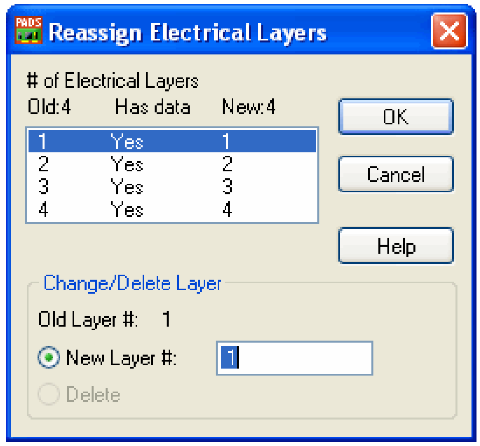


Table 45-281. Reassign Electrical Layers Dialog Box Contents

Name	Description
# of Electrical Layers	• **Old Layer Number**—The layer on which the data exists before you reassign the layer. • **Has Data**—Indicates whether or not data exists on the selected layer. • **New**—Displays the new layer number after reassignment. If you delete a layer, this column displays <delete> in that row.
Old Layer #	The number of the old layer selected in the # of Electrical Layers list.
New Layer #	Displays the layer where the data exists after you reassign the layer.
Delete	This is enabled when you decrease the number of layers and the selected layer has no data. For each layer you delete, <delete> appears in that layer's row under the New column.

Related Topics

To Reassign Electrical Layers

Rename Net Dialog Box

Use the Rename Net dialog box to change the name of a net. This changes the netlist, is considered an engineering change and requires ECO mode.

Restriction

A protected route or physical design reuse element can prevent you from renaming a net in ECO mode. You must unprotect the route or break the reuse to proceed. If you are recording changes to an ECO file, the action of removing the protection on traces or breaking out reuse elements is not saved.

Accessing

- select a net > **ECO Toolbar** > **Rename Net**

Figure 45-312. Rename Net Dialog Box

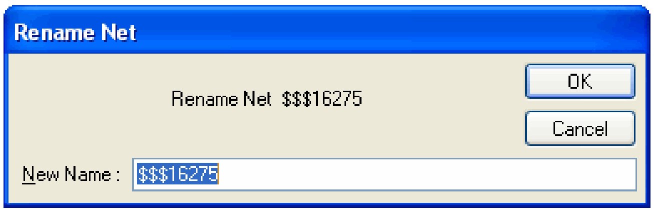


Table 45-282. Rename Net Dialog Box Contents

Name	Description
New Name	Type a new name. **Restriction:** The maximum netname length is 47 characters. **See also:** Illegal Characters in Netnames and Part Names

Related Topics

Renaming a Net in ECO Mode

Renumber Pins Dialog Box

Use the Renumber Pins dialog box to renumber pins (terminals). You can renumber decal pins from the PCB Decal Editor, or part type pins from the Pins tab of the Part Information dialog box.

Accessing

- From the PCB Decal Editor:

  PCB Decal Editor > select a starting terminal > right-click > **Renumber Terminals**

- From the Part Information dialog box:

  Part Information dialog box **>** **Pins** tab > select pin(s) > **Renumber** button

Figure 45-313. Renumber Pins Dialog Box

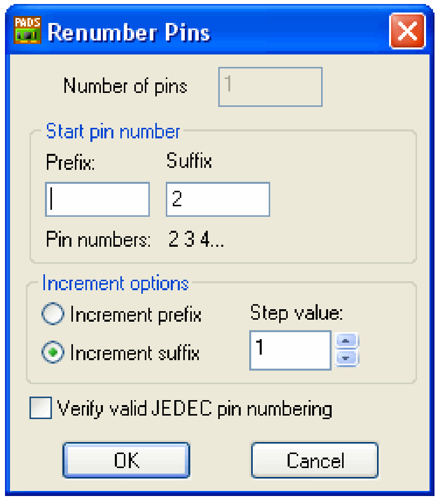


Table 45-283. Renumber Pins Dialog Box Contents

Name	Description
Number of pins	The number of pins available for renumbering.
Prefix/Suffix	For a single pin number, use either Prefix or Suffix box, and void the other box. Use both boxes if you want to increment one of the values. Alphabetic and numeric values can be used in either box.
Pin numbers	A preview of pin numbers based on your prefix/suffix input.
Increment prefix	Sets the prefix as the part of the pin number to increment.
Increment suffix	Sets the suffix as the part of the pin number to increment.
Step value	Sets the step value. Type a positive or negative number by which to increase or decrease the pin number with consecutive or stepped values. **Restriction:** Step value must be non-zero and be in the range -10 to +10. Zero would replicate a single pin number and is not allowed.
Verify valid JEDEC pin numbering	If using alphanumerics, you can select the **Verify valid JEDEC pin numbering** check box to ensure that legal alphanumeric values are used. **Tip:** This option only ensures that legal alpha and numeric combinations are used. To arrange rows and columns according to JEDEC, use the Assign JEDEC Pinning option on the Tools Menu.

Related Topics

Renumbering a Terminal

Report Manager Dialog Box

Use the Report Manager dialog box to generate a report about the parts in a library. You can specify the parts and the attributes to include in the report.

Accessing

- **File** menu > **Library** > **List to File** button

Figure 45-314. Report Manager Dialog Box

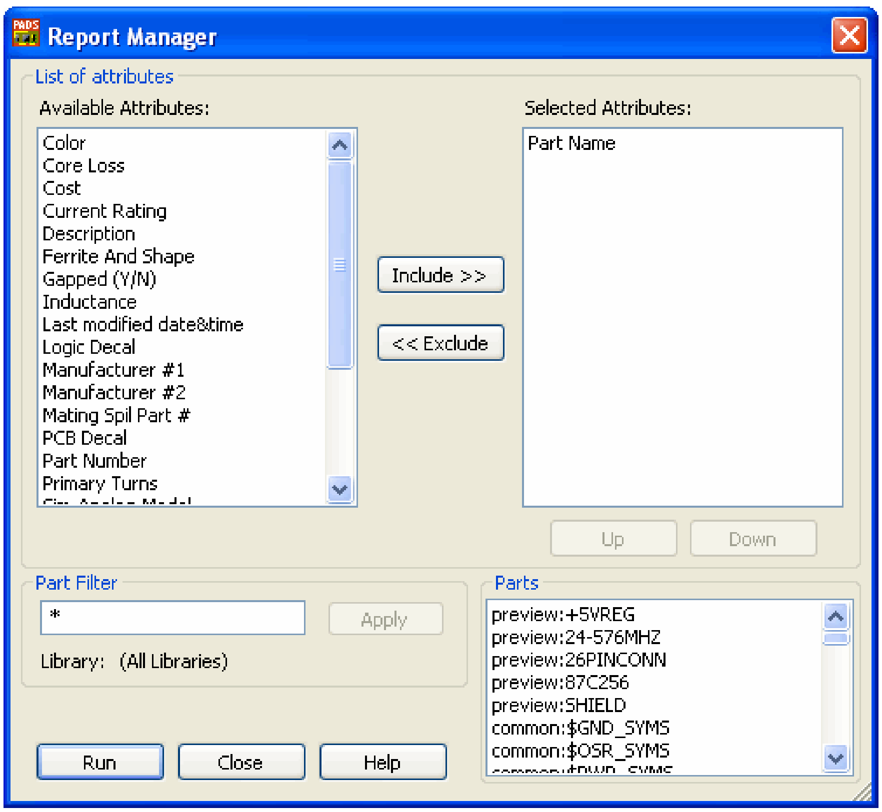


Table 45-284. Report Manager Dialog Box Contents

Field or Button	Description
Available attributes	All attributes of the part types in the selected library. Click an attribute in the list of select it. (To select additional attributes, press CTRL and click each attribute.) Click **Include** >> to include selected attributes in the report.

Table 45-284. Report Manager Dialog Box Contents (cont.)

Field or Button	Description
Selected attributes	Attributes in the report. Click an attribute in the list of select it. (To select additional attributes, press CTRL and click each attribute.) Click **<< Exclude** to remove selected attributes from the report. The order of attributes in the list is the order of columns in the report. Select an attribute and click **Up** or **Down** to change the order.
Include >>	Includes the selected attributes in the report (moves the attributes to the Selected attributes list). Select one or more attributes on the Available attributes list and click **Include >>**.
<< Exclude	Excludes the selected attributes from the report (moves the attributes from the Selected attributes list back to the Available attributes list). Select one or more attributes on the Selected attributes list and click **<< Exclude**.
Up / Down	Moves a selected attribute up or down on the Selected attributes list. List order determines the order in which columns appear in the report.
Part Filter	Specifies the part types to include in the report. Type a part type name in the field or use wildcards (*) to specify a group of part types. For example: * Specifies all part types in the library. +5* Specifies all part types that begin with the characters +5, such as +5volt and +5LS07.
Apply	Filters the part types.
Parts	Lists part types included in the report (as determined by the Part Filter).
Run	Generates the report and lets you save it either in lst for viewing or printing or in csv format for use with MS Excel.
Close	Cancels the operation and closes the dialog box.

Related Topics

Creating a Report of the Parts in a Library

Creating a Report of Decals, Lines or Logic Symbols in a Library

Reports Dialog Box

You can use the Reports dialog box to create reports containing properties or netlist information for the design. In addition to several predefined report formats, you can create custom report formats.

Accessing

- **File Menu > Reports**

Figure 45-315. Reports Dialog Box

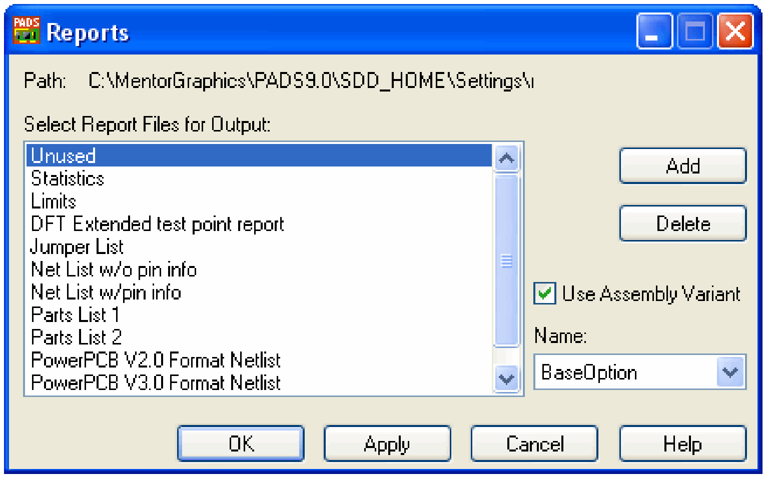


Table 45-285. Reports Dialog Box contents

Name	Description
Path	Displays the path to where the reports are located.
Select Report Files for Output	Lists the report files (formats) available to you.
Add	Opens the Report Format File dialog box where you can browse to the format file you want.
Delete	Removes the report format from the list and changes the file name extension from .fmt to .del.
Use Assembly Variant	Specifies to create a report based on an assembly variant instead of the base design.
Name list	Specifies the variant you want to use. **Restriction:** Available only when Use Assembly Variant is selected.

Related Topics

Creating Reports

Reuse Properties Dialog Box

Use the Reuse Properties dialog box to modify the selected physical design reuse.

Accessing

- **Select a reuse > Right-click > Properties**

Figure 45-316. Reuse Properties Dialog Box

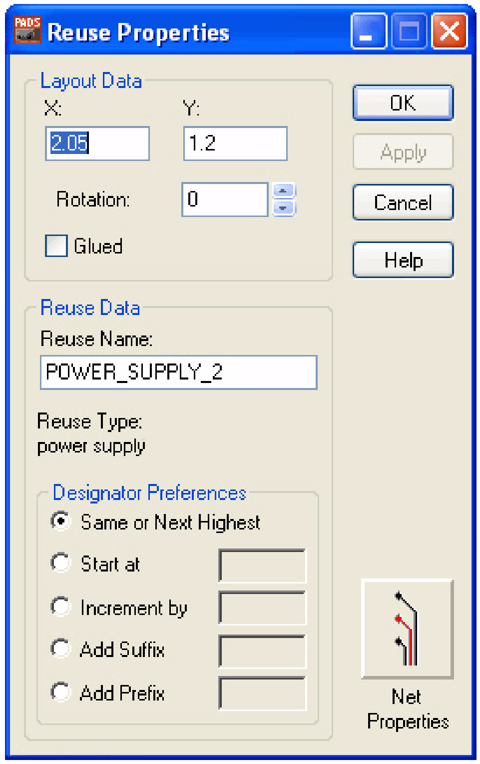


Table 45-286. Reuse Properties Dialog Box contents

Name	Description
X/Y coordinates	Displays the current coordinates of the physical design reuse origin. Type new values to move the physical design reuse to a new location.
Rotation	Displays the current rotation of the physical design reuse. Type a new value or use the arrow buttons to change the rotation angle.
Glued	Sets whether the physical design reuse is glued. Turn on Glued to glue the physical design reuse to the board and prevent moving the physical design reuse. If multiple physical design reuses are selected, and some of them are glued and others are not, this check box is unavailable. You can click Glued to glue all selected physical design reuses or click to clear to unglue all selected physical design reuses.

Table 45-286. Reuse Properties Dialog Box contents (cont.)

Name	Description
Reuse Name	Displays the name of the currently selected physical design reuse. Type a new name in the box to rename the physical design reuse. The default reuse name is based on the reuse type. The reuse name is checked to ensure it is not already used. If it is already used, an error message appears and you can specify a different name. **Tip:** There is no way to display the reuse name and reuse type in the design.
Designator Preferences area	• **Same or Next Highest**—Renames the reference designators for component elements and jumpers in the physical design reuse. By specifying a renaming method, you avoid creating duplicate reference designators between the physical design reuse and the design. This renaming option adds each component with the reference designator defined in the physical design reuse. If the reference designator is already used in the design, the next highest unused reference designator is assigned. For example, if you click this option, and R2 is already used in the design, then R1 in the physical design reuse is assigned R1 in the design, but R is assigned R3 in the design. Also, R1R1 is assigned R1R2 and R1R is assigned R2R. The reuse name is checked to ensure it is not already in use. If it is already used, an error message appears and you can specify a different name.

Table 45-286. Reuse Properties Dialog Box contents (cont.)

Name	Description
Designator Preferences area (Con't)	• **Start at—**Renames the reference designators for component elements and jumpers in the physical design reuse. By specifying a renaming method, you avoid creating duplicate reference designators between the physical design reuse and the design. This renaming option adds each component with a reference designator starting at the number you specify here. The numbering starts at the lowest, numbered reference designator. For example, if the Start at value is 100, R1 in the physical design reuse is assigned R100 in the design. R is also assigned R100 in the design, causing duplicate reference designators. In this case, an error would occur. Values range from 1 to 9999. The value you type here is checked against the design to ensure that duplicate reference designators are not created. If duplicates occur, an error message appears and you can specify a different value. • **Increment by—**Renames the reference designators for component elements and jumpers in the physical design reuse. By specifying a renaming method, you avoid creating duplicate reference designators between the physical design reuse and the design. This renaming option increments the reference designator by the specified number. For example, if the Increment by value is 100, the R1 in the physical design reuse is assigned R101 in the design. Legal characters are 1-9999. The value you type here is checked against the design to ensure that duplicate reference designators are not created. If duplicates occur, an error message appears and you can specify a different value. • **Add Suffix—**Renames the reference designators for component elements and jumpers in the physical design reuse. By specifying a renaming method, you avoid creating duplicate reference designators between the physical design reuse and the design. This renaming option adds the specified suffix, up to four characters long, to each reference designator. For example, with an A suffix, R1 in the physical design reuse is assigned R1A in the design. Illegal characters are brackets ({ }), asterisk (*), space, and period (.). You can, however, leave the Suffix box empty, and that is considered a valid entry. The value you type here is checked against the design to ensure that duplicate reference designators are not created. If duplicates occur, an error message appears and you can specify a different value.

Table 45-286. Reuse Properties Dialog Box contents (cont.)

Name	Description
Designator Preferences area (Con't)	• **Add Prefix—**Renames the reference designators for component elements and jumpers in the physical design reuse. By specifying a renaming method, you avoid creating duplicate reference designators between the physical design reuse and the design. This renaming option adds the specified prefix, up to four characters long, to each reference designator. For example, with an A prefix, R1 in the physical design reuse is assigned AR1 in the design. Illegal characters are brackets ({ }), asterisk (*), space, and period (.). You can, however, leave the Prefix box empty, and that is considered a valid entry. The value you type here is checked against the design to ensure that duplicate reference designators are not created. If duplicates occur, an error message appears and you can specify a different value.
Net Properties button	Opens the Net Properties dialog box where you can resolve netname conflicts between netnames in a physical design reuse and netnames in the design.

Related Topics

Modifying Physical Design Reuse Properties

Routing Rules Dialog Box

Use the Routing Rules dialog box at any level of the hierarchy to specify the topology type of the ratsnest connections, general and autorouting options, and layer and via biasing. You can access the Routing Rules dialog box from within PADS Layout or from within the Decal Editor.

-
- When you open the Routing Rules dialog box from the Decal Editor, only options in the Vias area are available.
- **When you** set up Routing rules for pin pairs, groups, components, or decals, the following rule settings are unavailable: topology type, copper sharing, maximum number of vias.

Accessing

- **Setup** menu > **Design Rules** > choose a hierarchy level > **Routing** button
- **Decal Editor** > **Setup** menu > **Decal Rules** > **Routing** button

Figure 45-317. Routing Rules Dialog Box

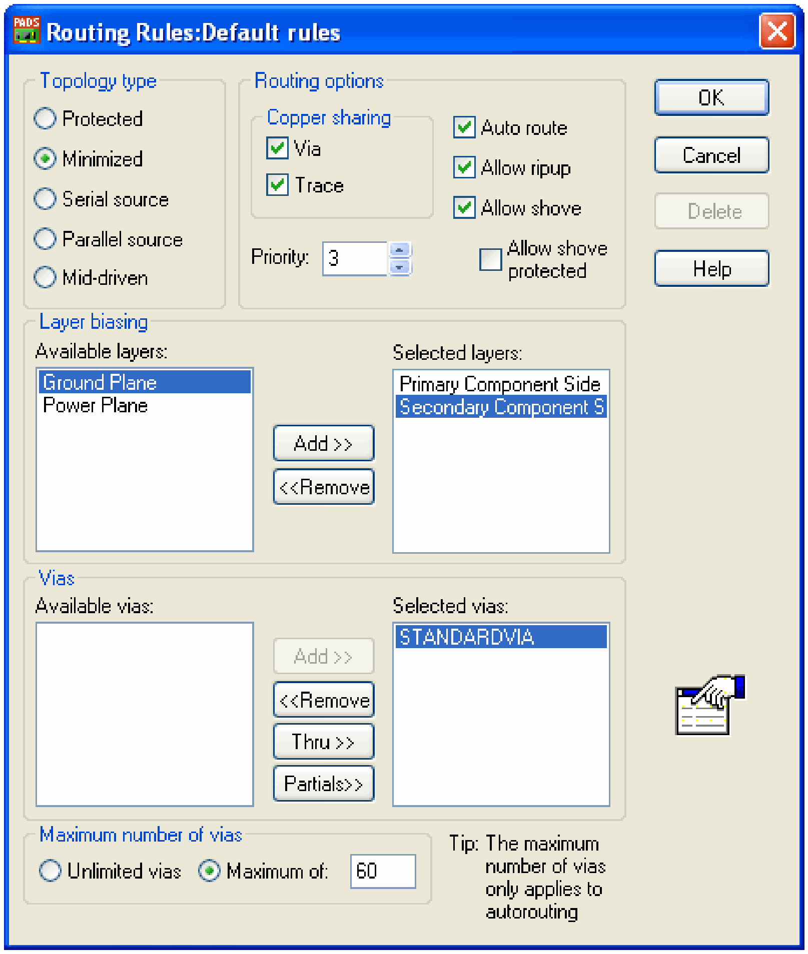


Figure 45-318. Vias Section of PCB Decal Editor Routing Rules Dialog Box

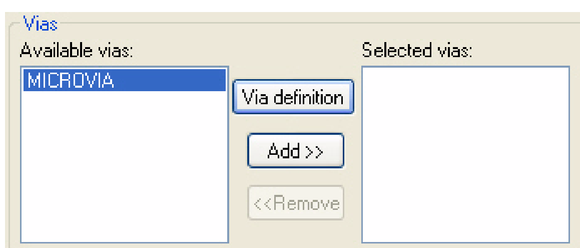


Table 45-287. Routing Rules Dialog Box

Area	Description
Topology type	Specifies the topology of the connections (ratsnest) to provide a visual pin-to-pin order when routing interactively or moving a part. **Restriction:** The Serial source, Parallel source, and Mid-driven settings will only function if you have correctly set the pin Type (such as Source, Load, Terminator) of the parts. • **Protected**—Do not change the order of the connectivity of the connections. **Tip:** This option disables length minimization. • **Minimized**—Order the net by the shortest distance between pins. Net reorder or reconnect is permitted. • **Serial source**—Order the net in a series order from source pins to load pins to a terminator. • **Parallel source**—Same as “Serial source” except order the net with parallel branches for each source-to-load connection. • **Mid-driven**—Divide the net into two branches and order each branch in a source to load to terminator order. **See also:** Placement and Length Minimization

Table 45-287. Routing Rules Dialog Box (cont.)

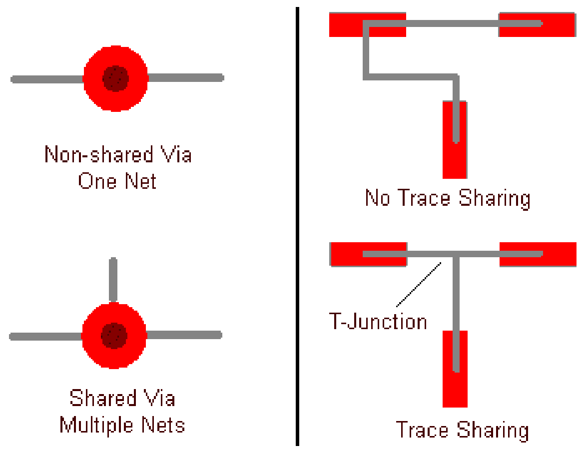
Area	Description
Routing options	• **Copper sharing**—Allow vias or traces to share copper with another object.  **Restriction:** This rule is used only in PADS Router, although you can define this rule in PADS Logic, PADS Layout, or PADS Router. • **Priority**—Assign a priority from 0 to 100. Nets with higher priority are routed first. **Restriction:** PADS Router does not use the priority value. This rule applies only to SPECCTRA. • **Auto Route**—Allow the autorouter to route nets. • **Allow Ripup**—Unroute existing traces and reroute the nets. **Tip:** Enable this option to ripup traces while DRC Warn or Prevent is enabled. • **Allow Shove**—Move unprotected traces aside to create room for new traces. **Tip:** Enable this option to shove traces while DRC Warn or Prevent is enabled. • **Allow Shove Protected**—Move protected traces aside to make room for new traces.
Layer biasing	Layers in the Available layers list can not be used for routing and vias can not start or end on those layers. Only layers in the Selected layers list can be used for routing. **Tip:** Double-click a layer to instantly move it to the opposite list, or use the Add>> or <<Remove buttons to move one or more layers at a time.

Table 45-287. Routing Rules Dialog Box (cont.)

Area	Description
Vias	Vias in the Available vias list can not be used during routing. Only vias in the Selected vias list can be used during routing. All other vias are restricted vias. **Tips:** • Double-click a via to instantly move it to the opposite list, or use the Add>> or <<Remove buttons to move one or more vias at a time. Alternatively, use the Thru>> or Partials>> buttons to move through-hole vias or partial vias to the Selected vias list. • This is one of the main criteria for automatic via selection (another is the active layer pair). Autoselect allows only vias that begin and end on the Layer Pair shown on the Routing tab of the Options dialog box. **Via definition**—This button is only available in the PCB Decal Editor version of the Routing Rules dialog box. This opens the Setup Via dialog box. The Setup Via dialog box does not know about via padstacks in the design. Use this dialog box to set up only via names, not the internal structure of the pad stacks. These via names should reference the real vias in your design, where the internal padstack structure is defined.
Maximum number of vias	**Restriction: This setting is only used by the autorouter.** The autorouter considers this to be a hard rule. Interactive routing and design verification check this rule. This option is available only when setting rules for default, net, and class properties. • **Unlimited Vias**—Allow an unlimited number of vias to be used. • **Maximum of**—Constrain the number of vias to be used. Type the number, 0 to 50000, in the box. **Tip:** An insufficient maximum number of vias might increase autorouting runtime and reduce completion rates.
Delete button	Removes non-default routing rules at the current level of the rules hierarchy. **Restriction:** You cannot delete the Default Routing rules.

Tip: To specify default trace widths, use the Clearance Rules dialog box.

Related Topics

Creating Via Routing Rules in the Decal Editor

Design Rule Categories

Design Rule Hierarchy

Routing Strategy Dialog Box

Use the Routing Strategy dialog box to define a strategy for autorouting your design in PADS Router. You indicate what passes PADS Router should perform, whether to protect the resulting traces, and what intensity to assign to objects.

Accessing

- **Tools** menu > **PADS Router** > **Setup** button

Figure 45-319. Routing Strategy Dialog Box

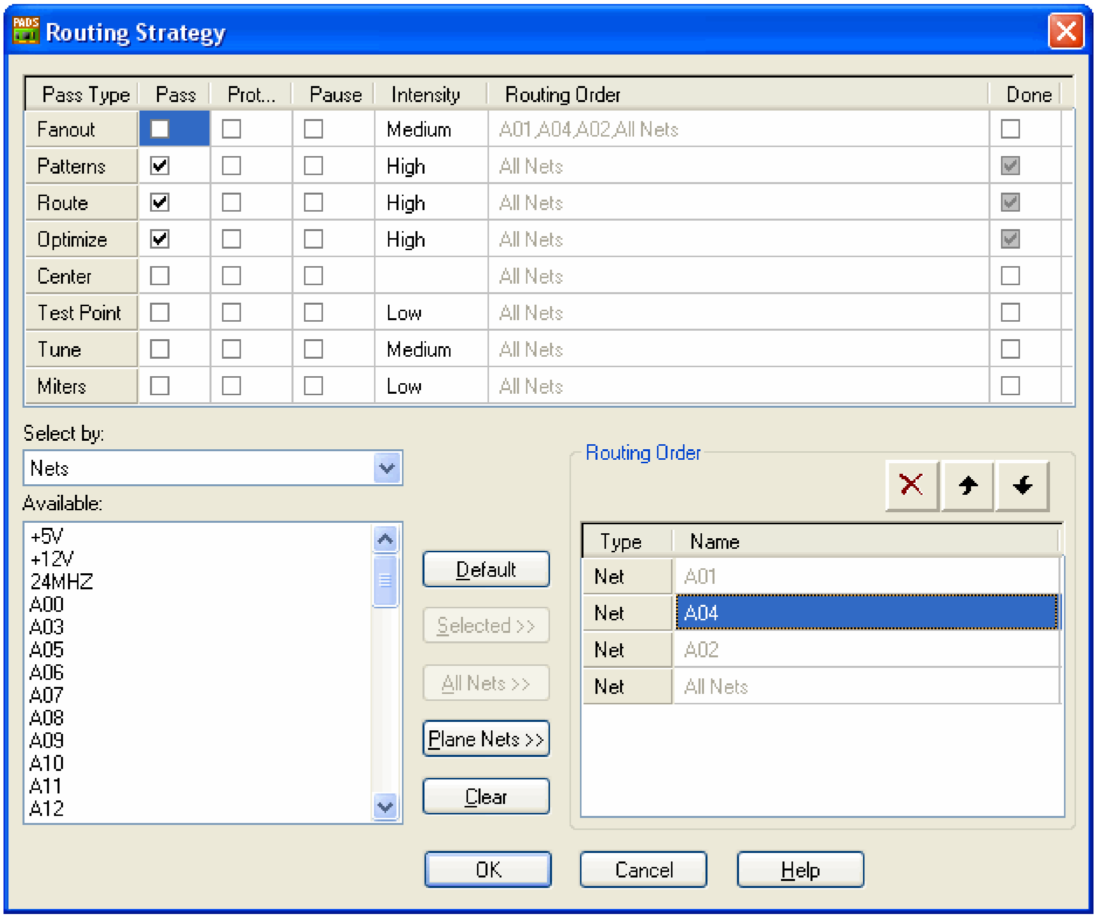


Table 45-288. Routing Strategy Dialog Box contents

Name	Description

Pass Type	Lists the types of passes that PADS Router can make. PADS Router uses the following pass types:
	• **Center pass**—Places traces equidistant from component pins or vias and each other to evenly distribute any available space in the channel.
	• **Fanout pass**—Places vias for inaccessible SMD component pins and routes, from the vias to the pins.
	• **Miters pass**—Converts all 90 degree route corners to diagonal corners.
	• **Optimize pass**—Analyzes each route and tries to improve the quality of the route pattern by removing extra segments, reducing via usage, and shortening routed trace lengths. This pass includes glossing and smoothing processes.
	• **Patterns pass**—Finds and routes groups of unrouted connections that can be completed using typical “c” routing patterns, “z” routing patterns, and memory patterns.
	• **Route pass**—The core pass that performs the majority of autorouting. During this pass, PADS Router attempts to sequentially route each unroute until all connections are attempted. The Route pass contains serial, rip up and retry, push and shove, and touch and cross processes.
	• **Test point pass**—Analyzes the testability of the design, determines which nets require testing, adjusts the routes, and inserts test points to improve testability. You can select whether to add test points during routing or after routing. **See also:** Adding Test Points, Assigning Test Points During Autorouting, and Assigning Test Points After Autorouting topics in the PADS Router Help for more information.
	• **Tune pass**—Adjusts the length of length-controlled traces. The pass examines trace lengths for only completely routed nets or pin pairs. The pass analyzes the current length of each net or pin pair if length rules and length control are enabled, based on the following conditions:
	    • If the cumulative length of the adjacent trace segments is within the range of minimum and maximum trace length, the tune pass skips the trace and does not adjust it.
	    • If the trace is longer than the maximum trace length, the tune pass rips it up and places it in a queue for routing.
	    • If the trace length is less than the minimum trace length, the tune pass changes the length by adding accordion patterns.

Pass	Enables or disables the corresponding autorouting pass type.

Table 45-288. Routing Strategy Dialog Box contents (cont.)

Name	Description
Protect	Enables or disables the protection of traces and vias completed during the corresponding pass type. Protected objects cannot be moved or modified. Traces are protected and vias are glued.
Pause	Enables or disables a pause at the end of the corresponding pass type. **Tip:** When you pause autorouting in PADS Router, the pass and the point within the pass are stored. When you resume autorouting, PADS Router begins where it was paused. Pause does not work when routing in background mode.
Intensity	Sets the level of rip-up effort and number of subpasses to apply to each pass. There are three intensity options: • **Low**—Sets low effort, routing as quickly as possible with a low completion rate, using maximum vias and trace length. Use this setting when you want to complete autorouting quickly. The trade-off is your completion percentage. • **Medium**—Sets medium effort, routing more slowly with a higher completion rate, using fewer vias and shorter traces. Use this setting when you want to balance autorouting time and completion percentage. • **High**—Sets high effort, routing slowly with the highest completion rate, using the fewest vias and shortest traces possible. Use this setting when you want to complete all traces. Your trade-off is the time required to complete routing.
Routing Order List	Displays the routing order of components, nets, and net classes for the selected pass. This list is empty until you click objects from the Object list and click Selected. Set the routing order using the controls in the Routing Order area.
Done	Displays the status of each routing pass. When a pass completes, a check mark appears in this column.
Select by list	Lists the objects in the design. Select an object type (components, differential pairs, differential pair pin pair groups, matched lengths, matched length pin pair groups, nets, or net classes) from the list to update the Available area.
Available area	Lists all of the available objects of the object type selected in the Select by list. Click individual objects to add to the routing order.
Default	Resets all controls to the default strategy settings.

Table 45-288. Routing Strategy Dialog Box contents (cont.)

Name	Description
Selected	Adds selected objects from the Object list to the Routing Order list. To enable Selected, select a pass type and select an object in the Object list.
All Nets	Adds all nets not currently in the Routing Order list to the Routing Order list. The All Nets entry appears in the Routing Order list. To enable All Nets, select a pass type and select an object in the Object list.
Plane Nets	Adds nets associated with plane layers to the Routing Order list. The Plane Nets entry appears in the Routing Order list. To enable Plane Nets, select a pass type and select an object in the Object list.
Clear button	Deletes all items from the Routing Order. To delete only one item, click Delete in the Routing Order area.
Routing Order	Displays the routing order of components, nets, and net classes for the selected pass. This list is empty until you click a pass type. To add an object to the routing order, click the object and click Selected. The following options are available for altering the routing order: • —Deletes selected nets from the Routing Order list. Nets are not deleted from the design; they are only removed from the routing order. • —Moves the selected object up one position in the routing order. • —Moves the selected object down one position in the routing order.

Rules Dialog Box

Use the Rules dialog box to assign place and route constraints to the design.

Accessing

- **Setup** menu > **Design Rules**

Figure 45-320. Rules Dialog Box

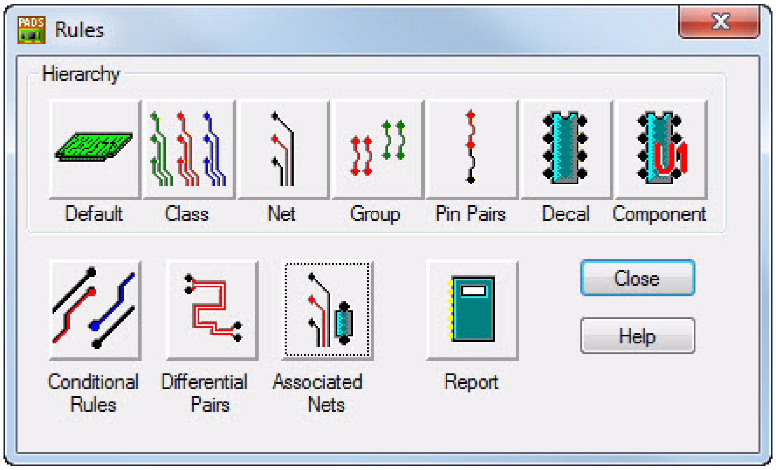


Table 45-289. Rules Dialog Box

Name	Description
Default	Opens the Default Rules Dialog Box.
Class	Opens the Class Rules Dialog Box.
Net	Opens the Net Rules Dialog Box.
Group	Opens the Group Rules Dialog Box.
Pin Pairs	Opens the Pin Pair Rules Dialog Box.
Decal	Opens the Decal Rules Dialog Box. **Restriction:** You can define Decal Rules in PADS Layout; however, these rules are used in PADS Router only.
Component	Opens the Component Rules Dialog Box. **Restriction:** You can define Component Rules in PADS Layout; however, these rules are used in PADS Router only.
Conditional Rules	Opens the Conditional Rule Setup Dialog Box.
Differential Pairs	Opens the Differential Pairs Dialog Box. **Restriction:** You can define Differential Pairs Rules in PADS Layout; however, these rules are used in PADS Router only.
Associated Nets	Opens the Associated Net Rules Dialog Box.
Report	Opens the Rules Report Dialog Box.

Related Topics

Design Rule Hierarchy

Rules Report Dialog Box

You use the Rules Report dialog box to create reports containing information about design rules for the design.

Accessing

- **Setup** menu > **Design Rules** > **Report** button

  Or

- **Setup** menu > **Design Rules** > choose a hierarchy level > **Report** button

Figure 45-321. Rules Report Dialog Box

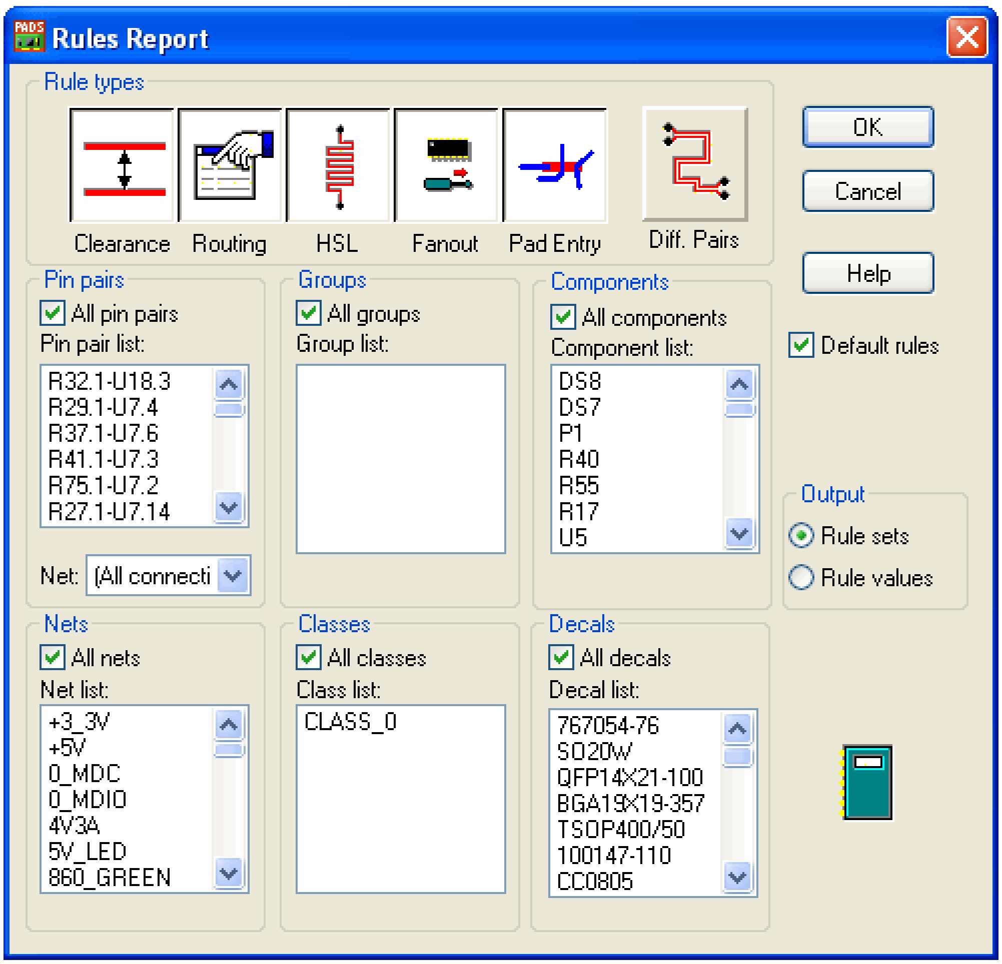


Table 45-290. Rules Report Dialog Box contents

Name	Description
Rules Type area	Specifies the rule types on which you want to report.
Pin Pairs area	Specifies the pin pairs on which you want to report. **Tip:** Select All pin pairs to report on all of them.
Net	Filters the contents of the Pin pair list.
Groups area	Specifies the groups on which you want to report. **Tip:** Select All groups to report on all of them.
Components area	Specifies the components on which you want to report. **Tip:** Select All components to report on all of them.
Nets area	Specifies the nets on which you want to report. **Tip:** Select All nets to report on all of them.
Classes area	Specifies the classes on which you want to report. **Tip:** Select All classes to report on all of them.
Decals area	Specifies the decals on which you want to report. **Tip:** Select All decals to report on all of them.
Default rules	Specifies to report the default rules for each enabled rule type in the Rules Types area.
Output area	• **Rule Sets**—Report all rules that are different from the default rules. • **Rule Values**—Report the values of all rules, even if they match the default rules values.

Related Topics

Creating a Report of the Design Rules

Save CAE Decal to Library Dialog Box

Use the Save CAE Decal to Library dialog box to copy a CAE decal to another name or another library.

Accessing

- **File** menu > **Library** > select a library > **Logic** filter type > select a CAE decal > **Copy**

Figure 45-322. Save CAE Decal to Library Dialog Box

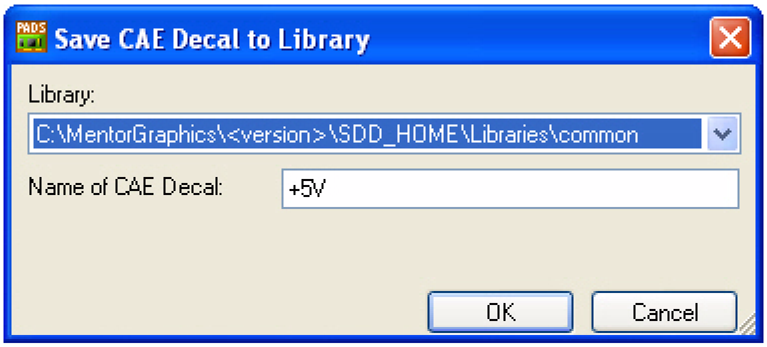


Table 45-291. Save CAE Decal to Library Dialog Box Content

Name	Description
Library	Select the library for the copied CAE decal.
Name of CAE Decal	Type the name for the copied CAE decal.

Related Topics

Copying a Library Item

Save Configuration Dialog Box

Use the Save Configuration dialog box to save the color assignments and settings you've made in the Display Colors Setup dialog box.

Accessing

- **Setup** menu > **Display Colors** > **Save button**

Figure 45-323. Save (color) configuration Dialog Box

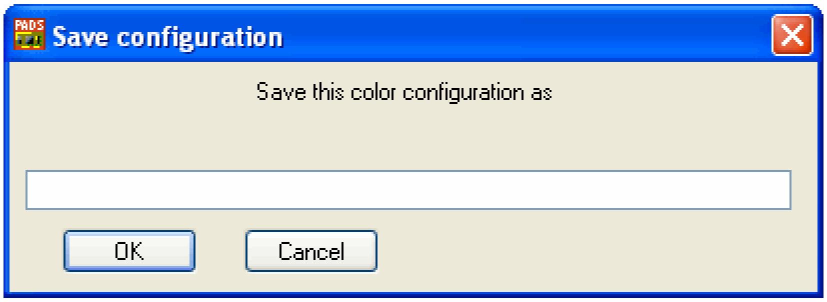


Table 45-292. Save configuration Dialog Box Content

Name	Description
text box	Type the name of the new color configuration. The name will appear in the Configuration list in the Display Colors Setup dialog box.

Related Topics

Display Colors Setup Dialog Box

Display Colors Setup Dialog Box in the Decal Editor

Save Configuration Dialog Box in the Decal Wizard Options

Use the Save Configuration dialog box to save the settings you've made in the Decal Wizard Option dialog box.

Accessing

- **Decal Wizard Options** dialog box > **Save As** button

Figure 45-324. Save (Decal Wizard Options) Configuration Dialog Box

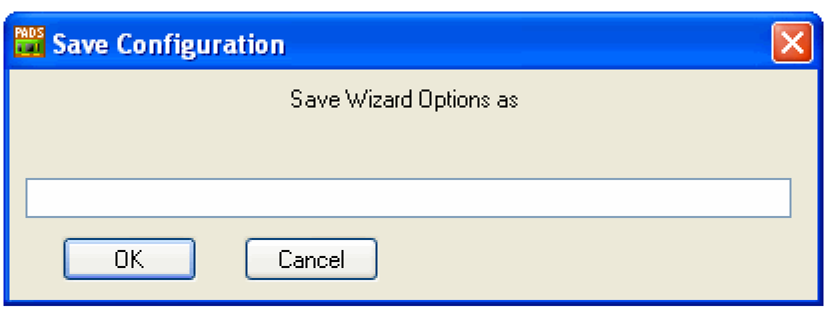


Table 45-293. Save (Decal Wizard Options) Configuration Dialog Box Content

Name	Description
text box	Type the name of the new Decal Wizard Options configuration. The name will appear in the Configuration name list in the Decal Wizard Options dialog box.

Save Drafting Item to Library Dialog Box

Use the Save Drafting Item to Library dialog box to copy a line item to another name or another library.

Accessing

- **File** menu > **Library** > select a library > **Drafting** filter type > select a line item > **Copy**

Figure 45-325. Save Drafting Item to Library Dialog Box

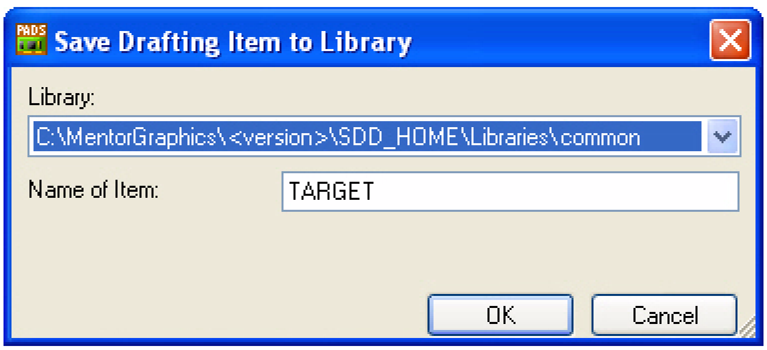


Table 45-294. Save Drafting Item to Library Dialog Box Content

Name	Description
Library	Select the library for the copied line item.
Name of Item	Select the name for the copied line item.

Related Topics

Copying a Library Item

Save Part Types and Decals to Library Dialog Box

Use the Save Part Types and Decals to Library dialog box to save modified decals or parts to libraries.

Accessing

- Select one or more components in the design> **Right-click** > **Save to Library**

Figure 45-326. Save Part Types and Decals to Library Dialog Box

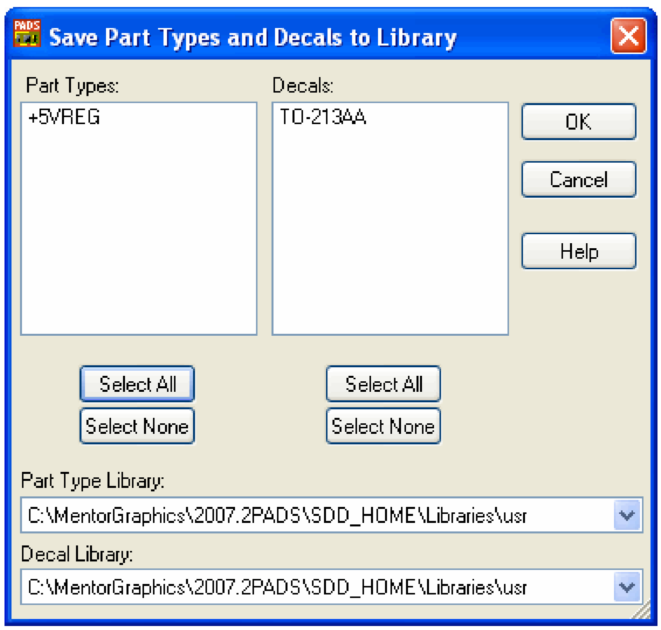


Table 45-295. Save Part Types and Decals to Library Dialog Box Contents

Name	Description
Part Types	The list of Part Types available to save to the library.
Decals	The list of Decals available to save to the library.
Select All	Selects all items in the list above the button.
Select None	Deselects all items in the list above the button.
Part Type Library	A list of all the libraries available to which to save this Part Type.
Decal Library	A list of all the libraries available to which to save this Decal.

Related Topics

Saving Modified Decals and Parts to Libraries

Save Part Type to Library Dialog Box

Use the Save Part Type to Library dialog box to copy a Part Type to another name or another library.

Accessing

- **File** menu > **Library** > select a library > **Parts** filter type > select a part type > **Copy**

Figure 45-327. Save Part Type to Library Dialog Box

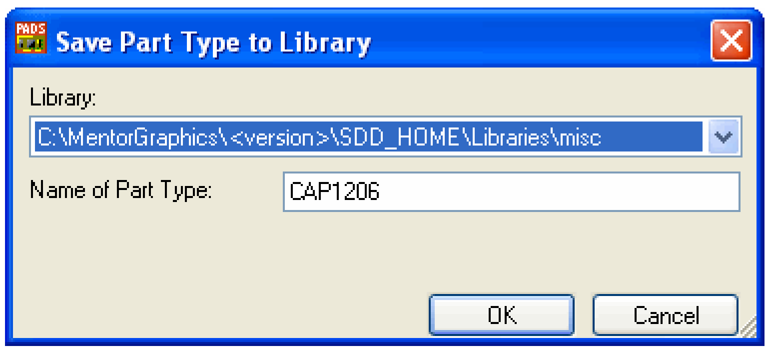


Table 45-296. Save Part Type to Library Dialog Box Content

Name	Description
Library	Select the library for the copied part type.
Name of Part Type	Type the name for the copied Part Type.

Related Topics

Copying a Library Item

Save PCB Decal to Library Dialog Box

Use the Save PCB Decal to Library dialog box to copy a PCB decal to another name or another library.

Accessing

- **File** menu > **Library** > select a library > **Decal** filter type > select a PCB decal > **Copy**

Figure 45-328. Save PCB Decal to Library Dialog Box

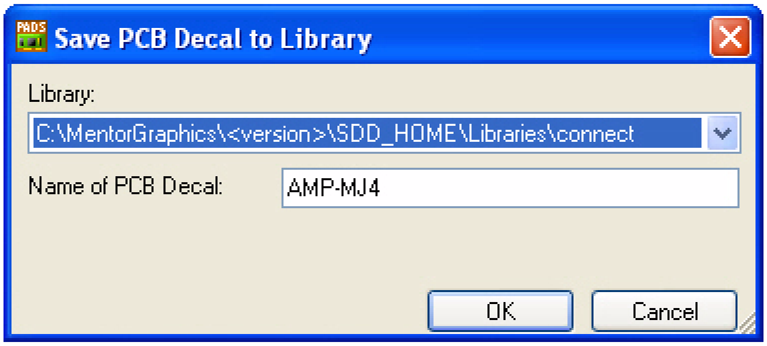


Table 45-297. Save PCB Decal to Library Dialog Box Content

Name	Description
Library	Select the library for the copied PCB decal.
Name of PCB Decal	Type the name for the copied PCB decal.

Related Topics

Copying a Library Item

Save View Dialog Box

Use the Save View dialog box to save a work area view. You can save up to 9 different views.

Accessing

- **View** menu > **Save View**

Figure 45-329. Save View Dialog Box

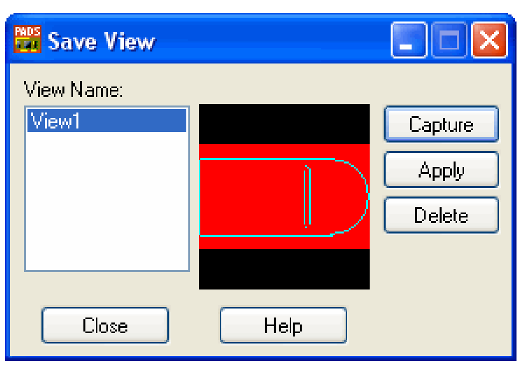


Table 45-298. Save View Dialog Box Contents

Name	Description
View Name	The different views available to you.
Preview area	Shows the view selected in the View Name list.
Capture	Opens the Capture a New View dialog box.
Apply	Applies the selected view.
Delete	Removes the selected view from the View Name list.

Related Topics

Saving a View

SBP Naming Dialog Box

Use the SBP Naming dialog box to specify the numbers and functions of all newly created substrate bond pads, if different from the defaults.

Restriction: This information applies only to the BGA toolkit.

Accessing

- **BGA Toolbar** button > **Wire Bond Wizard** button > **SBP Naming** button

Figure 45-330. SBP Naming Dialog Box

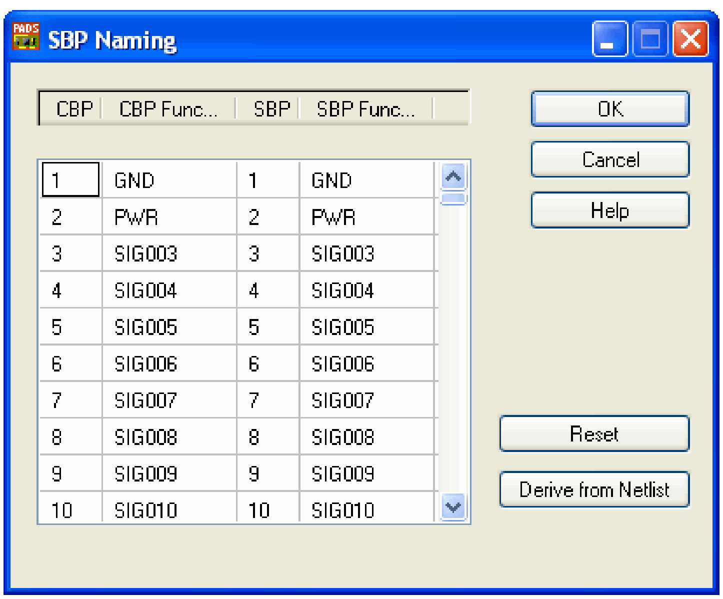


Table 45-299. SBP Naming Dialog Box contents

Name	Description
CBP column	Lists the numbers of the component bond pads.
CBP Function column	Lists the function names of the component bond pads.
SBP column	Use this column to view and specify the numbers of the substrate bond pads listed in the same row.
SBP Function column	Use this column to view and specify the functions of the substrate bond pads listed in the same row.
Reset button	Resets the substrate bond pad numbers and functions to the default, which matches them to the component bond pads of the same number and function.
Derive from Netlist button	Imports substrate bond pad functions from a PADS-ASCII format netlist file. Choose the file you want to load from the Open File dialog box.

Related Topics

To Set SBP Names

SBP Properties Dialog Box

The SBP Properties dialog box displays the pin name, function, position, and dimensions of the selected substrate bond pad. In the Wire Bond Editor, with one or more SBPs selected, right-click and click Properties to edit the SBP Properties.

Restriction: This information applies to only the BGA toolkit.

Note: If you have multiple objects selected, only those properties that are common to all selected objects will appear.

Accessing

- **BGA Toolbar** button > **Wire Bond Editor** button > Select an SBP > right-click > **Properties**

Figure 45-331. SBP Properties Dialog Box

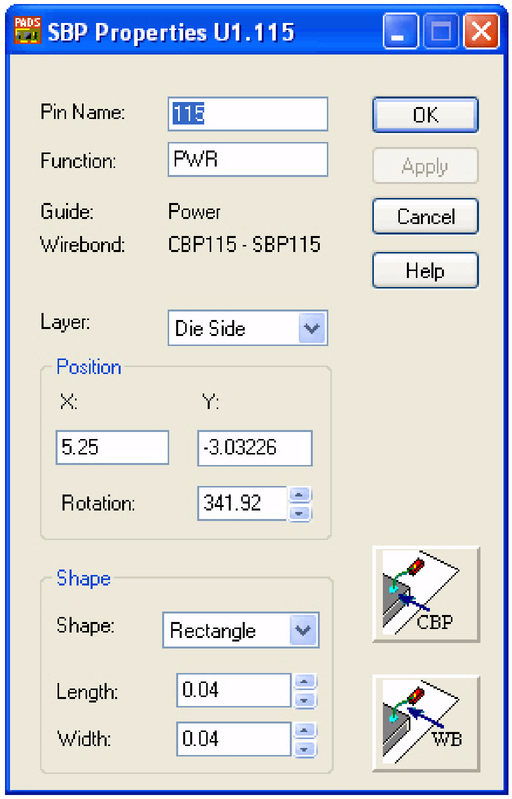


Table 45-300. SBP Properties Dialog Box contents

Name	Description
Pin Name	Assigns a pin name to the currently selected substrate bond pad.
Function	Defines the function of the currently selected bond pad.
Guide	Displays the name of the SBP Guide to which the specified SBP is assigned. If the SBP is not assigned to any guide, this field is blank.
Wirebond	Displays the name of the substrate bond pad and the component bond pad that are connected by the wire bond.
Layer list	Lists all electrical layers for creating SBPs, allowing you to create an SBP on a specific layer.
X and Y	Displays the X and Y coordinates of the bond pad. Type new values to move the bond pad.
Rotation	Assigns a rotation angle, in degrees, for the currently selected substrate bond pad.

Table 45-300. SBP Properties Dialog Box contents (cont.)

Name	Description
Shape list	Assigns a shape to the currently selected bond pad: **Rectangle** or **Oval**.
Length	Assigns a physical length, in current design units, for the currently selected bond pad.
Width	Assigns a physical width, in current design units, for the currently selected bond pad.
CBP button	Opens the CBP Properties dialog box for the component bond pad connected to the currently selected substrate bond pad. This button is unavailable if there is no connected component bond pad.
WB button	Click WB to open the Wire Bond Properties dialog box for the wire bond connected to the currently selected pad. This button is unavailable if there is no connected wire bond.

Related Topics

To Edit Substrate Bond Pads

Select Assembly Variant Dialog Box

You can create assembly drawings of your assembly variants by selecting a variant in the Select Assembly Variant dialog box.

Accessing

- **File** menu > **CAM** > **Add** button > Select **Assembly** from the Document Type list > **Assembly** button

  or

- **File** menu **>** **CAM** **>** select a document name **>** **Edit** button > Select **Assembly** from the Document Type list > **Assembly** button

Figure 45-332. Select Assembly Variant Dialog Box

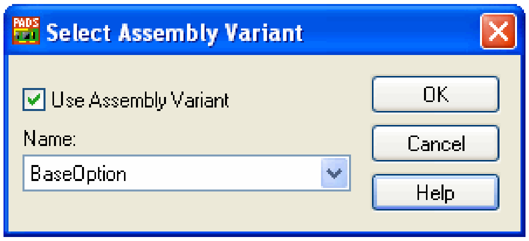


Table 45-301. Select Assembly Variant Dialog Box contents

Name	Description
Use Assembly Variant	Specifies to use an assembly variant as your design input for the assembly drawing. **Tip:** Clear the check box to use all parts in the database, known as the raw database.
Name	Specifies the variant you want to use.

Related Topics

Selecting an Assembly Variant for Assembly Drawings

Defining CAM Documents

Select Graphically Dialog Box

Graphical Selection mode has two modes, Include and Exclude. The default mode is Include mode. The pins that are currently included on the Substrate Bond Pads and BGA Pins lists are highlighted in the work area.

Restriction: This information applies to only the BGA toolkit.

Accessing

- **BGA Toolbar** button > **Route Wizard** button > **Select Pads** tab > **Select Graphically** button

Figure 45-333. Select Graphically Dialog Box

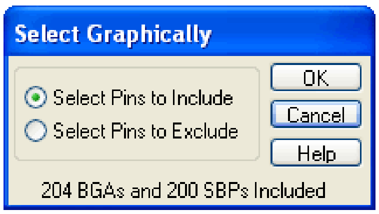


Table 45-302. Select Graphically Dialog Box contents

Name	Description
Select Pins to Include/Exclude	Places Graphical Selection in Include or Exclude mode.

Table 45-302. Select Graphically Dialog Box contents (cont.)

Name	Description
Included/Excluded BGAs and SBPs	Lists the number of BGA pads and SBP pins currently selected for inclusion in or exclusion from processing.

Related Topics

To Use Graphical Selection Mode

Select Items Dialog Box

When you are adding a CAM document, you can use the Select Items dialog box to define which layers and items should appear in a particular document. You can also define colors for layers and items.

Accessing

- **File** menu > **CAM** > **Add** button > **Layers** button

  or

- **File** menu > **CAM** > select a document name > **Edit** button > **Layers** button

Figure 45-334. Select Items Dialog Box

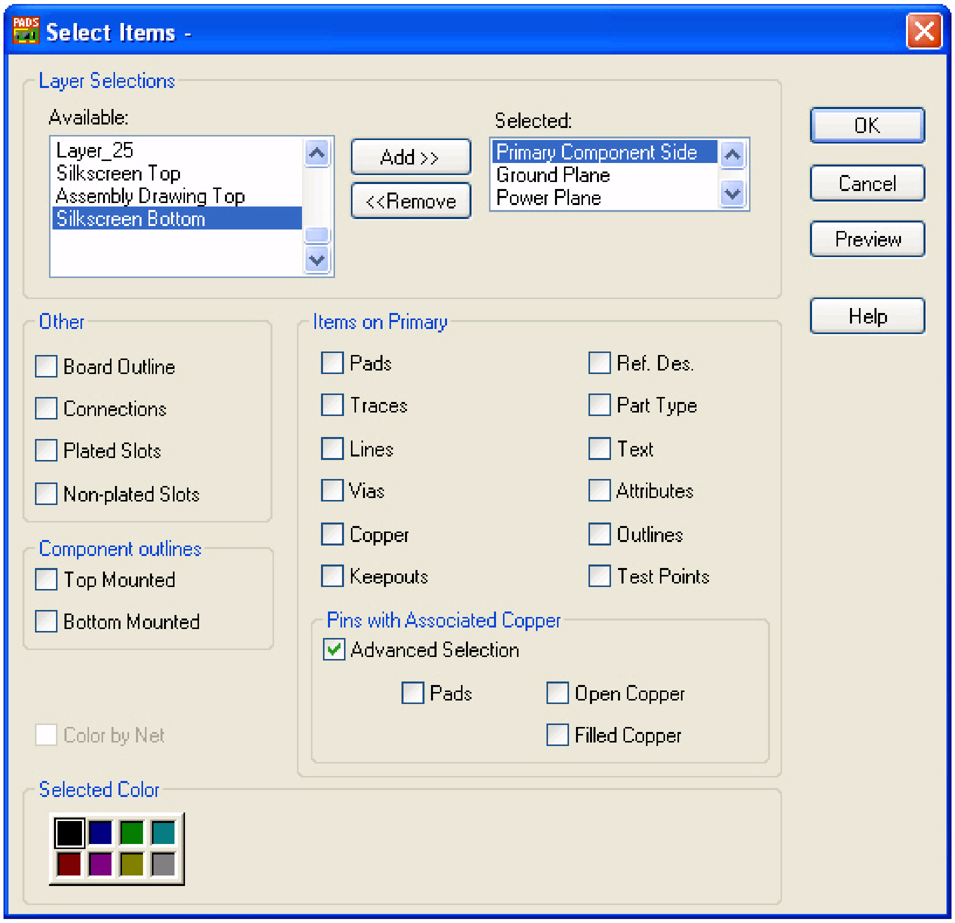


Table 45-303. Select Items Dialog Box contents

Name	Description
Available list	Lists the layers available for display in the output.
Selected list	Lists the layers you selected from the Available list.
Add button	Moves the layer from the Available list to the Selected list. **Tip:** The layers you select are not saved as defaults when you click Save As Defaults on the Add Document dialog box.
Remove button	Moves the layer from the Selected list to the Available list.
Other area	Specifies the color you want for the Board Outline, Connections, Plated Slots, and Non-plated Slots.

Table 45-303. Select Items Dialog Box contents (cont.)

Name	Description
Items on Primary area	Specifies the color you want for Pads, Traces, Lines, Vias, Copper, Keepouts, Reference Designators, Part Types, Text, Attributes, Outlines, and Test Points. **Tip:** Text that is combined with 2D lines is made visible when you select 2D Lines. Combined text is not affected by the Text check box.
Pins with Associated Copper area	Click **Advanced Selection** to enable the selection of the object check boxes. These options allow for selection of associated copper items independently of regular options for pads, vias and traces. • **Pads—**displays pads with associated copper. • **Open Copper—**displays the open copper that is associated to pins. • **Filled Copper—**displays the closed copper shapes that are associated to pins.
Component outlines area	Specifies the color you want for outlines that are Top Mounted and Bottom Mounted.
Color by Net	Specifies to use the View Nets colors in the output.
Selected Color	**Provides a palette of grayscales or colors to use for objects in the output document.** **Restriction: This feature is only available if you select a printer or a plotter as your Output Device. It is not available for Photo output.** **Tip:** Click a color in the palette and then click the box beside a design object. You can assign colors to the items in the Other, Component outlines, and Items on Primary areas.
Preview	Opens the CAM Preview dialog box displaying the colors you've chosen.

Related Topics

Making Design Objects Visible in CAM Documents

Creating CAM Outputs to Manufacture Your PCB

Adding or Editing CAM Documents

Selection Filter Dialog Box, Layer Tab

The Layer Tab displays a list of enabled layers (available for selection). Select layers to expand the selection of objects specified in the Object tab. Click the check box accompanying the layer name.

Tip: By selecting multiple layer names, you can change their selectability in a single step.

Accessing

- **Edit** menu > **Selection Filter** > **Layer tab**

Figure 45-335. Layer Tab

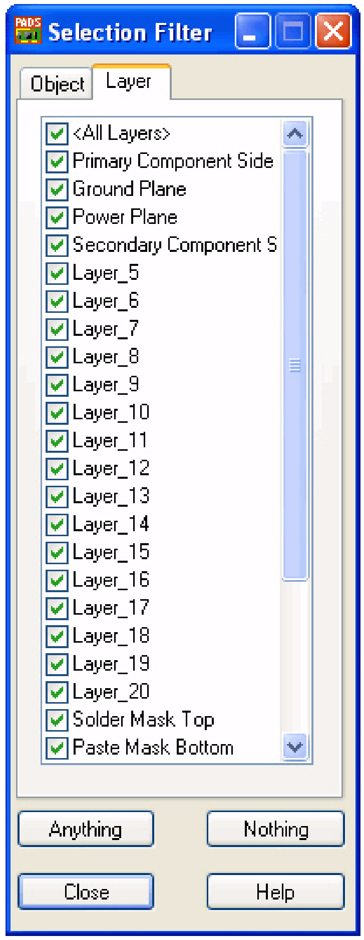


Table 45-304. Layer Tab Contents

Name	Description
Layers list	Specifies the layers on which you want to enable selection.
Anything	Specifies that you want to select anything in the design. **Exception:** Clusters, unions, stitching vias, pin pairs, nets, and board outline shapes are not selected.
Nothing	Specifies that you don't want to select anything in the design.

Related Topics

Using the Selection Filter

Selection Filter Dialog Box, Object Tab

Use the Selection Filter Object tab to specify which objects you can select. Select a check box to enable the object for selection or clear the check box to disable the object for selection.

Accessing

- **Edit** menu > **Selection Filter** > **Object tab**

Figure 45-336. Object Tab

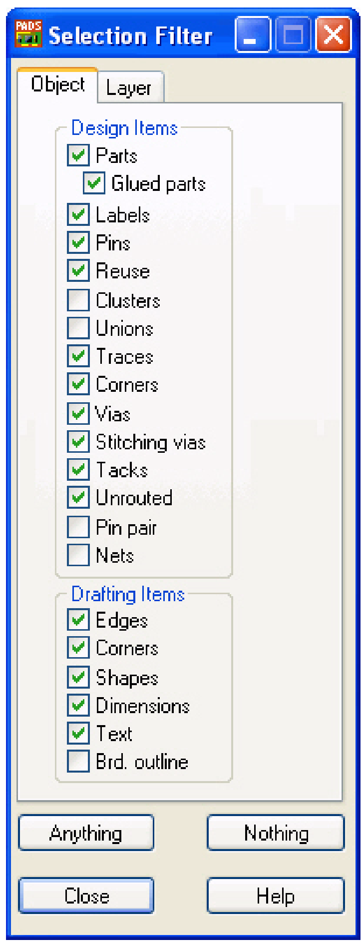


Table 45-305. Object Tab Contents

Name	Description
Design Items	Specifies the design items you want to be able to select in the design. **Tips:** • When Parts is selected, but Glued Parts is clear, you enable the selection of all parts except Glued Parts. When both Parts and Glued Parts are selected, you enable the selection of both glued and unglued parts. Selecting Glued Parts does not modify the jumper selection or selection of unions and reuses. • When Vias is selected, but Stitching is clear, you enable the selection of all vias except stitching vias. When both Vias and Stitching are selected, you enable the selection of both vias and stitching vias.
Drafting Items	Specifies the design items you want to be able to select in the design.
Anything	Specifies that you want to select anything in the design. **Exception:** Clusters, unions, stitching vias, pin pairs, nets, and board outline shapes are not selected.
Nothing	Specifies that you don't want to select anything in the design.

Related Topics

Using the Selection Filter

Select Vault Dialog Box

Use the Select Vault dialog box to:

- Select an existing vault to open.
- Create a new vault.

Accessing

- **Vault view > Select Vault button**

Figure 45-337. Select Vault Dialog Box

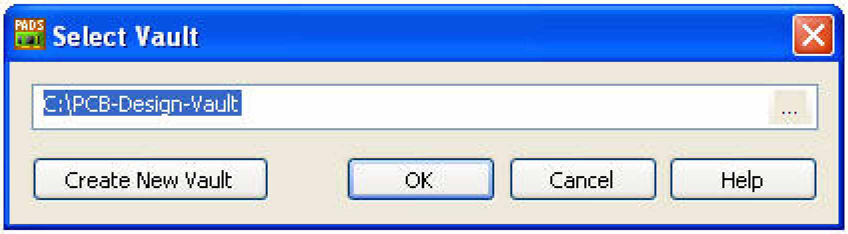


Table 45-306. Select Vault Dialog Box Contents

Name	Description
Browse button	Opens a Browse for folder dialog box, where you can select a vault to open.
Create New Vault button	Opens a Browse for folder dialog box, where you can select or create a folder to contain the new vault. **Requirement:** If you select an existing folder, it must be empty.

Related Topics

Creating a Vault

Changing Vaults

Set Start-up File Dialog Box

Use the Set Start-up File dialog box to select the startup file to use for new design files. A startup file contains global settings such as layer definitions, grids, clearance rules, the attribute dictionary, and so on.

The startup file affects only new designs and specifying a new startup file does not affect existing designs.

Accessing

- **File** menu > **Set Start-up File**

Figure 45-338. Set Start-up File Dialog Box

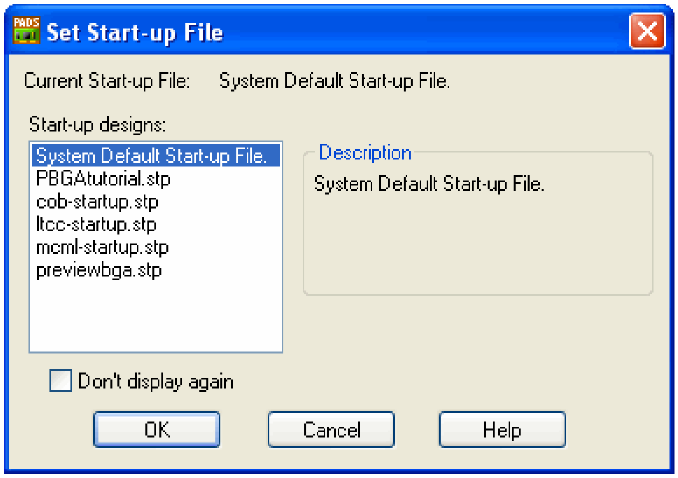


Table 45-307. Set Start-up File Dialog Box Contents

Name	Description
Current Start-up File	The name of the start-up file you're creating.
Start-up designs	A list of the design types available.
Description	A description of the design type.
Don't display again	Specifies to use the selected type for all new design files.

Related Topics

Creating Start-up Files

Specifying the Start-up File

Start-up Files

Setup DXF Drill Size and Symbols Dialog Box

Use the Setup DXF Drill Size and Symbols dialog box to specify drill sizes and symbols in the DXF export. You can also substitute 2D line library items for each drill size in the design.

Accessing

- **File** menu > **Export** > **Select DXF File** > **Save** > **Setup**

Figure 45-339. Setup DXF Drill Size and Symbols Dialog Box

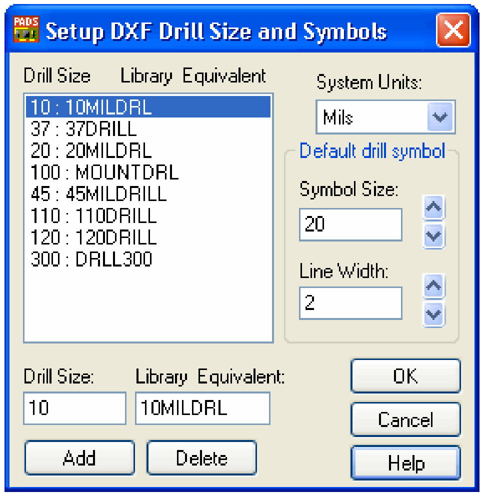


Table 45-308. Setup DXF Drill Size and Symbols Dialog Box Contents

Name	Description
Drill Size Library Equivalent	A list of the items using a 2D-line library item to draw a specific drill size instead of using the default drill symbol.
Drill Size	Specifies the drill size of the drill hole for the item selected in the Drill Size Library Equivalent list.
Library Equivalent	Specifies the 2D-line library item equivalent for the item selected in the Drill Size Library Equivalent list.
Add	Adds a row at the bottom of the Drill Size Library Equivalent list.
Delete	Removes the selected row from the Drill Size Library Equivalent list.
System Units	The units of the symbols.
Symbol Size	Specifies the length/width of the drill hole. **Tip:** By default, drill holes appear in the DXF file as plus signs (+).
Line Width	Specifies the line size used to draw the plus sign. **Tip:** By default, drill holes appear in the DXF file as plus signs (+).

Related Topics

Specifying DXF Drill Sizes and Symbols

Setup SPECCTRA Finish Dialog Box

Use the Setup SPECCTRA Finish dialog box to specify output file locations and to set instructions for the SPECCTRA router regarding actions performed when routing is completed, such as running the mitering pass, running re-cornering, and insertion of test points.

Accessing

- **File** menu > **Export > select SPECCTRA Files > Save > in the SPECCTRA** Link, click **DO File** button > **Finish** button

Figure 45-340. Setup SPECCTRA Finish Dialog Box

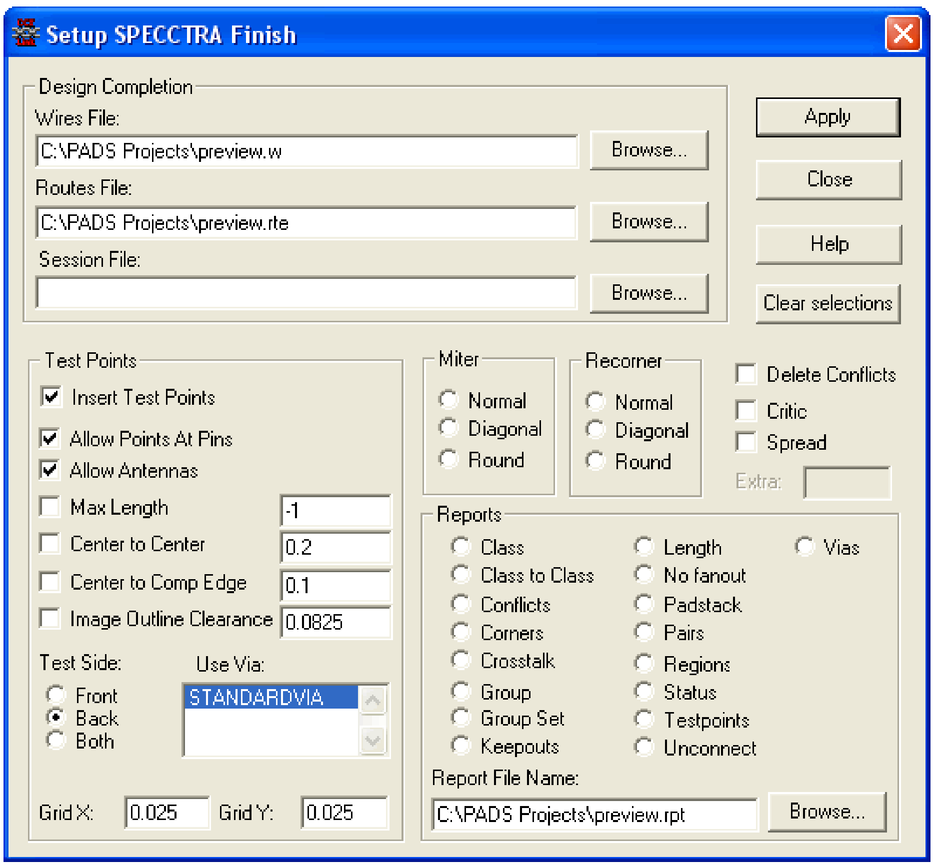


Table 45-309. Setup SPECCTRA Finish Dialog Box contents

Name	Description
Wire File	Specifies the location of the wires file. **Tip:** Click Browse to locate the file.
Route File	Specifies the location of the route file. **Tip:** Click Browse to locate the file.

Table 45-309. Setup SPECCTRA Finish Dialog Box contents (cont.)

Name	Description
Session File	Specifies the location of the session file. **Tip:** Click Browse to locate the file.
Test Points area	Specifies the options you want for test points installed by SPECCTRA. **See also:** Passing DFT Audit Settings to SPECCTRA, “Testpoint” topic in the *SPECCTRA Help*
Miter area	Specifies the miter conversion type: Normal, Diagonal, Round.
Recorner area	Specifies the recornering options: Normal, Diagonal, Round.
Delete Conflicts	Specifies to remove crossover and clearance violations
Critic	Specifies to eliminate notches and remove extra bends.
Spread	Specifies to add extra space if there is room. Type the value of the spread in the Extra box.
Report area	Specifies the type of data you want to include in the report: Class, Class to Class, Conflicts, Corners, Crosstalk, Group, Group Set, Keepouts, Length, No fanout, Padstack, Pairs, Regions, Status, Testpoints, Unconnect, Vias.
Report File Name	Specifies the name of the report. **Tip:** Click browse to locate the file.
Clear selections button	Clears all box entries, check boxes, and radio button selections you have made.

Related Topics

SPECCTRA Output File Location and Router Settings

Setup SPECCTRA Startup Dialog Box

Use the Setup SPECCTRA Startup dialog box to include a line referencing previously entered routes, saved in Wires or Best Save files, in your .do file. SPECCTRA refers to these files upon startup. You can also include the name of the status file and parameters for Via at SMD, Seed Via, and Seed Via minimum distance.

For details on these files and functions see the SPECCTRA Design Language Reference PDF file spdlr.pdf in the SPECCTRA group.

Accessing

- **File** menu > **Export >** select **SPECCTRA Files > Save** > in the SPECCTRA Link, type a name or browse to the file you want in the DO File Name box **> Do File** button **> Startup** button

Figure 45-341. Setup SPECCTRA Startup Dialog Box

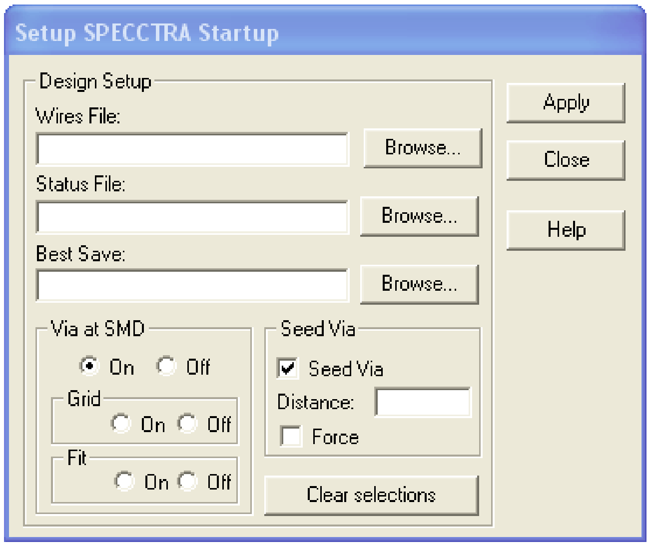


Table 45-310. Setup SPECCTRA Startup Dialog Box contents

Name	Description
Wires File	Specifies the wires file. **Tip:** Click Browse to locate the file.
Status File	Specifies the status file. **Tip:** Click Browse to locate the file.
Best Save	Specifies the best save file. **Tip:** Click Browse to locate the file.
Vias at SMD area	Specifies whether to allow vias at SMD: On or Off.
Grid area	Specifies whether the via is on a grid point: On or Off.
Fit area	Specifies whether the via fits within the pad: On or Off.
Seed Via	Specifies to break up two-pin connections that are larger than a certain length. Type the length value in the Distance box.
Force	Specifies to force the break up of two-pin connections as indicated above.
Clear Selections button	Clears all selections made.

Related Topics

Setting up SPECCTRA .do File Startup Options

Setup Via Dialog Box

Use the Setup Via dialog box to add new via names to the Available Vias list. The Setup Via dialog box does not know about via padstacks in the design. These via names should reference the real via names in your design, where the internal padstack structure is defined.

Accessing

- **PCB Decal Editor** > **Setup** menu > **Design Rules** > **Routing** button > **Via definition** button

Figure 45-342. Setup Via Dialog Box

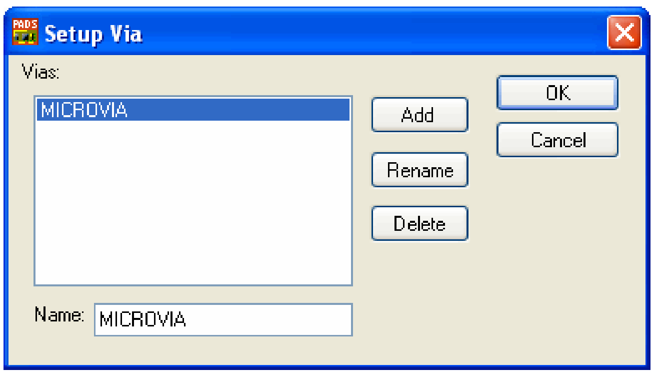


Table 45-311. Setup Via Dialog Box Contents

Name	Description
Vias list	Lists the vias defined and available for the Vias section of the Routing Rules dialog box. Select a name in the list to rename or delete the via.
Name box	Type a new name in the dialog box and then click **Add** to add it to the list. Use the Name box to rename vias in the Vias list.
Add button	Type a new name in the Name list and then click Add to add the via name to the Vias list. **Restriction**: The button is unavailable until you type a name in the Name box.
Rename button	With a via name selected in the Vias list, type a new name in the Name field and click **Rename** to edit the name of the via within the Vias list.

Table 45-311. Setup Via Dialog Box Contents

Name	Description
Delete button	With a via name selected in the Vias list, click **Delete** to delete the name of the via.

Related Topics

Routing Rules Dialog Box

Creating Via Routing Rules in the Decal Editor

Show Attributes Dialog Box

Use the Show Attributes dialog box to select attributes to list in the multi-column list of the Attribute Manager.

Accessing

- **Edit** menu > **Attribute Manager** > **Show button**

Figure 45-343. Show Attributes Dialog Box

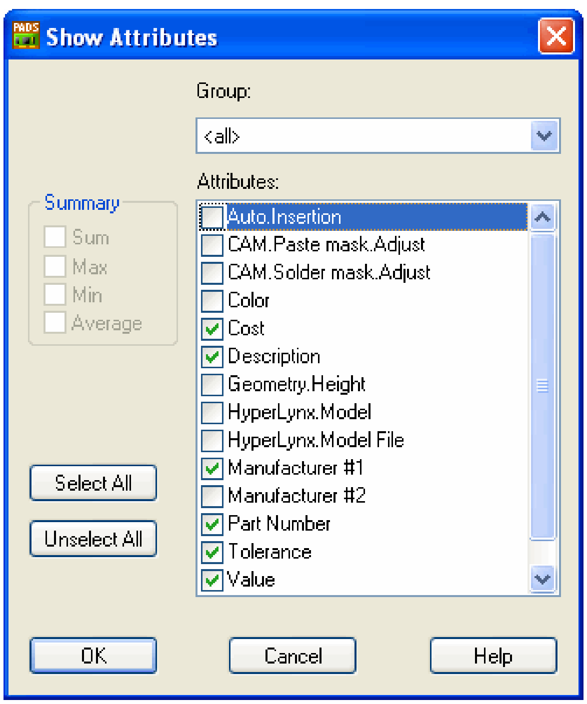


Table 45-312. Show Attributes Dialog Box

Name	Description
Group	Filters the Attributes list. You can choose an attribute group to view.
Attributes list	Specifies the attributes you want to view in the Attribute Manager dialog box.
Summary	Creates summaries of every value of an attribute assigned to a particular objects type. In other words, summaries will appear at the bottom of attribute columns. • **Sum**—Shows the total of the attribute values. The summary applies to the attribute you select in the Attributes list. The summary displays in the Attribute Manager dialog box. • **Max**—Shows the maximum value used by the attribute. The summary applies to the attribute you select in the Attributes list box. The summary displays in the Attribute Manager dialog box. To create a summary that is the range of attribute values, click both the **Min** and **Max** check boxes. • **Min**—Shows the minimum value used by the attribute. The summary applies to the attribute you select in the Attributes list box. The summary displays in the Attribute Manager dialog box. To create a summary that is the range of attribute values, click both the **Min** and **Max** check boxes. • **Average**—Shows the average of the attribute values. The average is the sum of all attribute values divided by the number of values assigned. The summary applies to the attribute you select in the Attributes list. The summary displays in the Attribute Manager dialog box. **Restriction:** Summaries are only available for Number, Decimal Number, and Measure attribute types.
Select All	Selects all check boxes in the Attributes list.
Unselect All	Clears all check boxes in the Attributes list.

Related Topics

Using the Attribute Manager

Showing Attributes in the Attribute Manager

SPECCTRA DO File Dialog Box

The .do file is an editable batch script file, which controls SPECCTRA operation. You can add or edit command lines in a .do file. When you start the editor, the .do file you specify in the SPECCTRA Link dialog box is read for editing.

Accessing

- **File** menu > **Export >** select **SPECCTRA Files > Save** > in the SPECCTRA Link, type a name or browse to the file you want in the DO File Name box **> Do File** button

Figure 45-344. SPECCTRA DO File Dialog Box

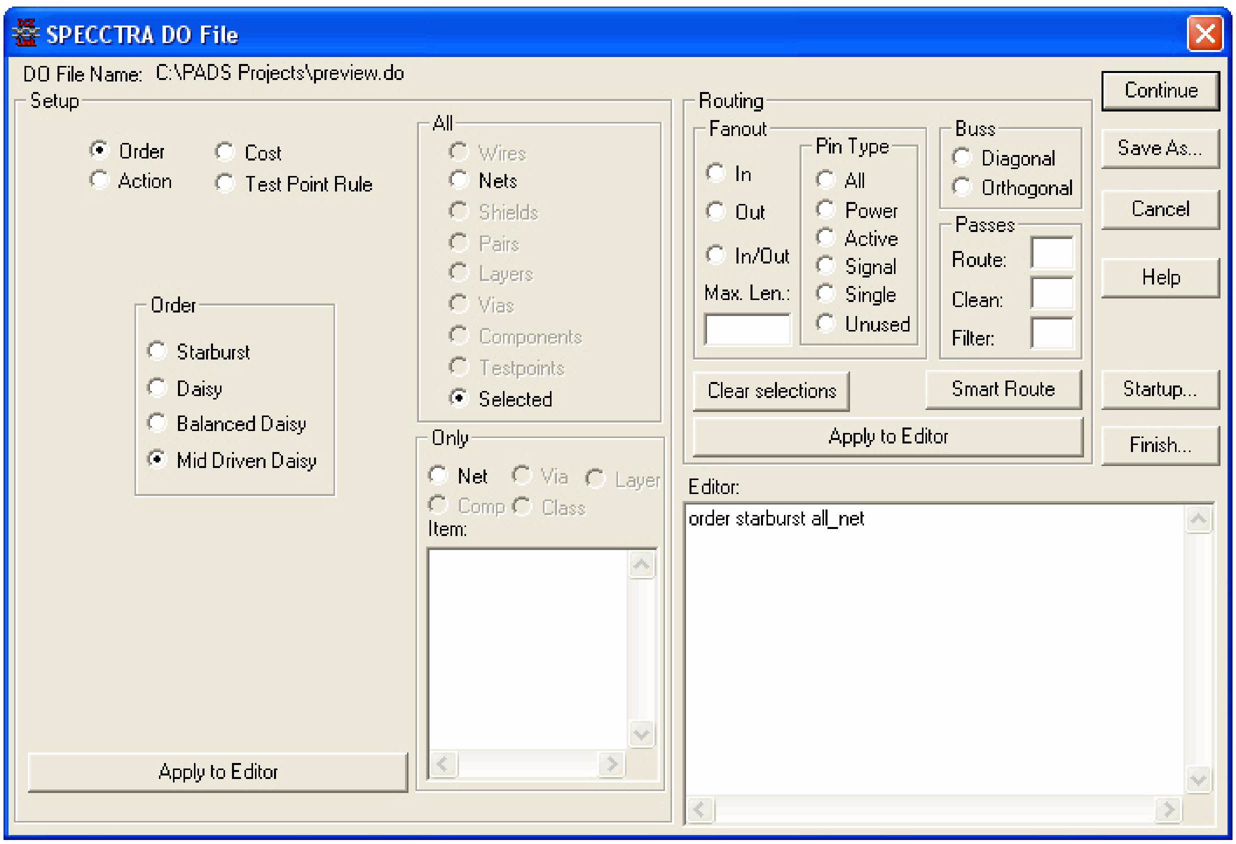


Table 45-313. SPECCTRA DO File Dialog Box contents

Name	Description
Setup area	The contents of the **Setup** area change according to the options you select in the Setup area. • **Order** -change the original net ordering and control whether nets are routed in daisy-chain or starburst fashion. **See also:** "Order" and "Choosing Starburst or Daisy-Chain Wiring" topics in the *SPECCTRA Help* • **Action** - control wire rerouting, net routing, and the availability of connections, vias, and layers for autorouting. **See also:** "Protect/Unprotect"," "Fix/Unfix," and "Select/Unselect" topics in the *SPECCTRA Help* • **Cost** - control routing costs and override the autorouter internal cost table. **See also:** "Cost," "Limit," "Tax," and "Using Standard Autorouting Commands" topics in the *SPECCTRA Help* • If you click Cost in the Cost area and Layer in the All area, options in the Type area appear. **See also:** "Type," "Length," "Way (Cost)," and "Way (Limit)" topics in the *SPECCTRA Help* • **Test Point Rule** - control passing DFT Audit test point placement options to SPECCTRA for its test point placement routine. **See also:** Passing DFT Audit Settings to SPECCTRA and the "Testpoint" and "Testpoint Antennas" topics in the *SPECCTRA Help.*
All area	Limits actions to certain selected objects. The content of this area changes depending on the options you select in the Setup area.
Only area	Limits actions to certain selected objects.

Table 45-313. SPECCTRA DO File Dialog Box contents (cont.)

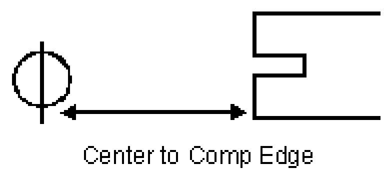
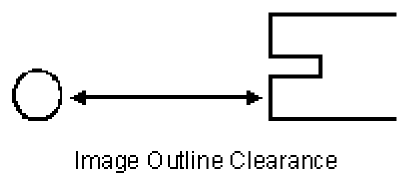
Name	Description
Test Points area	Available only when Test Points is selected. • **Insert Test Points**—Allows insertion of test points and makes the options in the Test Points area available. • **Allow Points at Pins**—Allows placement of test points on component pins. There is no equivalent in DFT Audit; test points are always allowed on pins. • **Allow Antennas**—Allows antennas. Set a length. There is no equivalent in DFT Audit. • **Max Length**—Sets the length restriction for antennas. The default maximum length is negative one (-1), no length restriction. • **Center to Center**—The distance between the centers of test points.  • **Center to Comp Edge**—The distance between the center of the test point and the component outline.  • **Image Outline Clearance**—The clearance between the component outline and the test point carrier (via or component pin). If the Image Outline Clearance is negative, a zero (0) is set.  • **Test Side**—Searches the specified side for test point placement. • **Use Via**—Uses vias as test points. • **Grid X, Y**—The test point grid. **See also:** Options Dialog Box, Grids page
Routing area	Set fanout rules: direction, pin type, and maximum length. Set the **Bus** direction, and enter the number of **passes** for each type.
Clear selections button	Clears all of the settings for the area.
Smart Route button	Specifies to autoroute your design based on how your design is converging. **See also:** the *SPECCTRA Help.*
Apply to Editor button	Specifies to write the commands from this area to the .do file.
Editor area	The contents of the .do file. Command appear here when you click the Apply to Editor button.

Table 45-313. SPECCTRA DO File Dialog Box contents (cont.)

Name	Description
Continue button	Starts the conversion and loading process.
Save As button	Specifies that you want to save the .do file with a specific name and location.
Startup button	Opens the Setup SPECCTRA Startup dialog box.
Finish button	Opens the Setup SPECCTRA Finish dialog box.

Related Topics

Creating or Editing a .do File

SPECCTRA Link Dialog Box

When you start SPECCTRA from within PADS Layout, the SPECCTRA Link dialog box enables you to load in and out of SPECCTRA automatically.

Alternative: If you start SPECCTRA independently of PADS Layout, use the Stand-alone SPECCTRA Link dialog box to load in and out of SPECCTRA manually.

Accessing

- **File** menu > **Export > select SPECCTRA Files > Save**

  or

- Use Windows Explorer to navigate to your ...\SDD_HOME\Programs directory, and double-click **pads2sp.exe**

Figure 45-345. SPECCTRA Link Dialog Box - from Layout

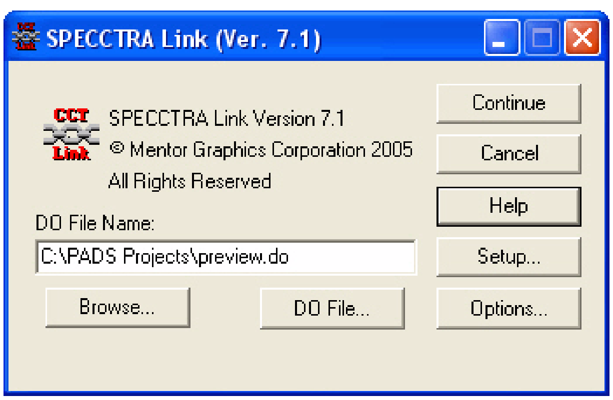


Figure 45-346. SPECCTRA Link Dialog Box - Stand-alone

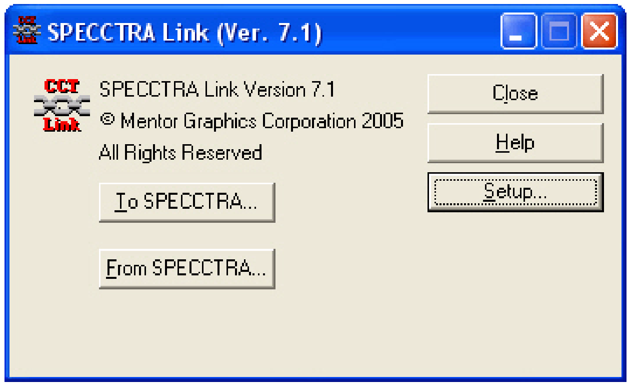


Table 45-314. SPECCTRA Link Dialog Box contents

Name	Description
DO File Name	Specifies the DO file you want to use. **Tip:** Click Browse to locate the file. **Restriction:** From Layout only.
Continue button	Loads the design into SPECCTRA and the router runs in batch mode.
Setup button	Opens the SPECCTRA Setup dialog box.
DO File button	Opens the SPECCTRA DO File dialog box. **Restriction:** From Layout only.
Options button	Opens the Options dialog box. **Restriction:** From Layout only.
To SPECCTRA button	Opens the To SPECCTRA dialog box. **Restriction:** Stand alone only.
From SPECCTRA button	Opens the From SPECCTRA dialog box. **Restriction:** Stand alone only.

Related Topics

Loading In and Out of SPECCTRA Automatically

SPECCTRA Options Dialog Box

The Options dialog box appears when you click the Options button on the SPECCTRA Link dialog box, the TO SPECCTRA dialog box (stand-alone), or the FROM SPECCTRA dialog box (stand-alone).

This dialog box controls options for sending via keepout information, passing advanced rules to SPECCTRA, setting a mode for trace arc translation, and returning the unused pins net from SPECCTRA.

See also: Unused Pins Net

Accessing

- **File** menu > **Export** > select **SPECCTRA Files > Save** > in the SPECCTRA Link, click the **Options** button

Figure 45-347. Specctra Options Dialog Box

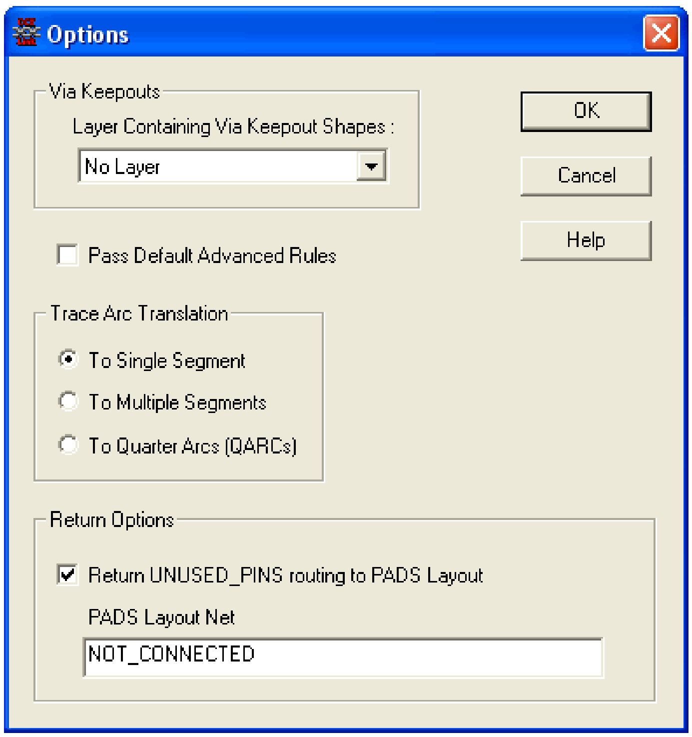


Table 45-315. Specctra Options Dialog Box contents

Name	Description
Layer Containing Via Keepout Shapes list	Specifies the layer that contains the via keepout areas, and that you want to send them from your decals to SPECCTRA.
Pass Default Advanced Rules	Specifies to pass default Selected Layer and Selected Via rules to SPECCTRA. These rules require the Advanced Rules option in SPECCTRA. Turn this option on if you have this SPECCTRA option; otherwise, leave this option off. **See also:** PADS Layout to SPECCTRA Rules Conversion

Table 45-315. Specctra Options Dialog Box contents (cont.)

Name	Description
Trace Arc Translation area	Specifies the mode to perform trace arc translation. • **To Single Segment**—Replaces each trace arc with a single segment. This is the default mode. • **To Multiple Segments—**Replaces a trace arc with multiple segments. The original trace arc is divided into smaller arcs (equal to approximately 5 degrees) and then each smaller arc is replaced by a single segment. The result is a polyline of multiple segments instead of the arc. • **To Quarter Arcs (QARCs)**—Breaks existing arcs into quarter arcs and other segments. (Quarter arcs are arcs whose start and end points are exactly 0–90, 90–180, 180–270, and 270–360 degrees.) The quarter arcs are translated to the SPECCTRA QARC structure. The remaining parts of arcs are translated to polylines.
Return Options area	Specifies to return unused pin and fanout information to PADS Layout. Type the name of the net in the PADS Layout design that will contain the unused pins. Provide a new name if you do not want to use the default. Clear this option to ignore unused pin and fanout information when returning to PADS Layout. **Tip:** SPECCTRA names the unused pins net +UNUSED_PINS+ while previous versions named it *UNUSED_PINS*. The SPECCTRA Translator interprets both names. The maximum netname length in PADS Layout is 47 characters. You can use any alphanumeric characters except for brackets{ }, asterisks *, or spaces.

Related Topics

Setting SPECCTRA Options

SPECCTRA Setup Dialog Box

Use the SPECCTRA Setup dialog box to set SPECCTRA automatic startup information.

Accessing

- **File** menu > **Export >** select **SPECCTRA Files > Save** > in the SPECCTRA Link, click the **Setup** button

Figure 45-348. SPECCTRA Setup Dialog Box

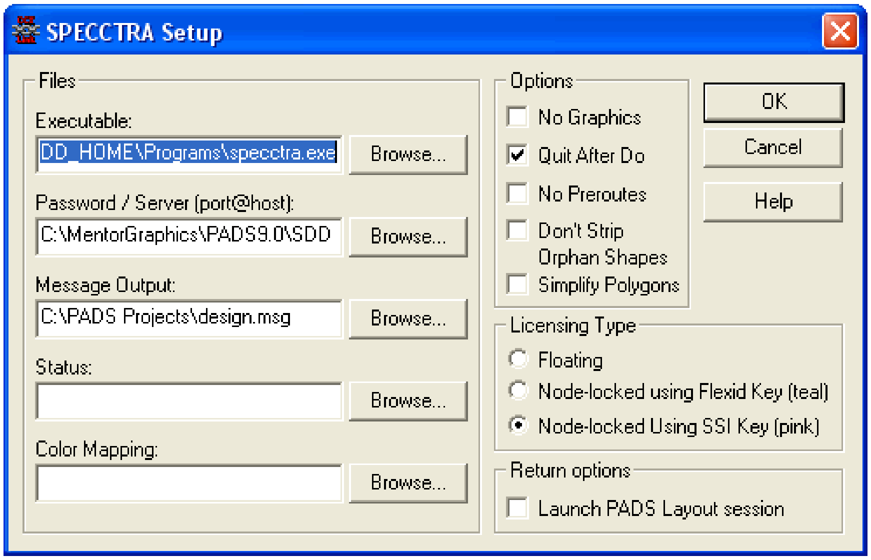


Table 45-316. SPECCTRA Setup Dialog Box contents

Name	Description
Executable	Specifies the executable needed to run SPECCTRA.
Password/Server	Specifies the password file needed to run SPECCTRA. **Tip:** For teal key node-locked or floating licensing, point to your license server in the standard port@host format. For example, 7508@myserver.
Message Output	Specifies the SPECCTRA message output file.
Status	Specifies the SPECCTRA status file.
Color Mapping	Specifies the SPECCTRA color mapping file.
No Graphics	Specifies to disable the SPECCTRA graphic display and make SPECCTRA run faster.
Quit After Do	Specifies to close SPECCTRA after it processes the .do file commands.
No Preroutes	Specifies to delete all prerouted traces before entering SPECCTRA.
Don’t Strip Orphan Shapes	Specifies to ignore copper without net assignments, which have no net association in SPECCTRA.
Simplify Polygons	Specifies to convert one-inch square, or smaller, polygons to simple rectangles.

Table 45-316. SPECCTRA Setup Dialog Box contents (cont.)

Name	Description
Licensing Type area	Select the licensing type you want: Floating, Node-locked with Flexid Key, or Node-locked with SSI key.
Launch PADS Layout session	Specifies to reload the routed design back into PADS Layout after it processes the .do file commands.

Related Topics

Setting the SPECCTRA Automatic Startup Information

Start-up File Output Dialog Box

Use the Start-up File Output dialog box to create a startup file that contains global settings such as layer definitions, grids, clearance rules, the attribute dictionary, and so on. You can create different startup files and specify which one to use when creating a new design file. This capability enables you to save setup time when creating a new design by reusing global settings that you saved in the startup file.

Accessing

- **File** menu > **Save As Start-up File**.

Figure 45-349. Start-up File Output Dialog Box

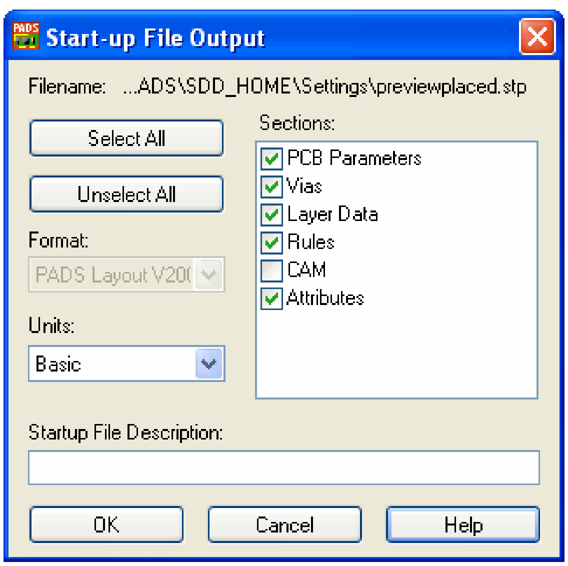


Table 45-317. Start-up File Output Dialog Box Contents

Name	Description
Filename	The name of this startup file.
Select All	Selects all check boxes in the Sections list.
Unselect All	Deselects all check boxes in the Sections list.
Sections	The items available to you to include in the startup file. • **PCB Parameters**—Global information, such as colors, layer definitions, and grids • **Vias**—Via information, such as default via type, jumpers, padstack definitions and locations • **Layer Data**—Layer information specified in the Layers Setup dialog box, such as number of layers, layer names, routing direction for the layer, electrical type, and associations • **Rules**—Rules information, such as clearance, routing, and high-speed • **CAM**—CAM information related to the plot file configurations • **Attributes**—Attribute information, such as the attribute dictionary, all attributes assigned to objects in the design, and attribute status (read only, system, ECO registered, or hidden). Values in the attribute hierarchy are not saved.
Format	Sets the format for this startup file.
Units	Sets the units you want to use for this startup file. **Tip:** Current units provide more information than Basic units, such as grid positions.
Startup File Description	A place to type a brief description of the global settings you are saving. The description appears when you select the startup file in the Set Start-up File dialog box, and should help remind you of the settings in the startup file.

Related Topics

Creating Start-up Files

Specifying the Start-up File

Start-up Files

Status Dialog Box

Use the Status window to view selection information, and gain access to frequently used Options settings.

Accessing

- Ctrl+Alt+S

Figure 45-350. Status Dialog Box

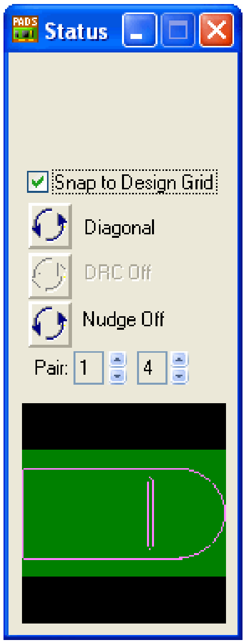


Table 45-318. Status Dialog Box Contents

Name	Description
Information area (blank area in figure)	Displays information about the currently selected object.
Snap To Grid	Toggles the Snap to grid check box of the Design grid in the Grids Options.
Line/Trace Angle	Toggles the Line/trace angle of the Design Options.
On-line DRC	Toggles the On-line DRC setting of the Design Options. **Restriction:** If the On-line DRC is set to Off, this button is unavailable and you must enable it from the Options dialog box.
Nudge	Toggles the Nudge setting of the Design Options.

Table 45-318. Status Dialog Box Contents

Name	Description
Layer Pair	Toggles the Layer pair setting of the Routing Options.
Postage Stamp	Displays a miniature of the current view.

Step and Repeat Dialog Box

You can define complex, repetitive array patterns of objects in the PCB Decal Editor using the Step and Repeat dialog box. You can select multiple or single items for replication. Step and Repeat also automatically increments text and pin numbers.

Three types of Step and Repeat array replications are available:

Linear Tab	For planar replication, in the horizontal and vertical directions.
Polar Tab	For angular replication, rotation around the decal origin. This is useful for creating polar arrays of terminals.
Radial Tab	For radial replication, along the radial direction starting from the decal's origin. This is useful for creating polar arrays of terminals.

Accessing

1. In the PCB Decal Editor, click the **Drafting toolbar button.**
2. Click the **Select** button.
3. Select one or more terminals, 2D line items, text items, copper items, copper cut outs, or keepouts.
4. Right-click and click **Step and Repeat**. The Step and Repeat dialog box appears.

Linear Tab

Use the Linear tab in the Step and Repeat dialog box to create planar arrays.

See also: Incrementing Texts and Pin Numbering, Using Step and Repeat

Figure 45-351. Step and Repeat Dialog Box Linear Tab

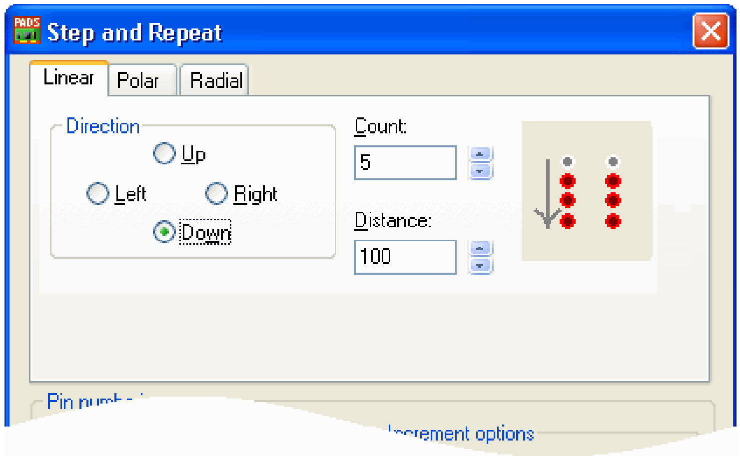


Table 45-319. Linear Tab Contents

Command	Description
Direction	Sets the direction of the replication in the array: Up, Down, Left, or Right.
Count	Sets the number of replications in the array.
Distance	Sets the distance in the array: X distance in current units when the Linear Direction is Left or Right; Y distance in current units when the Linear Direction is Up or Down. Negative values reverse the direction.

Polar Tab

Use the Polar tab in the Step and Repeat dialog box to create angular, or circular, arrays.

See also: Incrementing Texts and Pin Numbering, Using Step and Repeat

Figure 45-352. Step and Repeat Dialog Box Polar Tab

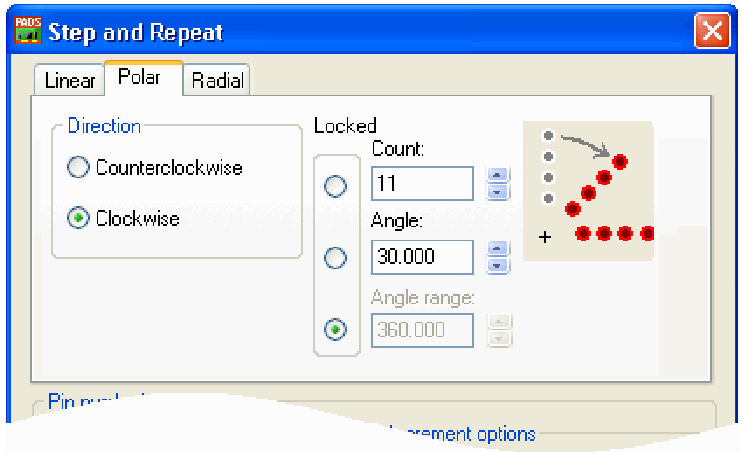


Table 45-320. Step and Repeat Polar Tab Contents

Command	Description
Direction	Sets the direction of replication in the array: Counterclockwise or Clockwise.
Locked area	Controls the automatic adjustment of Angle Range, Delta Angle, and Sites Per Ring for Radial Move. Controls the automatic adjustment of Count, Angle, and Angle Range for Polar Step and Repeat. These three settings are interdependent; each value depends on the values in the other two. Set one of the values and lock it. Set one of the unlocked values; the other unlocked value automatically updates. For example, if you set Angle Range to 360 and Sites Per Ring to 36, Delta Angle updates to 10.
Count	Sets the number of replications in the array.
Angle	Specifies the angle of the replication in the array. Negative values reverse the direction. You can type angle values with .001 degree precision. When you perform an angular replication on a terminal with noncircular pads, the pad stacks are copied and the pad stack offset value is maintained. **Restrictions**: • When you try to rotate a text item or a terminal with a noncircular pad with .001 degree precision, replication angles are rounded to the nearest whole degree value. • When replicating square-shaped pads, PADS Layout converts them to rectangular-shaped pads during angular replication.
Angle Range	Sets the range within which you want to place objects. 360 sets a full circle grid or array; smaller values set sector-shaped grids or arrays.

Radial Tab

Use the Radial tab in the Step and Repeat dialog box to create radial arrays.

See also: Incrementing Texts and Pin Numbering, Using Step and Repeat

Figure 45-353. Step and Repeat Dialog Box Radial Tab

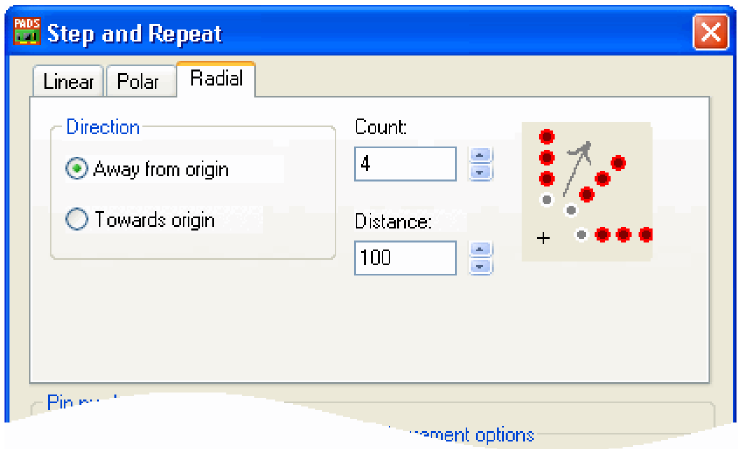


Table 45-321. Radial Tab Contents

Command	Description
Direction	Sets the direction of replication in the array; away from the PCB Decal Editor origin (Away From Origin) or towards the PCB Decal Editor origin (Towards Origin).
Count	Sets the number of replications in the array.
Distance	Specifies the linear distance, or the distance along the radius, between neighboring copies of each replicated object. Negative values reverse the direction.

Incrementing Texts and Pin Numbering

Figure 45-354. Step and Repeat Dialog Box - Texts

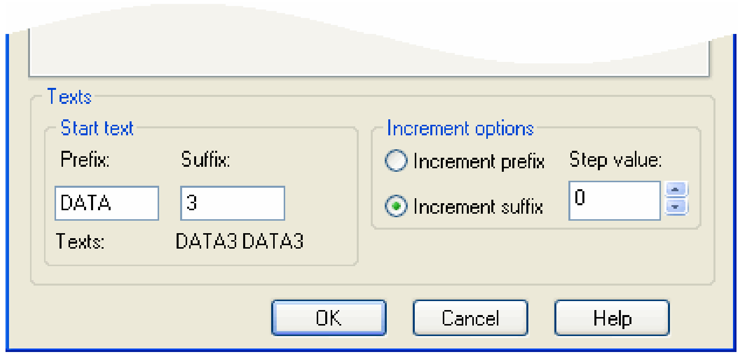


Table 45-322. Texts Area Contents

Command	Description
Prefix/Suffix	For a single text string, use either Prefix or Suffix box, and void the other box. Use both boxes if you want to increment one of the values. A preview of the texts based on your input is displayed below the boxes. Alphabetic and numeric values can be used in either box.
Increment prefix/ Increment suffix	Choose to increment the value of either the Prefix or Suffix box.
Step value	Type a positive or negative number by which to increase or decrease the Text string with consecutive or stepped values. You can use a value of zero to repeat the exact same text and prevent the text string from incrementing. **Restriction:** Step value must be in the range -10 to +10.

Figure 45-355. Step and Repeat Dialog Box - Pin Numbering

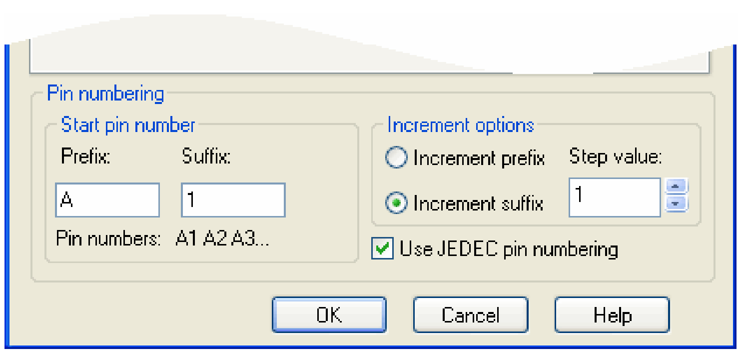


Table 45-323. Pin Numbering Area Contents

Command	Description
Prefix/Suffix	For a single pin number, use either Prefix or Suffix box, and void the other box. Use both boxes if you want to increment one of the values. A preview of the pin numbers based on your input is displayed below the boxes. For example, A1 or 1A. Alphabetic and numeric values can be used in either box.
Increment prefix/ Increment suffix	Choose to increment the value of either the Prefix or Suffix box.
Step value	Type a positive or negative number by which to increase or decrease the pin number with consecutive or stepped values. **Restriction:** Step value must non-zero and be in the range -10 to +10. Zero would replicate a single pin number and is not allowed.
Use JEDEC pin numbering	If using alphanumerics, you can select the **Use JEDEC pin numbering** check box to ensure that legal alphanumeric values are used. **Tip:** This option only ensures that legal alpha and numeric combinations are used. To arrange rows and columns according to JEDEC, use the Assign JEDEC Pinning option on the Tools Menu.

Synchronize Die Part Dialog Box

Use the Synchronize Die Part dialog box to update die part data in PADS Layout with die part data from Library IQ. You can also use the Synchronize Die Part dialog box to update die part data in Library IQ with die part data from PADS Layout.

Restriction: This information applies to only the BGA toolkit.

Note: Beginning with PADS 9.0, dies and flip chips are identified by the Special Purpose settings in the Part Type rather than by the DIE and FLP logic families. With this change, *any* reference designator (logic family) can be assigned to a die or flip chip. If you are exporting to LIQ, and you have dies or flip chips of a family other than DIE or FLP in your design, remember that *all* parts lose their family designation when exported to LIQ, and are assigned either the DIE or FLP family when imported back into PADS Layout; so the original family designation (and reference designator) of these parts is lost in the export/import process.

Accessing

- **BGA Toolbar** button > **Synchronize Die Parts** button

Figure 45-356. Synchronize Die Parts Dialog Box

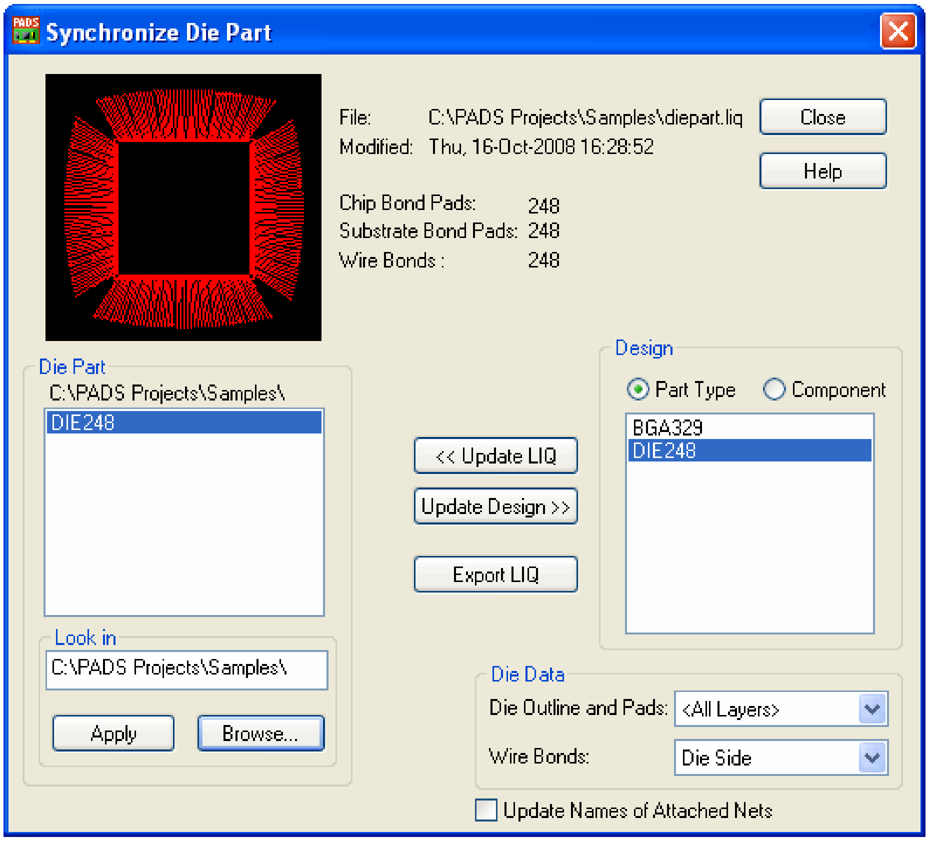
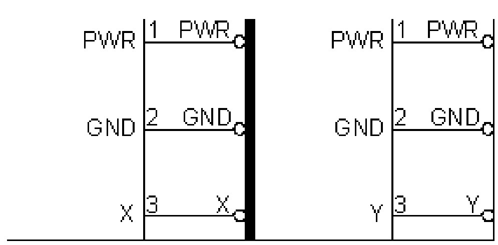


Table 45-324. Synchronize Die Parts Dialog Box contents

Name	Description
Preview area	Displays the Library IQ die part selected in the Die Part list.
File	The die part filename
Modified	The last modification date
Chip Bond Pads	The number of chip bond pads in the die part
Substrate Bond Pads	The number of substrate bond pads in the die part
Wire Bonds	The number of wire bonds in the die part
Flip Chip	The Flip Chip identification for the die part
File	The die part filename
Modified	The last modification date
Chip Bond Pads	The number of chip bond pads in the die part
Die Part	Lists all Library IQ die parts in the current folder.

Table 45-324. Synchronize Die Parts Dialog Box contents (cont.)

Name	Description
Look in	Displays the die part search folder. **Tip:** Click Browse to look for a folder.
Apply button	Sets the die part search folder.
Update LIQ button	Updates die part data in Library IQ with die part data from the design.
Update Design button	Updates die part data in the design with die part data from Library IQ.
Export LIQ button	Saves die part data files to the default \My Documents\PADS Projects folder. The .liq extension is automatically added to the saved file.
Design area	• **Part Type—**Lists all die part components in the design by part type. • **Component—**Lists all die part components in the design by reference designator.
Die Outline and Pads	Sets the layer on which the die outline and pads appear. Select a layer from the list.
Wire Bonds	Sets the layer on which the wire bonds appear. Select a layer from the list.
Update Names of Attached Nets	The name of the net is updated when the following conditions occur: • The net has the same name as the updated pin name. • There are no other component pins in the net with a pin name matching the netname. • The new netname doesn't duplicate netnames in the design. The graphic on the left shows a die part with pin names and netnames. If the previously described conditions are met when you update die parts, the graphic on the right occurs. This only occurs when this option is selected. 

Related Topics

To Synchronize Die Parts with LIQ

To Update Die Parts in Library IQ

Tack or Trace Corner Properties Dialog Box

Use the Tack or Trace Corner Properties dialog box to obtain information on and to modify a selected trace corner or tack.

Accessing

- Select a trace corner > **right-click** > **Properties**.

Figure 45-357. Tack Properties Dialog Box

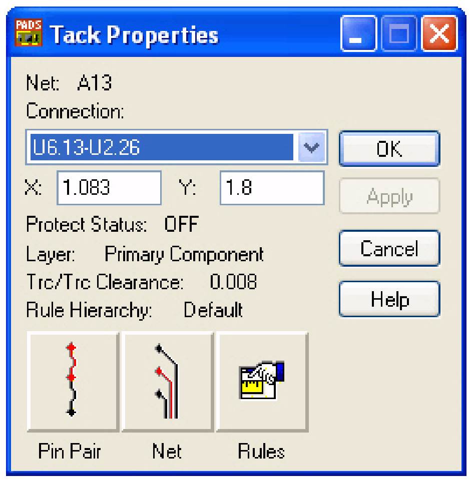


Table 45-325. Tack Properties Dialog Box contents

Name	Description
Net	Displays the name of the net.
Connection	Lists the connections available in the design.
X/Y	The X and Y location of the tack. Type in these fields to change the location.
Project Status	Shows whether the net (to which the tack belongs) is protected.
Layer	Displays the layer on which the tack is located.
Trc/Trc Clearance	Displays the trace to trace clearance for this tack.
Rule Hierarchy	Displays the rule hierarchy for this tack.
Pin Pair button	Opens the Pin Pair Properties dialog box.
Net button	Opens the Net Properties dialog box.
Rules button	Opens the Pin Pair Rules dialog box.

Related Topics

Modifying Trace Corner or Tack Properties

Teardrop Properties on Traces Dialog Box

Use the Teardrop Properties on Traces dialog box to modify the teardrop shape, length ratio, and width ratio for any selected teardrop, teardrops on all layers, or all teardrops. You can also remove an individual teardrop from a design.

Accessing

- Select the trace to which the teardrop is attached > **Right-click** > **Properties**

Figure 45-358. Teardrop Properties on Traces Dialog Box

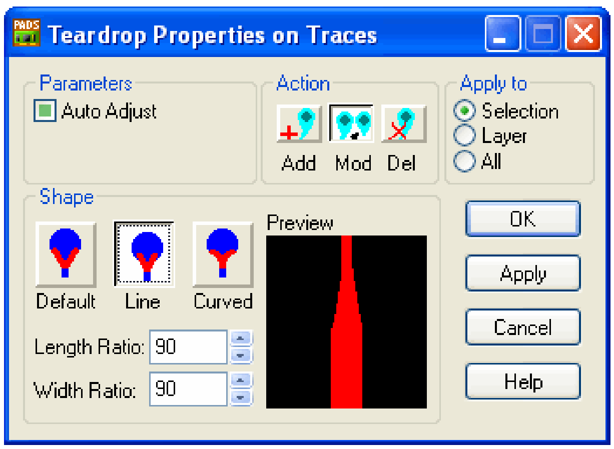


Table 45-326. Teardrop Properties on Traces Dialog Box contents

Name	Description
Auto Adjust	Use Auto Adjust to set a custom length and width ratio. With Auto Adjust selected, PADS Layout attempts to adjust the length of the teardrop on traces where the trace corner is inside the pad or via or the segment is too short to contain the specified length ratio.

Table 45-326. Teardrop Properties on Traces Dialog Box contents (cont.)

Name	Description
Action area	• **Add**—Adds a teardrop to the selected trace or to the setting in the Apply To area. Once you add a teardrop, you can modify its settings by choosing Modify. When you add teardrops to several traces, those traces already containing teardrops retain the existing teardrops with their original settings. • **Mod**—Modifies the teardrop on the selected trace and applies changes to what you click in the Apply to area. • **Del**—Deletes teardrops from the selected trace and applies changes to the setting in the Apply To area.
Apply to area	Applies changes in the Teardrop Properties dialog box to teardrops within the Selection, the Layer, or All teardrops.
Default button	Uses the standard teardrop shape from previous PADS Layout (that is, PowerPCB) versions. You cannot set a length or width ratio with a Default-shaped teardrop.
Line button	Uses a line-shaped teardrop. You can set a length and width ratio for this teardrop. You may want to use line- or curved-shaped teardrops on high-frequency analog boards or very dense boards for smoother connections.
Curved button	Uses a curved-shaped teardrop. You can set a length and width ratio for this teardrop. You may want to use curved- or line-shaped teardrops on high-frequency analog boards or very dense boards for smoother connections.
Length ratio	Sets the length of the teardrop relative to the pad to which it is attached. You cannot set a ratio over 1000. The formula to calculate the length ratio is: (pad diameter) * (length ratio in %) = length of the teardrop For example, if the length ratio is 200 (200% of the pad diameter) and the pad diameter is 60 mils, then the length of the teardrop is 120 mils.
Width ratio	Sets the width of the teardrop relative to the pad to which it is attached. You cannot set a ratio over 100.
Preview	Shows the currently configured teardrop.

Related Topics

Modifying Teardrop Properties

Using Teardrops

Templates Dialog Box

When you are creating an archive, you can use an archive configuration template to set all the fields in the Add Archive to Vault dialog box.

Use the Templates dialog box to select a template for creating a new archive, to create a new template, or to edit an existing template.

An archive template specifies:

- The schematic project (.prj) file
- Zero or more layout project (.pcb) files
- Zero or more additional files to archive
- A name and text description to identify the archive in the vault (optional)

Tip: When specifying filenames in the Templates dialog box, you can enter the variable “<projectname>”; this variable will be replaced by the name of the Vault Project when you use the template. For example, if you enter “<projectname>.pcb” in the Layout project files field, when you use this template to archive the Processor project, “Processor.pcb” will appear in the Add Archive to Vault dialog box; but when you use it to archive the Timer project, “Timer.pcb” will appear.

Accessing

- **Working Folder view > Add Archive to Vault button >** click **Select**, **New**, or **Edit** from the Templates drop-down list

Figure 45-359. Templates Dialog Box

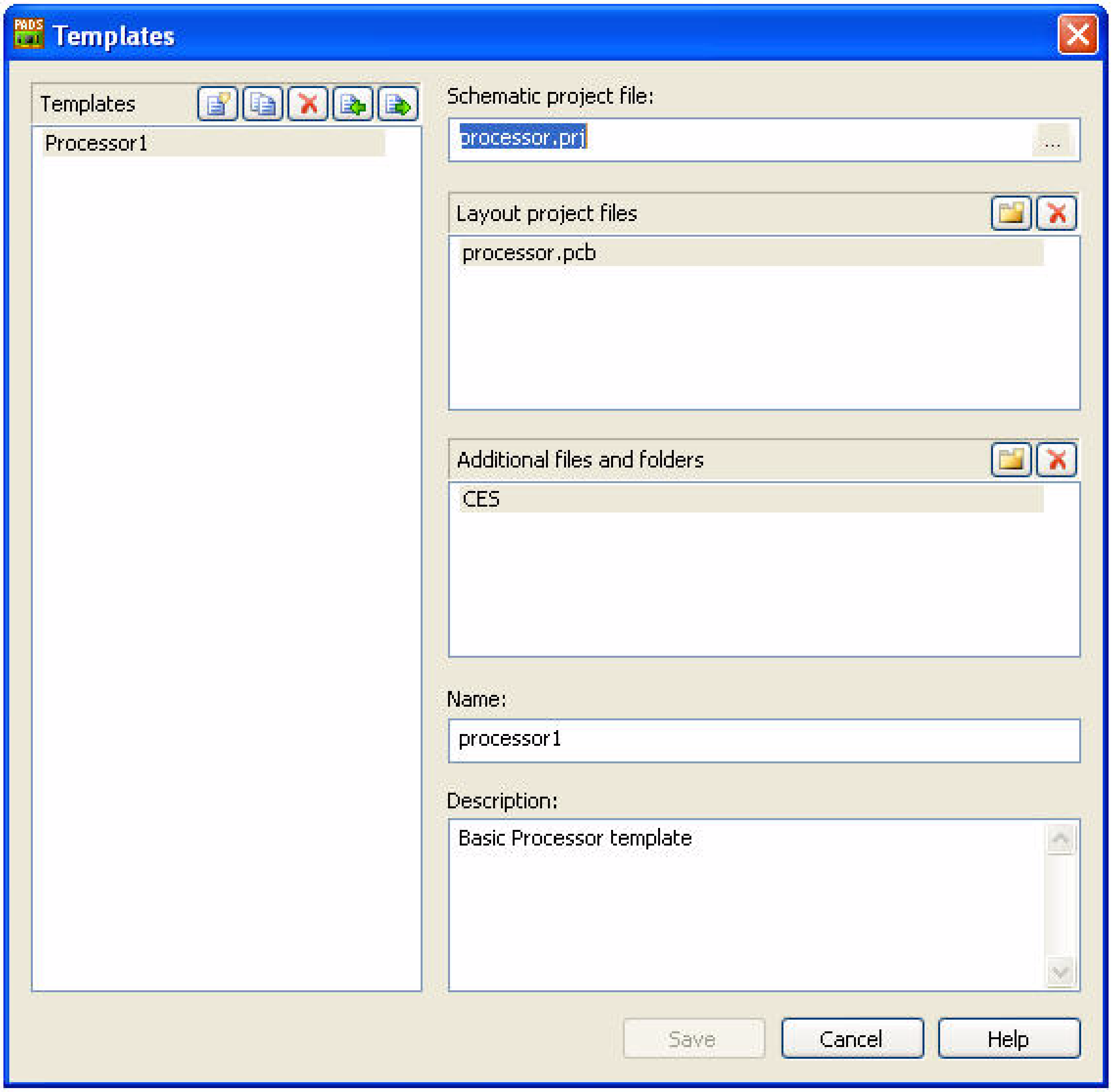


Table 45-327. Templates Dialog Box Contents

Name	Description
Templates list	Lists templates available for the current project.
 **New (Insert) button**	Adds a new item to the list. (Appears only when editing or creating a template.)
 Copy button	Copies the settings of the selected template. (Appears only when editing or creating a template.)

Table 45-327. Templates Dialog Box Contents (cont.)

Name	Description
Delete button	Deletes the selected list item. (Appears only when editing or creating a template.)
Import button	Opens a dialog box where you can select a previously exported template file to add to the list of available templates. (Appears only when editing or creating a template.)
Export button	Opens a dialog box where you can name, and select a location in which to save, the selected template. (Appears only when editing or creating a template.)
... **Browse button**	Opens a browser dialog box where you can select a file or folder to add to the template. (Appears only when editing or creating a template.)
Schematic project file (.prj)	Specifies the DxDesigner .prj file to be saved by the template. **Tip:** If you enter the variable "<projectname>" in this field, it will be replaced in the Add Archive to Vault dialog box with the name of the project you are archiving.
Layout project files (.pcb)	Specifies the PADS Layout .pcb file(s) to be saved by the template. **Tip:** If you enter the variable "<projectname>" in this field, it will be replaced in the Add Archive to Vault dialog box with the name of the project you are archiving.
Additional files and folders	Specifies other files and folders to be saved by the template. **Tip:** If you enter the variable "<projectname>" in this field, it will be replaced in the Add Archive to Vault dialog box with the name of the project you are archiving.
Name	Specifies a name for the template.
Description	Specifies a description of the template.
Save	Saves the new or edited template to the vault. (Appears only when editing or creating a template.)
Select	Specifies to use the selected template for the new archive. (Appears only when selecting a template.)

Terminal Number Properties Dialog Box

Use the Terminal Number Properties dialog box to modify the properties of the terminal's number.

Accessing

- **PCB Decal Editor** > select a terminal number > **right-click** > **Properties**

Figure 45-360. Terminal Number Properties Dialog Box

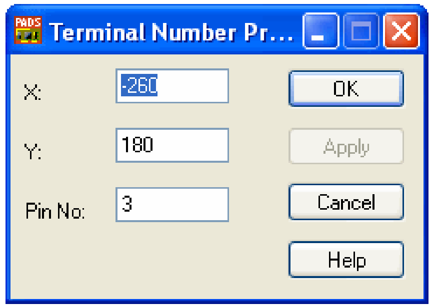


Table 45-328. Terminal Number Properties Dialog Box Contents

Name	Description
X, Y	The X,Y coordinates of the selected terminal number.
Pin No	The pin number of the selected terminal number

Related Topics

Modifying Terminal Number Properties

Terminal Properties Dialog Box

Use the Terminal Properties dialog box to modify terminal properties:

Accessing

- **PCB Decal Editor** > select a terminal > **right-click** > **Properties**

Figure 45-361. Terminal Properties Dialog Box

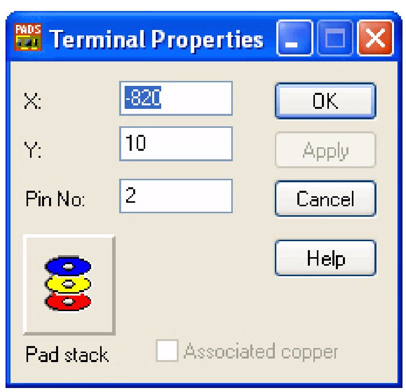


Table 45-329. Terminal Properties Dialog Box Contents

Name	Description
X, Y	The X,Y coordinates of the selected terminal.
Pin No	The pin number of the selected terminal
Pad stack	Opens the Pad Stack Properties for Pin dialog box.
Associated copper	Clear the check box to disassociate copper from the terminal. **Restriction:** The check box is only used to disassociate copper. For instructions on associating copper, see Associating Copper with Terminals.

Related Topics

Modifying Terminal Properties

Text Properties Dialog Box

Use the Text Properties dialog box to modify free text. You can change the font, font style, layer assignment, orientation, rotation, size, line width, if it is mirrored, and justification. You can also access the parent object if the text string is combined with a drafting object.

Restriction: Text can only be added one line at a time into the design. See the following topic for a tip on saving multiple lines of text to the library for reuse, Creating Reusable Fabrication Notes.

Accessing

- **Select a union** > **Right-click** > **Properties**

Figure 45-362. Text Properties Dialog Box

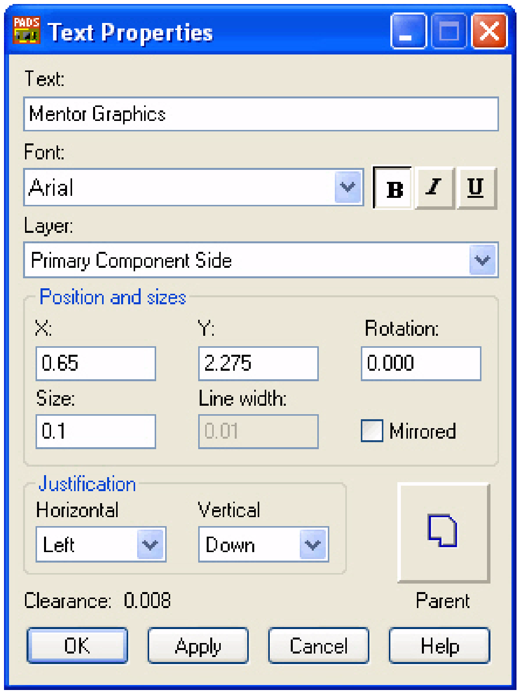


Table 45-330. Text Properties Dialog Box contents

Name	Description
Text	The text string you want to use. **Tip:** There is a maximum of 128 characters per text string.
Font	The fonts available to you. **Tips:** • Select stroke font or a system font. • For system fonts, you can also click a font style button, or any combination of styles: **B** for bold, **I** for italic, or **U** for underlined.
Layer	The layers available to you on which to place the text.
X,Y	Lists the X and Y location of the text. Type new values to change the location.
Rotation	Specifies the rotation angle of the text.

Table 45-330. Text Properties Dialog Box contents (cont.)

Name	Description
Size	Specifies the size of the font. **Size (pts):** This is font size in points and appears for system fonts **Size (mils):** This is font character height and appears for stroke fonts. The size refers to the height of the tallest characters. 
Line Width	Specifies the line width for stroke fonts only. 
Mirrored	Flips the label - text is considered readable from the bottom side of the board.
Horizontal, Vertical	Sets the horizontal and vertical justification of the text to ensure proper positioning between objects when text, attribute values, size, or width change. **Tips:** • For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**. • Optionally, set justification by selecting the text, then right-clicking and clicking **Justify Horizontally**, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking **Justify Vertically**, and then clicking **Up**, **Center**, or **Down**.
Parent	Opens the Drafting Properties dialog box for the parent object if a text string is combined with a drafting object.
Clearance	Specifies clearance values between the text and objects around it.

Related Topics

Modifying Text Properties

To SPECCTRA Dialog Box

Use the To SPECCTRA dialog box to translate a .pcb design file into a SPECCTRA design file.

Accessing

- Use Windows Explorer to navigate to your ...\SDD_HOME\Programs directory, and double-click **pads2sp.exe** > **To SPECCTRA button**

Figure 45-363. To SPECCTRA Dialog Box

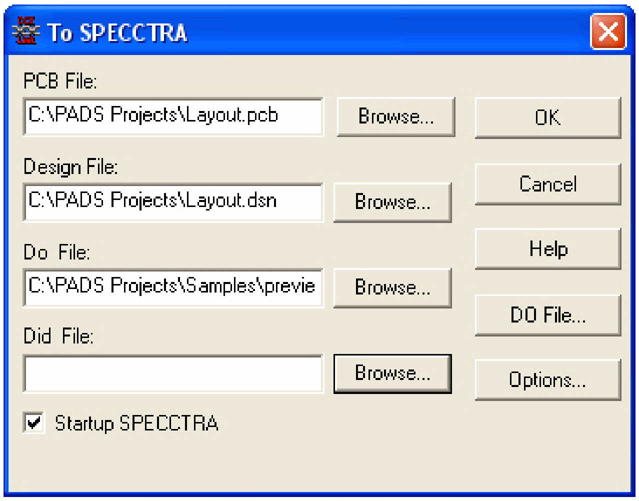


Table 45-331. To SPECCTRA Dialog Box contents

Name	Description
PCB File	Specifies the file to send to SPECCTRA. **Tip:** Click Browse to locate the file.
Design File	Specifies the design file (.dsn) that SPECCTRA inputs. **Tip:** Click Browse to locate the file.
Do File	Specifies the .do file to send to SPECCTRA. The .do file is the script file that controls SPECCTRA operation. **Tip:** Click Browse to locate the file.
Did File	Specifies the output file (.did) that SPECCTRA creates. This file serves as an input .do file in a subsequent SPECCTRA session. **Tip:** Click Browse to locate the file.
Startup SPECCTRA	Specifies to start SPECCTRA after the batch conversion is complete.
DO File button	Opens the SPECCTRA DO File dialog box.

Table 45-331. To SPECCTRA Dialog Box contents (cont.)

Name	Description
Options button	Opens the Options dialog box.

Related Topics

Translating Design Data from PADS Layout to SPECCTRA

Trace Copy Dialog Box

The Trace Copy dialog boxes appear when you perform a group operation that causes an illegal trace installation. These dialog boxes may appear when you move, rotate, or paste a group.

The dialog boxes describe the error with the trace and allow you to manage the error on the fly.

Figure 45-364. Trace Copy Dialog Box

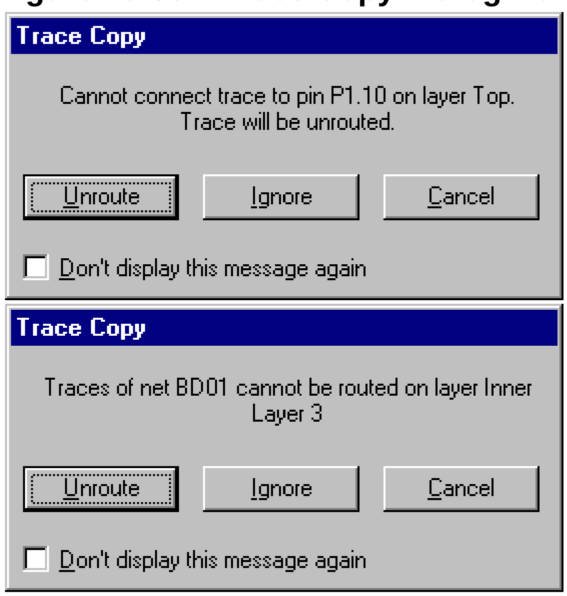


Table 45-332. Trace Copy Dialog Box contents

Name	Description
Unroute button	Creates an unroute instead of installing the trace.
Ignore button	Ignores the error and installs the trace. You are responsible for manually correcting the error.
Don't display this message again check box	Applies the action you click in this dialog box to all subsequent traces and prevents this dialog box from appearing again.

Trace Loop Created Dialog Box

Use the Trace Loop Created dialog box to delete a trace loop, cycle through which trace in the loop you want to delete, or keep the loop.

Accessing

- Automatically opens when you create a trace loop.

Figure 45-365. Trace Loop Created Dialog Box

Table 45-333. Trace Loop Created Dialog Box contents

Name	Description
OK	Specifies to delete the loop that was created.
Cycle	Specifies to cycle through the traces in the loop to determine which one you want to delete.
Create loop	Specifies to not delete the loop you just created.

Trace Properties Dialog Box

Use the Trace Properties dialog box to edit connection information, coordinate locations for both corners, segment length, and layer information. You can modify the trace width and beginning or ending coordinates on routed traces.

Accessing

- Select a trace > right-click > **Properties**.

Figure 45-366. Trace Properties Dialog Box

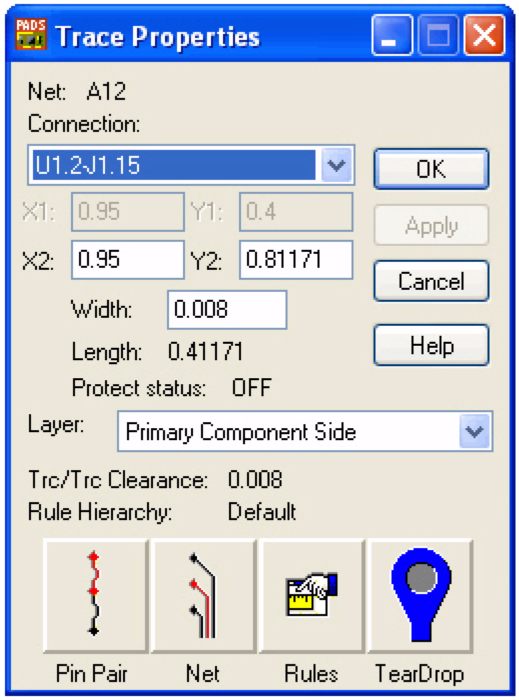


Table 45-334. Trace Properties Dialog Box contents

Name	Description
Net	Displays the name of the net to which this trace belongs.
Connection	Lists where the trace is connected.
X1/Y1	The X and Y starting location of the trace. Type in these fields to change the location.
X2/Y2	The X and Y ending location of the trace. Type in these fields to change the location.
Width	Specifies the width of the trace. **Restriction:** You cannot enter a value that isn't within the minimum and maximum trace width settings in the Clearance Rules. If you enter an illegal value, you get the error message, "Wrong width value."
Length	Displays the routed length of the trace.
Protect Status	Shows whether the net (to which the trace or trace corner belongs) is protected.
Layer	Lists the layer on which the trace or trace corner is located. You can also change the layer on which the trace resides. **See also:** Modifying Trace Segment Properties

Table 45-334. Trace Properties Dialog Box contents (cont.)

Name	Description
Trc/Trc Clearance	Displays the trace to trace clearance for this trace.
Rule Hierarchy	Displays which rules are apply to the trace.
Pin Pair button	Opens the Pin Pair Properties dialog box.
Net button	Opens the Net Properties dialog box.
Rules button	Opens the Pin Pair Rules dialog box.
TearDrop button	Opens the Teardrop Properties on Traces dialog box.

Related Topics

Modifying Trace Segment Properties

Union Properties Dialog Box

Use the Union Properties dialog box to obtain information on and to modify a selected union.

Accessing

- **Select a union** > **Right-click** > **Properties**

Figure 45-367. Union Properties Dialog Box

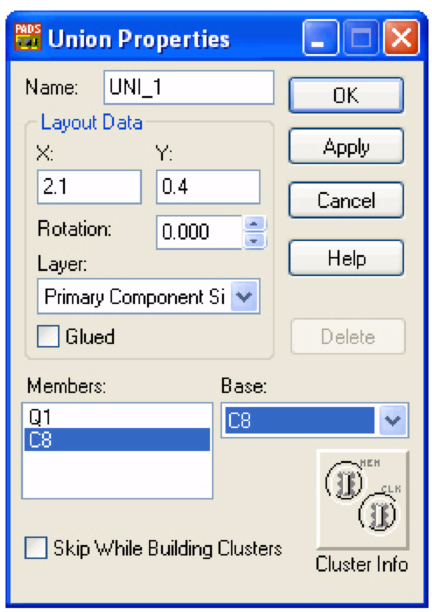


Table 45-335. Union Properties Dialog Box contents

Name	Description
Name	Name of the currently selected union. To rename the union type a new name.
X/Y Coordinates	Current coordinates of the union. To move the union to a new location type new values.
Rotation	Current rotation of the union. For a different rotation angle type new values.
Layer list	Layer on which the union members exist. To flip the union click a different layer.
Glued	Prevents the union from moving through manual or automatic placement processes.
Members	Individual parts that are members of the selected union.
Base list	Identifies the part used to determine the XY location of the union.
Skip while Building Cluster	Ignores the union or cluster during Grow Incremental and Grow Automatic operations.
Cluster Info	Opens the Cluster Information Properties dialog box.

Related Topics

Managing Unions

Update from Library Dialog Box

Use the Update from Library dialog box:

- To update a design with part types and decals from the library, or
- To compare part types and decals in a design with those in the library.

Tip: To update only selected components, select them before opening the dialog box.

Accessing

- **Tools** menu > **Update from Library**

Figure 45-368. Update from Library Dialog Box

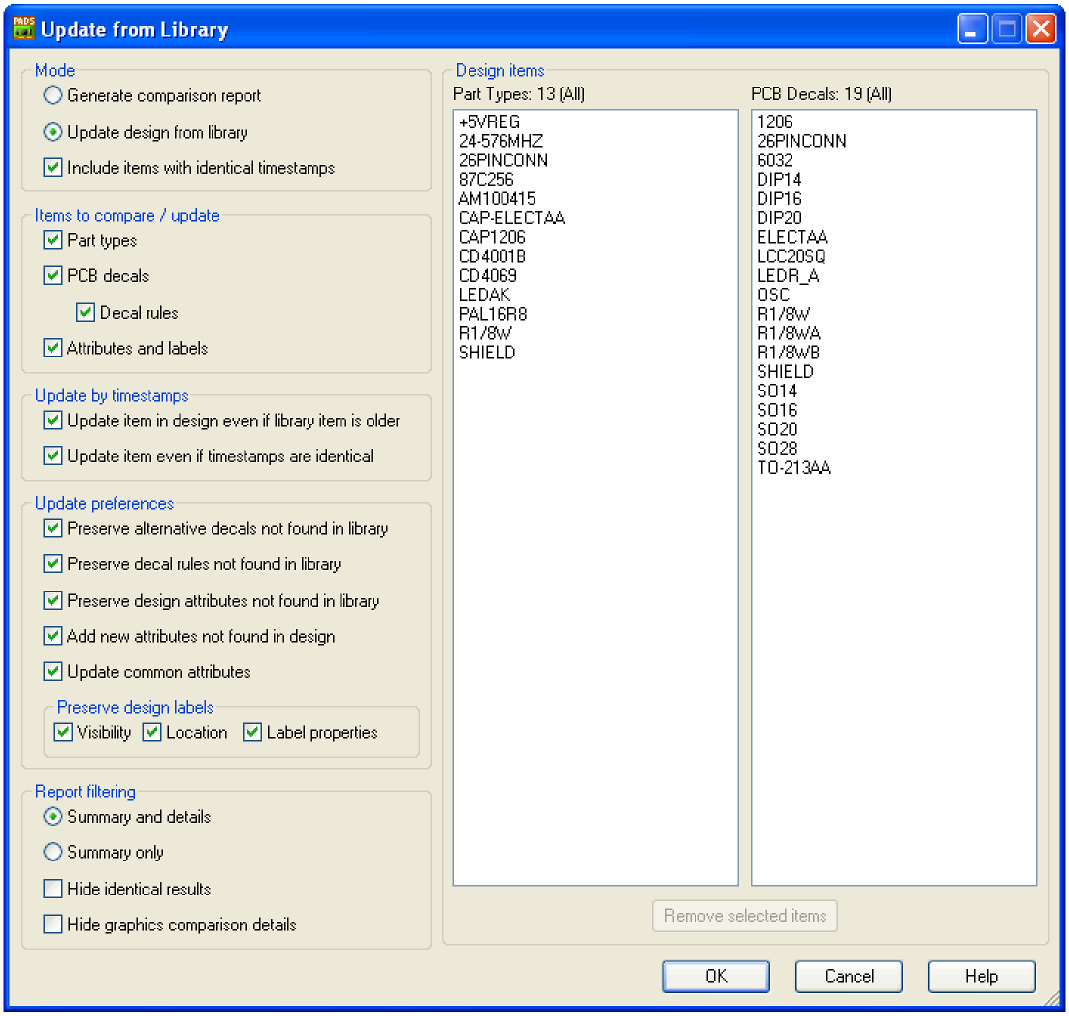


Table 45-336. Update from Library Dialog Box Settings

Control	Description
Generate comparison report	Select to compare library and design items and generate a report file. No changes are made to the design.
Update design from library	Select to compare library and design items, update the design from the library, and generate a report file.
Include items with identical timestamps	Select this check box to include in the compare/update items whose timestamps are the same in the library and design. Clear it to exclude these items from the compare/update. **Tip:** It is possible for items with identical timestamps to have different content in the library and design. For example, if you export a design to an ascii file, manually edit a part type in the ascii file, and import the design back into PADS Layout, the timestamp of the part type will be unchanged, but the content will be different.
Part types	Select this check box to include part types in the comparison/update.
PCB decals	Select this check box to include PCB decals in the comparison/update. Select this check box to update design components with the alternate decals information from the PCB Decals tab of the Part Information dialog box.
Decal rules	Select this check box to include decal rules in the comparison/update.
Attributes and labels	Select this check box to include part type attributes and decal attributes and labels in the comparison/update. Clear it to exclude them. **Tip:** A decal’s standard Name and Part labels—not just attribute labels—are included in the comparison/update.
Update item in design even if library item is older	Select this check box to update a design item even when its timestamp is newer than the library item’s timestamp. Clear it to prevent replacing a design item with an older library item.
Update item even if timestamps are identical	Select this check box to update a design item even when its timestamp is identical to the library item’s timestamp. **Tip:** If this check box is not set, design items with identical timestamps will not be updated, even if their content is different.

Table 45-336. Update from Library Dialog Box Settings

Control	Description
Preserve alternative decals not found in library	Use this check box to specify what you want to do with alternative decals in a design part type that do not exist in the corresponding library part type: • Select the check box to preserve the alternative decals in the design part type. • Clear the check box to remove them from the design part type (that is, leave only the alternative decals that exist in the library version).
Preserve decal rules not found in library	Use this check box to specify what you want to do with decal rule sets that are found in the design version of a decal, but not in the library version: • Select the check box to preserve these rule sets in the design decal. • Clear it to remove them from the design decal.
Preserve design attributes not found in library	Use this check box to specify what you want to do with attributes that are found in the design but not in the library: • Select the check box to keep these attributes in the design item. • Clear it to remove them from the design item.
Add new attributes not found in design	Use this check box to specify what you want to do with attributes that are found in the library but not in the design. • Select the check box to add these attributes to the design item. • Clear it to preserve the design item as it is (that is, do not add them to the design item.
Update common attributes	Use this check box to specify what you want to do with attributes found in both the design and the library. • Select the check box to update the design attribute with the library attribute’s values. • Clear it to preserve the design attribute’s values (that is, do not update them).

Table 45-336. Update from Library Dialog Box Settings

Control	Description
Visibility Location Label properties	Select the attribute properties you want to preserve in the design: • Visibility—Specifies whether and how an attribute associated with a label is displayed (none, value, name and value, full name and value). • Location—Specifies label x,y coordinates, layer, and rotation. • Label properties—Includes font, justification and right reading settings.
Summary and details Summary only	See How to Read the Update Report.
Hide identical results	Select this check box to see only the differences between library and design items in the Detailed Comparison Data section of the report. Clear it to see all comparison data.
Hide graphics comparison details	Select this check box to *exclude* graphical data of drawings, coppers, associated coppers and decal outlines from the report. Clear it to include graphical data.
Remove selected items	Click this button to remove items currently selected in the Design Items lists. **Tip:** The items appearing in these lists are determined as follows: If one or more design components are selected when Update from Library is started, only the part types and decals of the selected components appear in the lists. If no components are selected, the part types and decals of all design components appear in the lists.

Related Topics

Updating a Design from the Library

Update Pin Gate Dialog Box

Use the Update Pin Gate dialog box to update the Pin Group column of selected pins in the Pins table on the Pins tab of the Part Information dialog box.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > **New** > **Pins** tab > select cells in the Pin Group column > **Edit** button

  or

- **File** menu > **Library** > select a Library > **Parts** button > select a part > **Edit** button > **Pins** tab > select cells in the Pin Group column > **Edit** button

Figure 45-369. Update Pin Gate Dialog Box

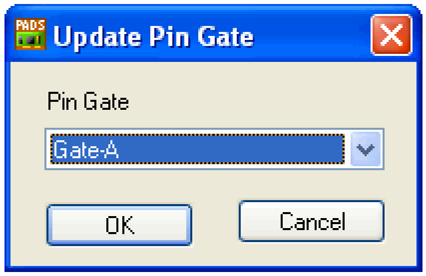


Table 45-337. Update Pin Gate Dialog Box contents

Name	Description
Pin Gate list	Specifies the setting to assign selected pins in the pins table. Possible settings include: **Gate-*n*—**assign to gate pins. This selection is only available if you've added gates on the Gates tab. **Signal pin—**assign to implicit pins (not shown on the schematic) **Unused pin—**assign to unused pins **Connector pin—**assign to connector pins. This selection is only available if the part is set as a Special Purpose > Connector on the General tab.

Update Pin Name Dialog Box

Use the Update Pin Name dialog box to update the Name column of selected pins in the Pins table on the Pins tab of the Part Information dialog box.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > **New** > **Pins** tab > select cells in the Name column > **Edit** button

  or

- **File** menu > **Library** > select a Library > **Parts** button > select a part > **Edit** button > **Pins** tab > select cells in the Name column > **Edit** button

Figure 45-370. Update Pin Name Dialog Box

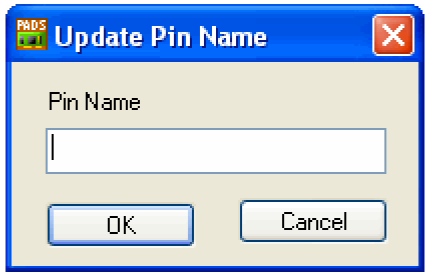


Table 45-338. Update Pin Name Dialog Box contents

Name	Description
Pin Name box	Specifies the name to assign selected pins in the pins table.

Update Pin Swap Dialog Box

Use the Update Pin Swap dialog box to update the Swap column of selected pins in the Pins table on the Pins tab of the Part Information dialog box.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > **New** > **Pins** tab > select cells in the Swap column > **Edit** button

  or

- **File** menu > **Library** > select a Library > **Parts** button > select a part > **Edit** button > **Pins** tab > select cells in the Swap column > **Edit** button

Figure 45-371. Update Pin Swap Dialog Box

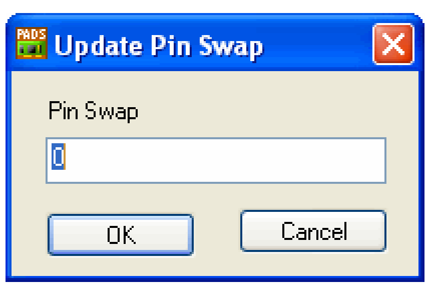


Table 45-339. Update Pin Swap Dialog Box contents

Name	Description
Pin Swap box	Specifies the swap number to assign selected pins in the pins table. Pins with the same swap number can have their connections swapped in the design using the Swap Pin or Auto Swap Pin features on the ECO toolbar.

Update Pin Type Dialog Box

Use the Update Pin Type dialog box to update the Type column of selected pins in the Pins table on the Pins tab of the Part Information dialog box.

Accessing

- **File** menu > **Library** > select a Library > **Parts** button > **New** > **Pins** tab > select cells in the Type column > **Edit** button

  or

- **File** menu > **Library** > select a Library > **Parts** button > select a part > **Edit** button > **Pins** tab > select cells in the Type column > **Edit** button

Figure 45-372. Update Pin Type Dialog Box

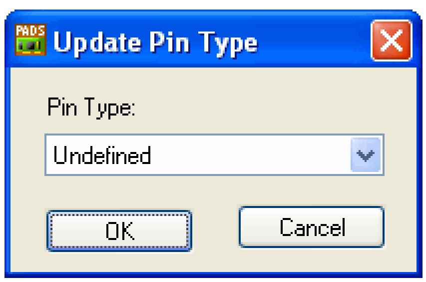


Table 45-340. Update Pin Type Dialog Box contents

Name	Description
Pin Type list	Specifies the type to assign selected pins in the pins table.

Variant/Substitute Dialog Box

The Variant/Substitute dialog box appears when you choose to substitute a component in an assembly variant. When you substitute a component, the substitution is referred to as the active component. The original component that you substituted is referred to as the default. The default component is what exists in the base option and the raw database.

Accessing

1. **Tools** menu > **Assembly Variants**
2. There are several ways to begin substituting a component:
   - Click an Assembly Variant in which you want to substitute the component from the Variant **Name** list, select the component name to change, and click **Substituted** in the **Status** area.

     Or
   - Select a component in the multicolumn list, and click **Substitute** in the **Status** area.

     Or
   - Click **Substitute** from the **Verb Mode** list, and select a component in the Layout Editor.

Figure 45-373. Variant/Substitute Dialog Box

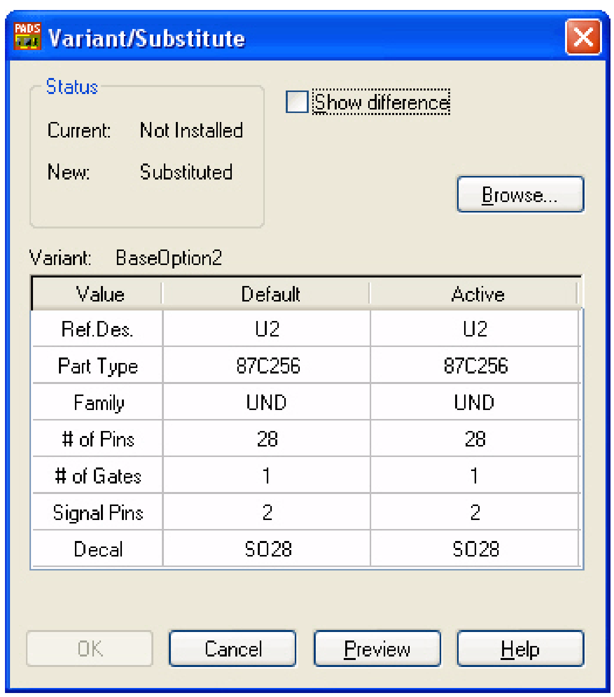


Table 45-341. Variant/Substitute Dialog Box contents

Name	Description
Current	Displays the state before you click Substituted.
New	Displays the state after you click Substituted.
Show difference	Specifies to display only the differing values of the Default and Active component in the Variant table.
Browse	Opens the Get Part Type from Library dialog box.
Variant	Displays the name of the selected variant.
Variant table	Displays the attributes (Value) of the component you are substituting, its Default value, and its Active value (the substitution).
Preview	Opens the Preview for dialog box where you can see the substitutions you've made.

Related Topics

Substitute a Component for Assembly Variants

Verify Design Dialog Box

You can check for individual or all design errors using the Verify Design dialog box. Check for the following types of errors: clearance, connectivity, high speed, number of vias, plane connection, test point, fabrication, wire bond. It does not check reference designator, part type, or attribute labels for clearance violations.

Accessing

- **Tools** menu > **Verify Design**

Figure 45-374. Verify Design Dialog Box

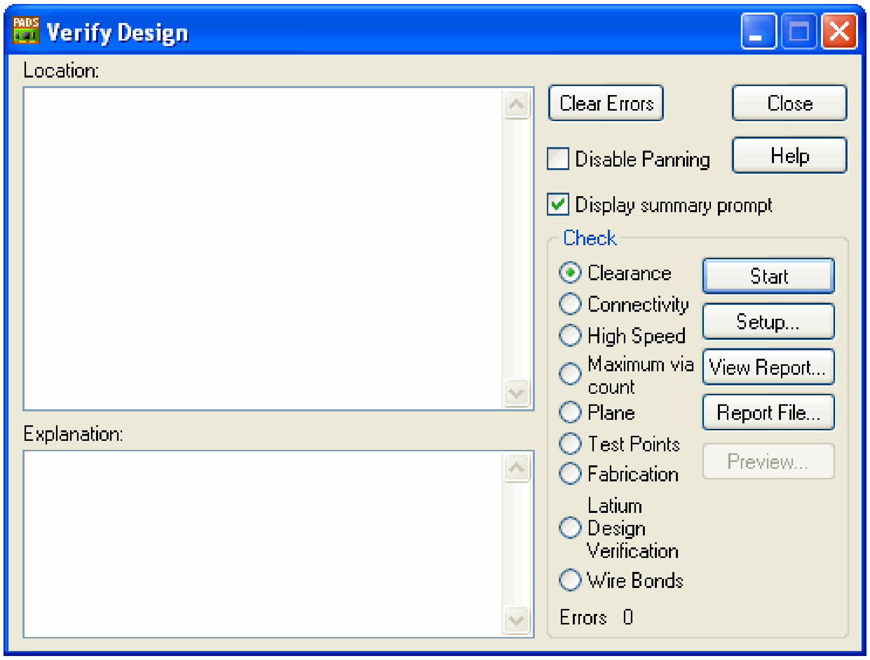


Table 45-342. Verify Design Dialog Box contents

Name	Description
Location list	Lists the error for the type of check you selected in the Check area.
Explanation list	Displays the reason for the error selected in the Location list.
Clear Errors button	Clears the Error Markers. **Important:** This does not clear the actual errors.
Disable Panning	Prevents the design area from panning to the error you select in the Location list.
Display summary prompt	Specifies to display the error-summary prompt window. The prompt window appears after the check has run and lists the total number of errors found.

Table 45-342. Verify Design Dialog Box contents (cont.)

Name	Description
Check area	Specifies the type of check you want to run, set up, view, or specify a report for. • **Clearance**—Performs clearance checking on only the **visible** area of the design. **See also:** Clearance Checking Setup Dialog Box **Tip:** When clearance checking against the board outline, the edge of the object is checked against the centerline of the board outline. • **Connectivity**—Performs connectivity checking on the entire design. **Tips:** • The connectivity check also detects instances where a drill size is larger than the pad it is assigned to. • Connectivity checking recognizes copper as a valid conductor, like a trace. For troubleshooting errors, see Connectivity Errors. • **High Speed**—Performs high-speed checking on the entire design. **See also:** Electrodynamic Check Dialog Box Traces on high-speed printed circuit boards can act like transmission lines that “broadcast” interference to adjacent conductors. Using the high-speed rules module, you can use Rules to set clearances on a net class, net, or pin to pin connection basis; then use high-speed checking to report on properties such as impedance, delay, track length, daisy chaining, and parallel routing. These issues cause interference and create costly problems in prototyping. You can run these checks against the entire board or against specific nets. • **Maximum Via Count**—Performs maximum via count checking on the entire design. • **Plane**—Performs plane checking on the entire design. Checks whether a pad exists in the connecting pad stack for the plane layer. For a pad size to exist it must be more than 0 or the drill size must exceed the pad size. For links to SMD pads, Plane checks whether the pad-to-via connection connects to the plane. **See also:** Mixed Plane Setup Dialog Box **Tip:** To view Plane clearance or connectivity errors after performing a Plane check, return to the Verify Design dialog box and click Clearance or Connectivity.

Table 45-342. Verify Design Dialog Box contents (cont.)

Name	Description
Check area (con't)	• **Test Points**—Performs test point checking on the entire design. Test Points checks for probe clearances, minimum via/pad sizes for probing, SMD pin probing, test points on component pin on the component side, test point count per net settings and nail diameter settings, and compares the settings against the setting you make in the DFT Audit program. **See also:** Performing a Test Point Audit **Tip:** Test point checking is the same whether you enable the checking on the Verify Design dialog box or on the Latium Checking Setup dialog box. If you plan to perform Latium Design Verification, you can eliminate an extra design transfer between PADS Layout and PADS Router by running test point checking along with the other Latium checks. • **Fabrication**—Performs DFF error checking on designs by either using CAM documents in PADS Layout or by using errors that are backward annotated from CAM350. **Requirement:** You need the CAM350 Link license option to use this. On the Help menu, click Installed Options and on the Options tab, check if the option is available in your license. **See also:** Fabrication Checking Setup Dialog Box, Back-annotating CAM350 Files • **Latium Design Verification**—Performs clearance checking on only the visible area of the design. Any Latium rule errors found appear in the Location list. The design is passed to PADS Router to perform the verify design check, and is passed back to PADS Layout when the process is complete. If you used PADS Router to work on your design, it might contain advanced rules (differential pairs, vias at SMD, matched length traces) which only PADS Router can check. The Latium Design Verification: a. Saves the current PADS Layout database to a temporary file b. Starts PADS Router (if not already running) c. Starts the PADS Router Monitor in Verify Design Mode d. Loads the saved PADS Layout design into PADS Router e. Executes the selected checking operations f. Saves the file in PADS Router g. Re-loads the PADS Router file into PADS Layout • **Wire Bonds**—Performs clearance checking on the entire design. Checks wire bond length, width, angle, and the clearance between wire bonds and substrate bond pads for all die parts in the design. **Tip:** Rules for individual die parts are set using the Wire Bond Rules dialog box when in the Wire Bond Editor.

Table 45-342. Verify Design Dialog Box contents (cont.)

Name	Description
Start button	Runs the check with your specified options.
Setup button	Opens the setup dialog box for the check you have selected. • **Clearance**—Clearance Checking Setup dialog box • **Connectivity**—Connectivity Checking Setup dialog box • **High Speed**—Electrodynamic Check dialog box • **Maximum Via Count**—Unavailable • **Plane**—Mixed Plane Setup dialog box • **Test Points**—Unavailable • **Fabrication**—Fabrication Checking Setup dialog box • **Latium Design Verification**—Latium Checking Setup dialog box • **Wire Bonds**—Wire Bond Checking Setup dialog box
View Report button	Opens the report in a text editor for the check you have selected. These reports are saved in the \PADS Projects folder by default. • **Clearance**—clear.lst • **Connectivity**—connect.lst • **High Speed**—hispeed.lst • **Maximum Via Count**—viacount.lst • **Plane**—chtie.lst • **Test Points**—testpt.lst • **Fabrication**—DFF.lst • **Latium Design Verification**—latium.lst • **Wire Bonds**—diecheck.lst
Report File button	Specifies where you want to save your report file. You can set a different location and name for each type of check.
Preview button	After you run a Fabrication check that reports errors, you can preview the CAM layer document associated with the error. The Preview button is available only when fabrication checking is enabled and lists errors.
Errors	Displays the number of errors for the type of check you ran after you click Start.

Related Topics

Verifying the Design (contains error troubleshooting)

Via Properties Dialog Box

The Via Properties dialog box displays the netname to which the via belongs, the via name, coordinates, and where the via is connected.

Accessing

- Select a via > right-click > **Properties**

Figure 45-375. Via Properties Dialog Box

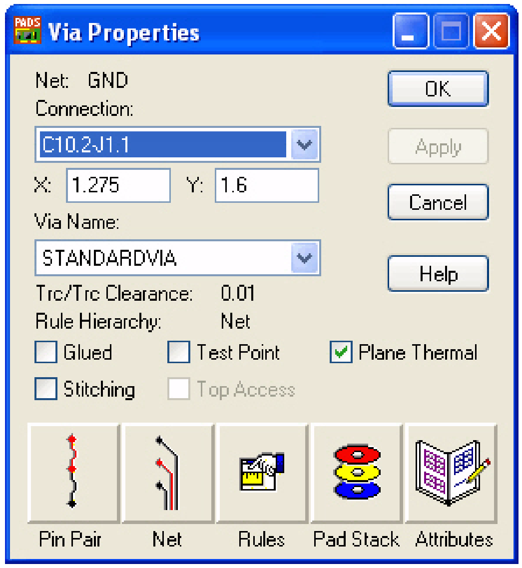


Table 45-343. Via Properties Dialog Box contents

Name	Description
Net	Displays the name of the net to which this via belongs.
Connection	Lists where the via is connected.
X/Y	The X and Y location of the via. Type in these fields to change the location. **Restriction:** These fields are not available when the Stitching check box is selected.
Via Name	Lists the via type. You can click a different type from the list and apply it. **Tip:** If Lock Test Point in the Routing / General page of the Options dialog box is on, you cannot reassign a via type. If you reassign a via type that is a locked test point, the Warning: Test Point Locked dialog box appears.
Trc/Trc Clearance	Displays the trace to trace clearance for this via.
Rule Hierarchy	Displays which rules are apply to the via.

Table 45-343. Via Properties Dialog Box contents (cont.)

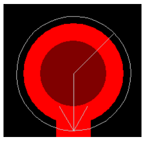
Name	Description
Glued	Glues the via so you cannot move it. If you try to move the via, a message will appear notifying you that the via is glued. You can override the glue setting. You can fix a via's location on the fabrication board by turning on the Glued check box in the Via Properties dialog box. The points of a bed of nails test apparatus are matched to the test point locations, which can be vias. So if you redesign and remanufacture the board, the glued test points prevent the vias from moving in the new design, thus preventing costly retooling of the test equipment.
Test Point	Makes the via or pin a test point. This is a three-state check box that depends on the state of the selected objects. If all of the selected vias or pins are a test point, then it is on. If none of the selected vias or pins are a test point, then it is off. If some of the selected vias or pins are a test point and some are not, then it is undefined. You can make all selected vias or pins test points by turning Test Point on and choosing Apply. You can remove the test point from all selected vias or pins by turning Test Point off and choosing Apply. When you click Apply the pad stack is automatically checked to see if the via or pin can be a test point; for example, you cannot make buried vias test points because a probe cannot access a buried via. **Tip:** When the via or pin is flagged as a test point, and Show Test Points is checked on the Routing / General page of the Options dialog box, an arrow is drawn on it in the design: 
Plane Thermal	Determines whether the pin or via is eligible to receive a thermal. The status of a thermal is set individually by pin and via, and the thermal indicators will appear on plane layers only. Click to clear this check box if you do not want the via or pin to connect to any plane. Once a pin or via is eligible, it is not automatically assigned a thermal attribute. **See also:** Connecting a Net with a Plane, Setting Pins and Vias as Thermals
Stitching	Determines that this is a stitching via. Vias that are not added during the process of routing a trace are marked as stitching vias. These vias are treated differently from regular vias (which can be ripped up or shoved by the dynamic router).

Table 45-343. Via Properties Dialog Box contents (cont.)

Name	Description
Top Access	Attempts to probe the test point from the top and bottom in DFT Audit. The default is bottom; so with Top Access off, DFT Audit automatically tries to probe the test point from the bottom. **See also:** Performing a Test Point Audit When you click Apply, the pad stack is automatically checked to see if top access is valid; for example, you must assign Top Access to partial vias with only top access if you want to use the vias as test points. You can only set the Top Access option if the via or pin is a test point (Test Point is on).
Pin Pair button	Opens the Pin Pair Properties dialog box.
Net button	Opens the Net Properties dialog box.
Rules button	Opens the Pin Pair Rules dialog box.
Pad Stack button	Opens the Pad Stacks Properties dialog box.
Attributes button	Opens the Object Attributes dialog box.

Related Topics

Modifying Via Properties

Vias Dialog Box

Use the Vias dialog box to determine what kind of via to install when you change layers during routing. When you choose a via type, that type is used with the Add Vias and Add Vias to Guide commands. You can also change the via type during routing.

Accessing

- Type **v** > press **Enter**

Figure 45-376. Vias Dialog Box

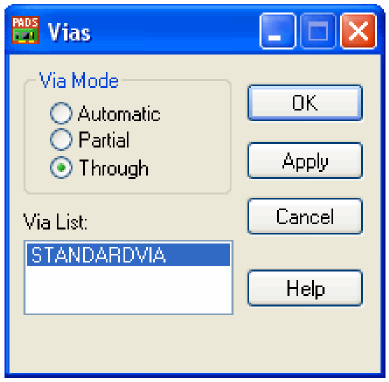


Table 45-344. Vias Dialog Box contents

Name	Description
Via Mode area	• **Automatic**—PADS Layout chooses from all vias, through or partial, that can handle the particular layer change. If PADS Layout finds partial vias dedicated to the layer change, it chooses from them. If PADS Layout can't find a dedicated partial via, it selects any through vias for a through or partial layer change even if the via is not one that is specifically selected in the Via list. It then checks the Routing Rules dialog box for vias that are allowed for the net you are routing. If more than one via still passes, PADS Layout installs the one with the smallest drill diameter or smaller pad size. Automatic allows only vias that begin and end on the Layer Pair shown on the Routing / General page of the Options dialog box. To use automatic via mode, the layer pair for routing and the layer pair for the partial via definition need to match. For example, if you have a partial via set up for layers 1 through 4, and the layer pair for routing is set for layers 1 through 8, automatic mode will not insert a via. • **Partial**—The automatic via selection still occurs, but it is limited to the partial vias only. • **Through**—The list of through vias becomes active. Click the via you want to use as the default and click Apply. This is the via which is installed every time you change between layer pairs.
Via List	Lists through hole vias available in the design; partial vias are not listed.

Related Topics

Setting a Via Type

To Change the Via Type While Routing

View Clearance Dialog Box

Use the View Clearance dialog box to find the minimum clearance between two items.

Accessing

- **View** menu > **Clearance** > click the button for the type of clearance > select two items in the design

Figure 45-377. View Clearance Dialog Box

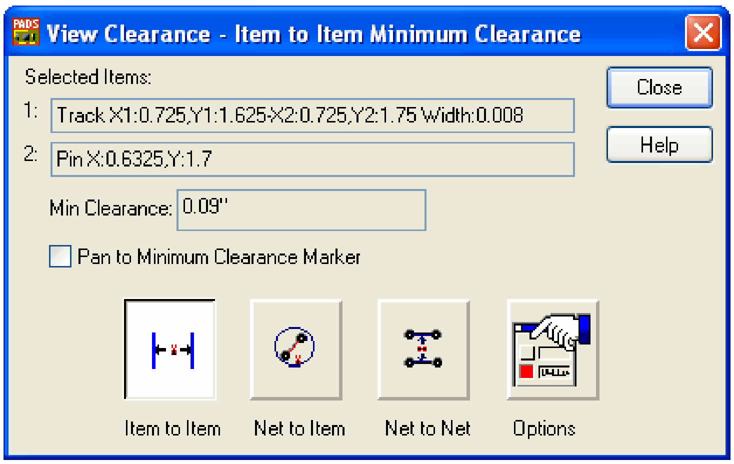


Table 45-345. View Clearance Dialog Box contents

Name	Description
Selected Items	Displays information about selected nets and items. The clearance button you select determines valid selections **Tip:** Jumper outlines, text, and teardrops are not valid item types for clearance.
Min Clearance	Calculates and displays the minimum clearance between the selected items, the selected net and its closest item, or between the selected nets.
Pan to Minimum Clearance Marker	Pans the view so the minimum clearance marker is in the center of the main workspace. This is available only for Net to Item and Net to Net.

Table 45-345. View Clearance Dialog Box contents (cont.)

Name	Description
Item to Item button	Finds the minimum clearance between two selected items. Supported item types are board outlines, pads, vias, jumpers, traces, 2D lines, copper, copper pour, and component outlines. **Tip:** Jumper outlines, text, and teardrops are not valid item types for clearance. Clearances are measured on the current layer. When one or both selected items are not on the current layer, the clearance is measured to the centerline of the item that is not on the current layer.
Net to Item button	Finds the minimum clearance between a selected net and the closest item. Items include: pads, vias, jumpers, traces, 2D lines, copper, copper pour, and component outlines.
Net to Net button	Finds the minimum clearance between two selected nets.
Options button	Opens the Dimensioning / General page of the Options dialog box where you can control the extension lines and arrows used to show minimum clearance. **Tip:** To modify the appearance of the text, lines, arrows, and so on, used to show clearances, use the Dimensioning / General page on the Options dialog box.

Related Topics

To View the Clearance Between Items

To View the Clearance Between a Net and an Item

To View the Clearance Between Nets

View Nets Dialog Box

Use the View Nets dialog box to:

- Selectively hide or display connections.
- Hide or show routed or unrouted paths by netname.

Tip: If View Nets hides unrouted connections, neither Verify Design nor Find can see them. Make sure View Nets does not disable any nets or traces you want to search for.

Accessing

- **View** menu > **Nets**

Figure 45-378. View Nets Dialog Box

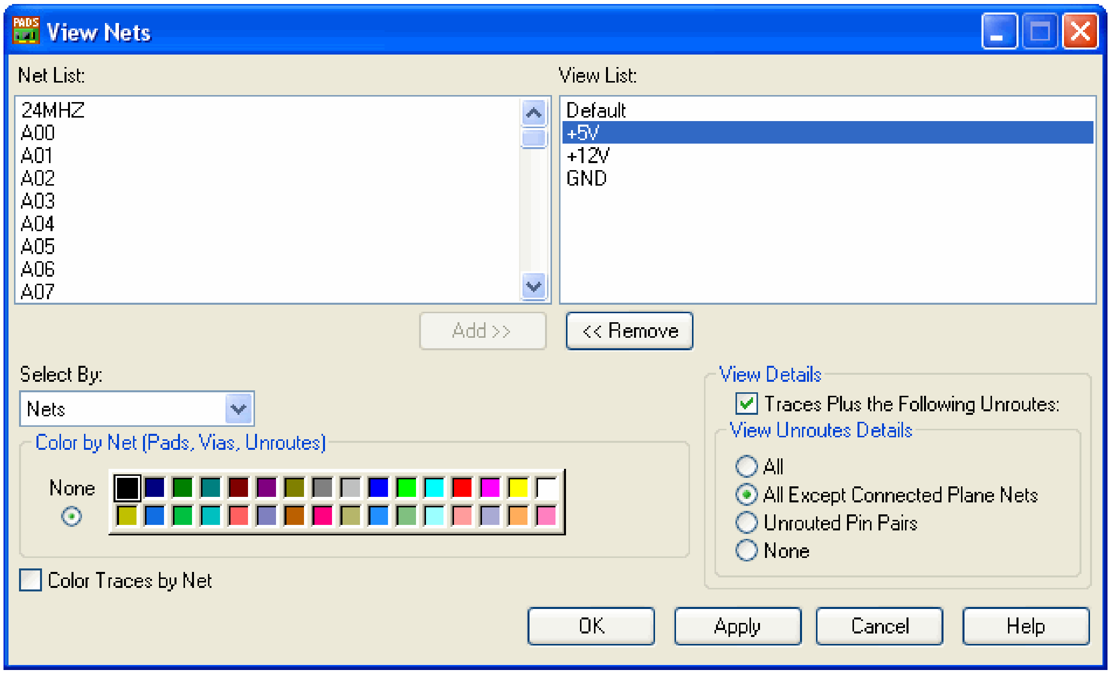


Table 45-346. View Nets Dialog Box contents

Name	Description
Net list	Lists all of the nets available in the design.
View list	Lists all of the nets for which you want to set view properties.
Add/Remove buttons	Add moves selected net names into the View List box where you can highlight net names and set view details for them. Use Ctrl for multiple selections. Click Remove to move the net from the View List to the Net List.
Select By list	Click a command from this list box to change the available nets in the Net List and View List boxes.

Table 45-346. View Nets Dialog Box contents (cont.)

Name	Description
Color by Net area	Sets a color to a selected net in the View Nets list. When you assign a color to a net, pads, unroutes, and vias on all layers, as well as traces optionally appear in that color, making it easier to identify nets when splitting planes. When you split planes, you can assign color to the critical signals and then create correct copper pours with greater ease. All net color assignments are saved with the design. Use Color by Net to: • Assign color to a net class or a class with rules. • Assign power and ground colors and turn off the visibility of their unroutes. Pins appear in the color you set. This aids in the placement of capacitors and resistors; you don't need to view the power and ground unroutes. • Assign color for critical nets so their static unroutes are clearly visible during placement.
Color Traces by Net	Shows traces, for nets in the View List box, using the assigned color. Traces, pins, copper pours, and unroutes appear in the assigned color.
Traces Plus the Following Unroutes check box	Enables filtering of the unrouted connections, where you can set the commands in the View Unroutes Details area.
View Unroute Details area	The Unroute Details area options are: • All—Displays all unroutes. • All Except Connected Plane Nets—Displays all unroutes except for those on plane nets whose connection to the plane have been satisfied. This option does not display unroutes to stitching vias that are embedded in copper pour, split mixed planes, or CAM planes. • Unrouted Pin Pairs—Displays only unroutes on fully unrouted pin pairs. • None—Displays no unroutes.

Related Topics

Modifying Net Properties

To Assign Colors to Nets

Using the Net Properties Dialog Box

Warning: Test Point Locked Dialog Box

This warning dialog box performs different functions depending on whether you are modifying vias, pins, or routes.

Accessing

To access the Warning dialog box, modify any of the following:

- A cluster that contains test points
- Vias, pins, or jumper pins that are locked test points
- Routes that are connected to locked test points

Figure 45-379. Warning Dialog Box

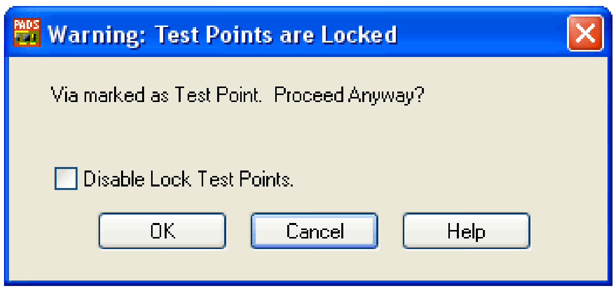


Table 45-347. Warning Dialog Box contents

Name	Description
Disable Lock Test Points	Turns the Lock Test Points check box off in the Routing / General page of the Options dialog box
OK button	Applies the change and maintains the test point status.

Related Topics

Warning: Test Point Locked Dialog Box

Wire Bond Checking Setup Dialog Box

Use the Wire Bond Checking Setup dialog box to specify which wire bond rules to check during a Wire Bonds verification.

Accessing

- **Tools** menu > **Verify Design** > **Wire Bonds** check > **Setup** button

Figure 45-380. Wire Bond Checking Setup Dialog Box

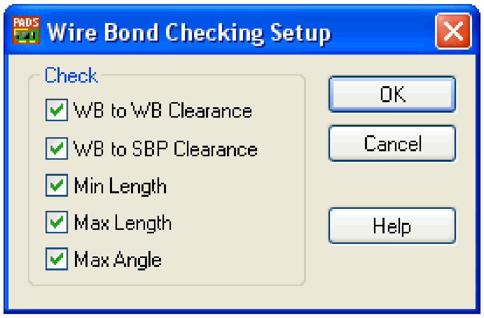


Table 45-348. Wire Bond Checking Setup Dialog Box contents

Name	Description
Wire Bond to Wire Bond Clearance	Assigns the clearance required, in current units, between adjacent wire bonds. All checking is done edge to edge.
Wire Bond to Substrate Bond Pad Clearance	Assigns the clearance required, in current units, between wire bonds and adjacent substrate bond pads. All checking is done edge to edge.
Minimum Length	Assigns the minimum length, in current units, for all wire bonds in the die part.
Maximum Length	Assigns the maximum length, in current units, for all wire bonds in the die part.
Maximum Angle	Assigns the maximum angle, in degrees, at which you can place wire bonds in the die part.

Related Topics

Setting Up Wire Bond Checking

Wire Bond Properties Dialog Box

In the Wire Bond Editor, with one or more wire bonds selected, right-click and click Properties to edit the Wire Bond properties.

Restriction: This information applies to only the BGA toolkit.

Note: If you have multiple objects selected, properties that are not common to all selected objects will not appear.

Accessing

- **BGA Toolbar** button > **Wire Bond Editor** button > Select a wire bond > right-click > **Properties**

Figure 45-381. Wire Bond Properties Dialog Box

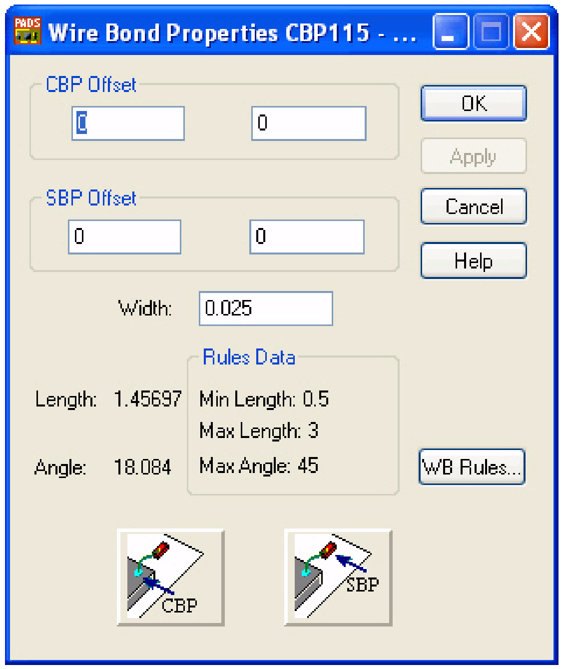


Table 45-349. Wire Bond Properties Dialog Box contents

Name	Description
CBP Offset	Specifies the X and Y values, with respect to the CBP orientation, at which to offset the wire bond from the center of the component bond pad.
SBP Offset	Specifies the X and Y values, with respect to the SBP orientation, at which to offset the wire bond from the center of the substrate bond pad.
Width	Type a width for the wire bond. The valid range is from 0 through 250 mils. If you enter 0, the display line has a width of 0.01 mil.
Length	Displays the length, in current design units, of the currently selected wire bond.
Angle	Displays the angle, in degrees, of the currently selected wire bond.

Table 45-349. Wire Bond Properties Dialog Box contents (cont.)

Name	Description
Rules Data	Displays the current wire bond rules information taken from the Wire Bond Rules dialog box.
CBP button	Opens the CBP Properties dialog box for the component bond pad connected to the currently selected substrate bond pad. This button is unavailable if there is no connected component bond pad.
SBP button	Opens the SBP Properties dialog box for the substrate bond pad connected to the currently selected component bond pad. This button is unavailable if there is no connected substrate bond pad.
WB Rules button	Opens the Wire Bond Rules dialog box.

Wire Bond Rules Dialog Box

Use the Wire Bond Rules dialog box to define rules that apply to wire bonds in the die part. All checking is done edge to edge.

Restriction: This information applies only to the BGA toolkit.

Accessing

- **BGA Toolbar** button > **Wire Bond Wizard** button > **Wire Bond Rules** button

Figure 45-382. Wire Bond Rules Dialog Box

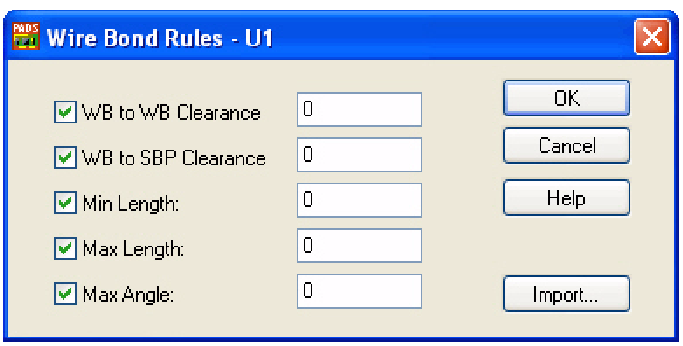


Table 45-350. Wire Bond Rules Dialog Box contents

Name	Description
WB to WB Clearance	Assigns the clearance required, in current units, between adjacent wire bonds. The rule is not checked unless you select it and enter a value. All checking is done edge to edge. **Tip:** Unchecked rules appear in the wire bond report as “Not Set.”
WB to SBP Clearance	Assigns the clearance required, in current units, between wire bonds and adjacent substrate bond pads. The rule is not checked unless you select it and enter a value. All checking is done edge to edge.
Min Length	Assigns the minimum length, in current units, for all wire bonds in the die part. The rule is not checked unless you select it and enter a value. **Tip:** Unchecked rules appear in the wire bond report as “Not Set.”
Max Length	Assigns the maximum length, in current units, for all wire bonds in the die part. The rule is not checked unless you select it and enter a value. **Tip:** Unchecked rules appear in the wire bond report as “Not Set.”
Max Angle	Assigns the maximum angle, in degrees, at which you can place wire bonds in the die part. The rule is not checked unless you select it and enter a value. **Tip:** Unchecked rules appear in the wire bond report as “Not Set.”
Import button	Imports wire bond rules saved in the Wire Bond Wizard dialog box from the Wire Bond Wizard Setup file, and assigns the values as rules for the currently open die part when you click **OK** in the Wire Bond Rules dialog box.

Related Topics

Checking Wire Bond Rules

Wire Bond Wizard Dialog Box

Use the Wire Bond Wizard dialog box to do the following:

- Define the SBP rings.
- Set rules.

- Assign the component bond pads to rings.
- Define the wire bond fanout geometry.
- Create a wire bond fanout.

The Die Wizard has 4 tabs:

- Guide
- Fanout Prefs
- Strategy
- CBPs

Accessing

- **BGA Toolbar** button > **Wire Bond Wizard** button > select die part

Figure 45-383. Wire Bond Wizard Dialog Box

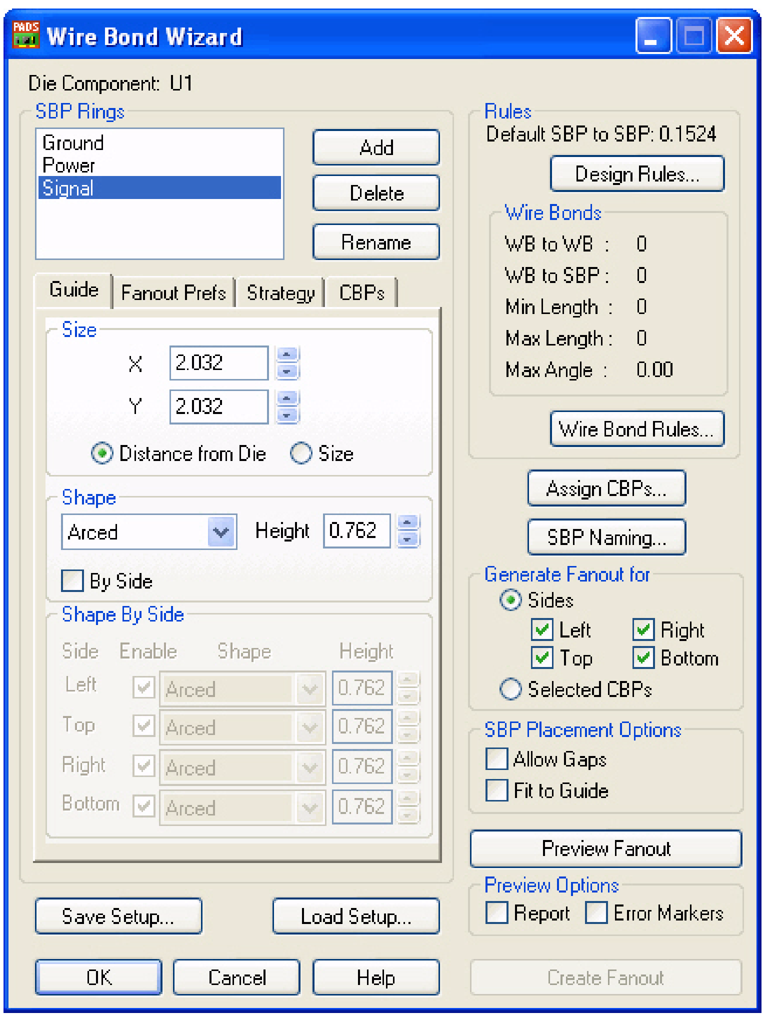


Table 45-351. Wire Bond Wizard Dialog Box contents

Name	Description
Die Component	Displays the selected Die Component.
SBP Rings list	Lists all currently defined SBP rings. Select an SBP ring or rings to view, edit, or define. You can also add new rings, delete rings, and rename rings.
Add button	Opens the Add SBP Ring dialog box, where you can insert a new SBP ring for the die.
Delete button	Deletes the selected SBP ring. This also unassigns all component bond pads assigned to this ring and deletes SBPs and wire bonds, but only if the CBPs are assigned to a single SBP ring. If a CBP is assigned to two or more SBP rings, and one of the rings is deleted, the CBP remains assigned to the remaining ring, and the corresponding SBPs and WBs are kept.
Rename button	Opens the Rename SBP Ring dialog box, where you can rename the currently selected SBP ring.
tabs	• Guide • Fanout Prefs • Strategy • CBPs
Save Setup button	Saves Wire Bond Wizard dialog box settings from the current design to use in another design. The settings are stored in a text file with a .wbw extension. Use Load Setup to open .wbw files and apply the settings to other designs.
Load Setup button	Loads the settings from the text setup file and uses them in the current design. The settings are stored in a text file with a .wbw extension. Use Save Setup to save .wbw files to use them in another design.
SBP to SBP Clearance Rules	Displays the default value for the SBP–to–SBP clearance rule, represented by SMD-to-SMD default clearance rules set on the Clearance Rules dialog box.
Design Rules button	Opens the Rules dialog box, where you can change the value for the SMD-to-SMD default clearance rule settings affecting the SBP-to-SBP clearance. From the Rules dialog box, click Default, then Clearance.
Wire Bonds area	Displays the settings for all wire bond rules for the currently selected die component.

Table 45-351. Wire Bond Wizard Dialog Box contents (cont.)

Name	Description
Wire Bond Rules button	Opens the Wire Bond Rules dialog box, where you can specify the wire-bond-to-wire-bond clearance, wire-bond-to-substrate-bond-pad-clearance, wire bond minimum and maximum lengths, and wire bond maximum angle.
Assign CBPs button	Opens the Assign CBPs to Rings dialog box, where you can assign component bond pads to one or more rings.
SBP Naming button	Opens the SBP Properties dialog box, where you can specify substrate bond pad numbers and substrate bond pad function names for newly created substrate bond pads.
Generate Fanout for area	Select component bond pads for fanout processing. • **Sides**—Select the SBP ring side or sides to process. Wire Bond Wizard processes all component bond pads assigned to the selected side or sides of the SBP rings. • **Selected CBPs**—Select component bond pads in the layout window.
Allow Gaps	Allows gaps when placing wire bonds in the wire bond fanout. Otherwise, Wire Bond Wizard evenly distributes all wire bond fanouts into a compact group.
Fit to Guide	Fits the substrate bond pads along the SBP guide using the preferred spacing values for the SBP rings. Otherwise, Wire Bond Wizard uses a smaller value, up to the minimum spacing allowed by the rules.
Preview Fanout button	Displays a wire bond fanout preview using the current settings. You can repeatedly view and modify the fanout until it satisfies the wire bond rules. If you change any of the settings that affect the fanout pattern, the fanout changes to Preview of CBP Assignment mode.
Report	Generates a report containing created object and wire bond rule violations.
Error Markers	Generates and displays error markers in the fanout preview.
Create Fanout button	Creates the new fanout and saves its settings in the design. This also generates a report that lists created objects and wire bond rule violations.

Figure 45-384. Wire Bond Wizard, Guide tab

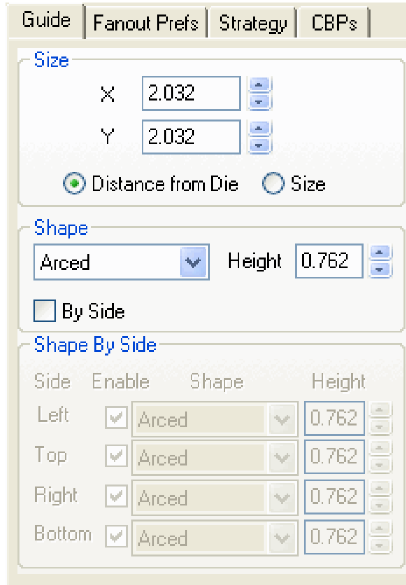


Table 45-352. Guide tab contents

Name	Description
Size area	Defines the size of the SBP guide: • **Distance from Die**—Click to enter X and Y values for the distance of the substrate bond pad guide box from the die outline. • **Size**—Click to set an absolute size for the guide box.
Shape list	Defines the shape of the SBP guide. Available selections are: rectangle, rounded rectangle, arced, and tent. If you select arced or tent, you can also select a height.
By Side	Creates a substrate bond pad guide with mixed shapes.
Shape by Side area	Defines a shape for each side of the SBP guide. To define a shape for a side, click Enable. In the Shape column, select a shape type for that side. When you select the arced or tent shape, select a height in the Height column. You can create the SBP guide with one or more sides missing.

Figure 45-385. Wire Bond Wizard, Fanout Prefs tab

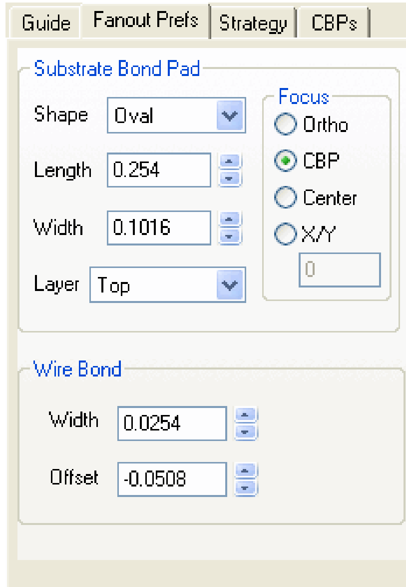


Table 45-353. Fanout Prefs contents

Name	Description
Shape	Select a shape for the substrate bond pads to create in the fanout of the currently selected SBP rings. Choices are Rectangle and Oval.
Length	Select a length for the substrate bond pads to create in the fanout of the currently selected SBP rings.
Width	Select a width for the substrate bond pads to create in the fanout of the currently selected SBP rings.
Layer	Select an electrical layer for the substrate bond pads of the currently selected SBP rings.
Focus area	Select the area on which you want the focus: Ortho, CBP, Center. or X/Y location.
Width	Select a width for the wire bond for the fanout of the currently selected SBP rings.
Offset	Select an offset for the wire bond for the fanout of the currently selected SBP rings.

Figure 45-386. Wire Bond Wizard, Strategy tab

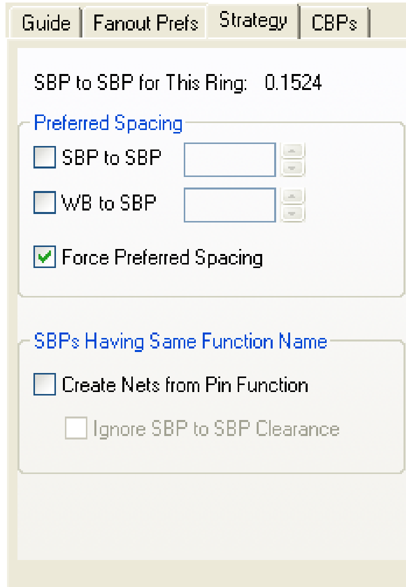


Table 45-354. Strategy tab contents

Name	Description
SBP to SBP for This Ring	Displays the substrate-bond-pad-to-substrate-bond-pad clearance value for the layer to create the substrate bond pads for the selected rings. The clearance value for this ring may be different from the default SBP–to–SBP clearance if specific design rules are used for the layer specified on the Fanout Prefs tab. The layer is specific to the substrate bond pads of this ring. The SBP–to–SBP clearance rule is represented by SMD-to-SMD clearance rules set on the Clearance Rules dialog box.
SBP to SBP Spacing	Defines the preferred spacing from substrate bond pad to substrate bond pad for the selected ring. The value must be equal to or larger than the current SBP–to–SBP rule, represented by SMD-to-SMD clearance rules set on the Clearance Rules dialog box.
WB to SBP Spacing	Defines the preferred spacing from wire bonds to substrate bond pads for the selected rings. The value must be equal to or larger than the WB–to–SBP rule set on the Wire Bond Rules dialog box.

Table 45-354. Strategy tab contents (cont.)

Name	Description
Force Preferred Spacing	Prevents automatic switching to the smallest allowed spacing when fitting the substrate bond pads of the selected rings along the substrate bond pad guide. If Fit to Guide is selected, you can affect the spacing of the wire bonds of the selected rings by clearing the Force Preferred Spacing option.
Create Nets from Pin Function	Incorporates all new substrate bond pads with the same function name into the net of the same function name. If this net does not already exist, Wire Bond Wizard creates it.
Ignore SBP to SPB Clearance	Ignores the SBP to SBP Clearance rules.

Figure 45-387. Wire Bond Wizard, CBPs tab

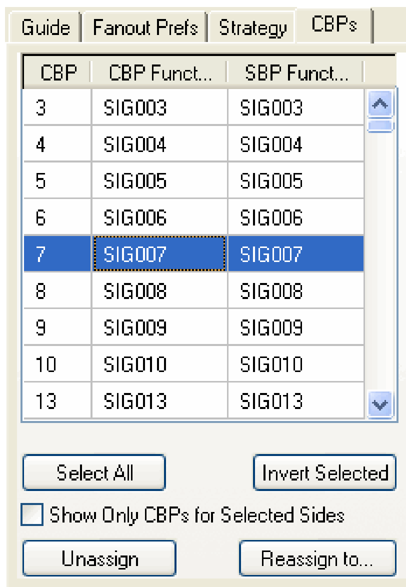


Table 45-355. CBPs tab contents

Name	Description
CBP column	Lists the numbers of all component bond pads assigned to the selected ring.
CBP Function column	Lists the functions of all component bond pads assigned to the selected ring.
SBP Function	Lists the functions of all substrate bond pads assigned to the selected ring.

Table 45-355. CBPs tab contents (cont.)

Name	Description
Select All button	Selects all component bond pads in the CBP list to assign to the currently selected rings.
Invert Selected button	Inverts all selections currently in the list. If one item is currently selected, this button will deselect that one item and select all other items instead.
Show Only CBPs for Selected Sides	Displays only component bond pads specified in the Generate Fanout for area of the Wire Bond Wizard dialog box. If you do not select this option, the list displays all component bond pads, no matter what the current settings are in the Generate Fanout for area.
Unassign button	Removes the assignments of the selected component bond pads from their rings and returns those component bond pads to the list of unassigned component bond pads, if the CBPs belong to a single SBP ring. Otherwise the CBPs still belong to the remaining rings to which they were assigned.
Reassign To	Opens the Reassign CBPs to Rings dialog box, where you can select a ring or rings in the list to reassign component bond pads.

Related Topics

To Create a Wire Bond Fanout

BGA Operations

Working Folder Browser Dialog Box

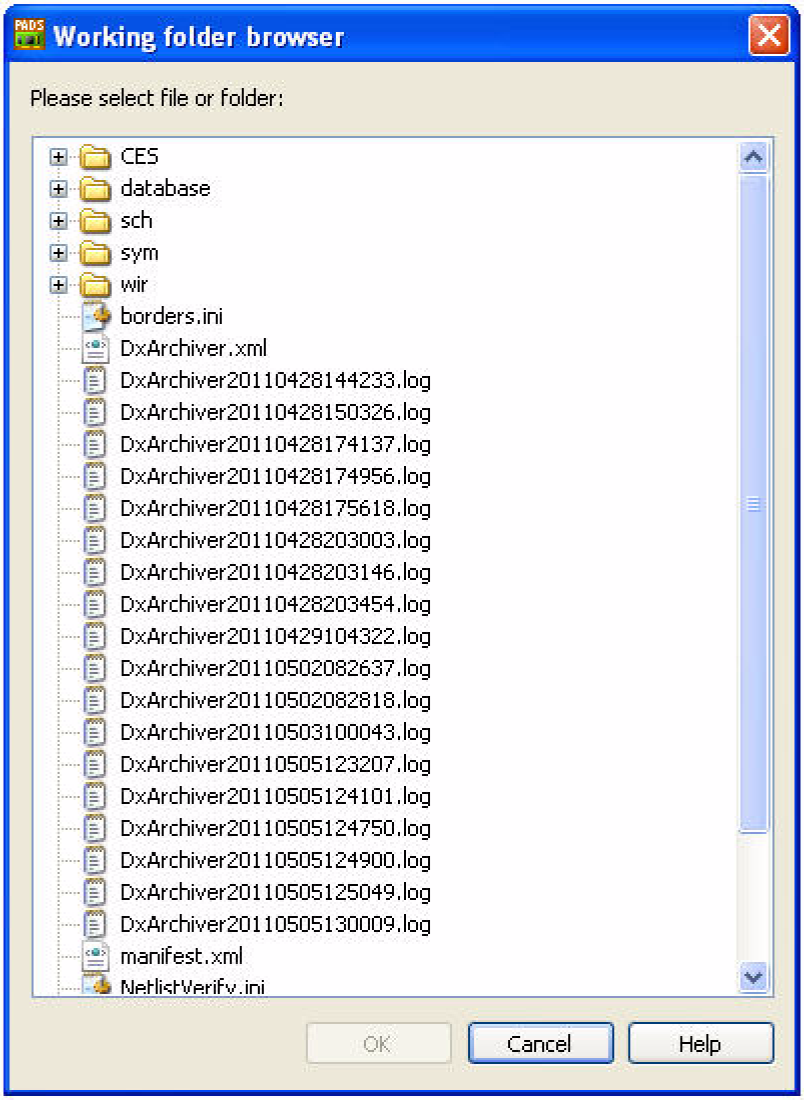
Use the Working Folder Browser dialog box to add project files to the Schematic project file, Layout project file, and Additional files and folders lists in the Add Archive to Vault and Templates dialog boxes.

Tip: Some files are automatically saved with a selected .prj or .pcb file; these files are not displayed in the browser file tree. For example, when you select a .prj file, its associated database folder is not displayed for you to select, because it is always saved with the .prj file.

Accessing

- In the **Add Archive to Vault** dialog box, click the Browse button ... .

Figure 45-388. Working Folder Browser Dialog Box



Chapter 46
PADS Layout Automation Server

Welcome to the Automation Server Help

This help file describes the Automation Server features contained in PADS Layout, and includes:

- OLE Background, a short description of OLE and Automation.
- The Automation Server Object Hierarchy
- PADS Layout Automation Samples showing how to get started with writing Automation clients, in Microsoft Visual Basic, Microsoft Visual C++, and Microsoft Excel

  Disclaimer: The code samples in PADS Layout Automation Server Help are freeware. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.
- PADS Layout Automation Server Reference, listing and detailing each Automation object, property, method, and event in the Automation Server.

OLE Background

"The fundamental question OLE addresses: How can a system be designed such that binary components from different vendors—written in different parts of the world at different times and using different programming languages—are guaranteed to interoperate?"

Dr. Dobbs's Journal, January 1995.

Technically, OLE stands for Object Linking and Embedding, but that is now an insufficient title. Today, OLE is a continuously growing set of features (including Object Linking and Embedding, Automation, Compound Documents, Visual Editing, ActiveX, DCOM, and so on) based on Microsoft's version of a framework allowing heterogeneous applications to communicate with each other.

Automation is a feature of OLE that defines a protocol for applications to share their data and functionality with any other application using that same protocol. Automation involves a *server* and a *client*. A server is the entity that makes available some data and functionality. A client, also called the controller, is the entity that uses the server's data and functionality. Different people at different organizations can author servers and clients at different times, using different development tools and languages. The only required commonality is that they use the same Automation communication protocol. PADS Layout is an Automation server because it makes some data (the database) and some functionality (such as opening design files and selecting objects) available to other applications. A client, such as Microsoft Excel, can access PADS Layout data and functionality using the Automation protocol.

The Automation implementation allows third party companies and users to:

- Integrate their products.
- Expand the PADS Layout set of functionality.
- Customize existing features.
- Automate tasks using standard scripting languages.

In addition, this can be done independently from release cycles.

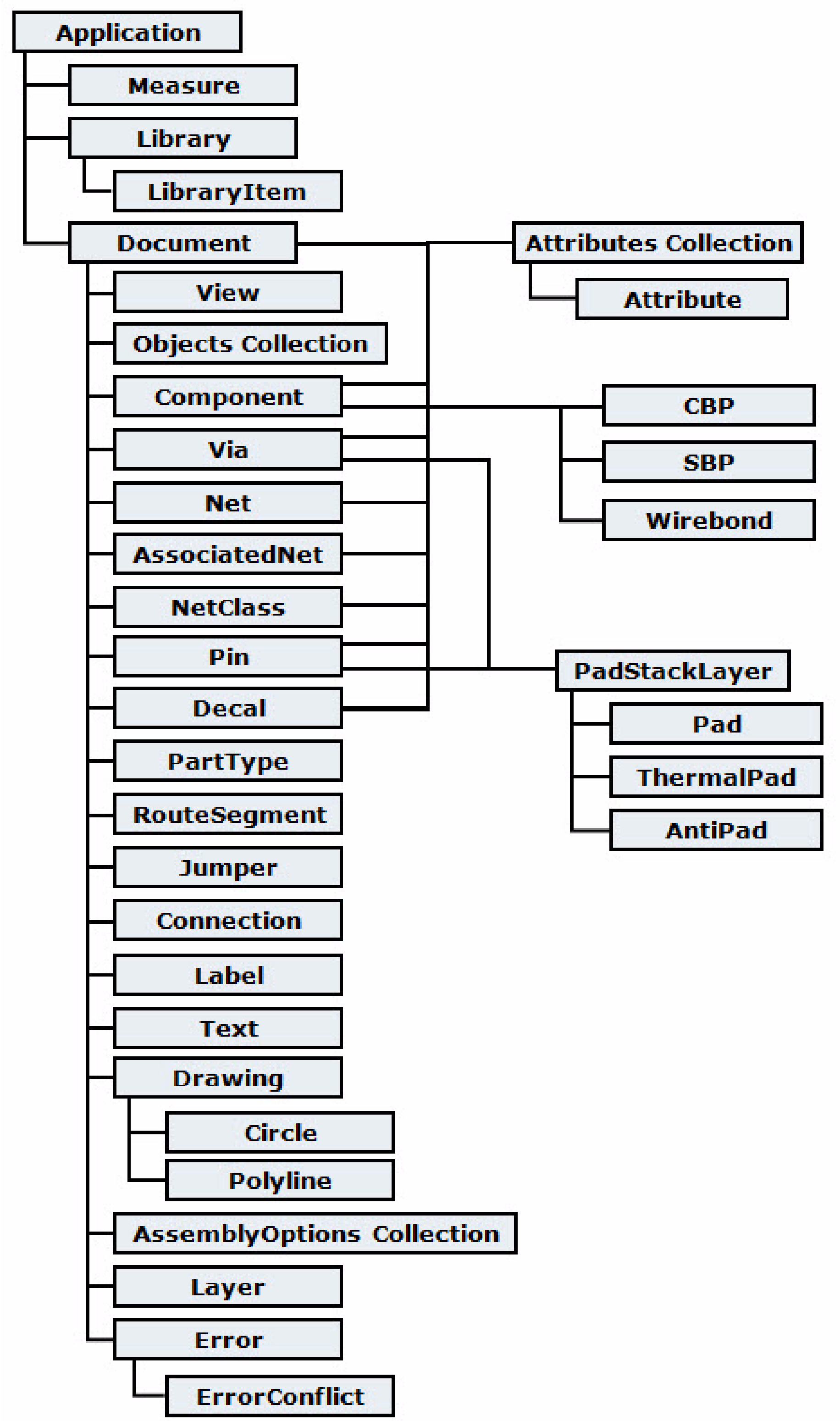
Sample 5 demonstrates exactly these benefits, as it integrates PADS Layout and Excel and expands the Report command with clean, customizable Excel reports; giving the end-user a tightly integrated and customizable Excel report solution.

The Automation Server Object Hierarchy

Tip: The CBP, SBP, and Wirebond objects are for the Advanced Packaging Toolkit only and are only applicable if the object is a die part (Component.IsDiePart)**.**

Figure 46-1. The Automation Server Object Hierarchy

(Click on an item to view its properties, methods and events.)



PADS Layout Automation Samples

Samples Summary

To make it easy for you to develop Automation clients designed for PADS Layout, or to integrate existing applications, several samples of this technology that you can use and modify are available. These samples are in the C:\PADS Projects\Samples\Scripts\Layout\samples folder. These samples include:

- Sample 1 details the basics of creating an Automation client for the PADS Layout server. This sample demonstrates how to connect to an existing or new instance, how to disconnect from it, and how to access simple data.
- Sample 2 extends Sample 1 by adding events, which notify clients when the state of PADS Layout changes.
- Sample 3 extends Sample 2 by adding access to the PADS Layout database.
- Sample 4 extends Sample 3 by adding a full-duplex cross-probing feature between the client and PADS Layout. Sample 4 also adds detailed database information retrieval.
- Sample 5 is a full-featured custom plug-in that generates custom “intelligent reports” of the current design in various formats, such as Microsoft Excel spreadsheets, Microsoft Word tables, and HTML pages. Intelligent reports are dynamically linked to the original PADS Layout data, allowing users to modify component data (such as set position, mounting layer, and glued status) from the generated report.

  This sample demonstrates that the Automation Server allows users and third party vendors to develop their own plug-ins, which may dynamically link data with the open design. You can easily extend this sample to include dynamic data links with other applications, such as Microsoft Access and Visual FoxPro.
- Sample 6 is a Visual C++ application that displays all PADS Layout geometric objects, accessible through the Automation interface.

Tips:

- You need to install the appropriate development tool on your system to run these samples. In other words, to run the Visual Basic version of any of these samples, you need Visual Basic on your system, or to run the Visual C++ version of any of these samples, you need Visual C++ on your system.
- If you encounter difficulty with one of the samples or with your own client application, see Troubleshooting Samples.
- If you want to create wrapper classes for PADS Layout objects, you can use your Visual C++ development tool to create them.

Disclaimer: The code samples in PADS Layout Automation Server Help are freeware. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.

Troubleshooting Samples

If a sample doesn't work as described, check the following:

- Make sure that only one instance of PADS Layout is running. By definition, a server is unique. Several instances of the same server may create confusion for clients.
- Exiting and restarting PADS Layout may help in some cases because these programs self-register as Automation servers upon start up. To make sure that you don't have another instance of PADS Layout running in the background, use the Task Manager.
- Make sure you disconnect all clients from PADS Layout before exiting the client application.
- If the above doesn't help, restart your system.

Creating Wrapper Classes for PADS Layout Objects

You can use your Visual C++ development tool to automatically create wrapper classes for PADS Layout objects from the PADS Layout type library. The PADS Layout type library is included into PADS Layout resources, so you can import the type library from the PADS Layout application file, powerpcb.exe.

The instructions in this topic apply to Microsoft Visual C++, version 6.0. If you use another Visual C++ development tool, refer to its documentation for information about importing type libraries.

To create wrapper classes for PADS Layout objects:

1. In Microsoft Visual C++, open a project.
2. On the View menu, click **ClassWizard**. The MFC ClassWizard dialog box opens.
3. Click **Add Class** and then, from the list that appears, select **From a type library**. The Import from Type Library dialog box opens.
4. Set the filter to All Files, browse to C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\powerpcb.exe, and then click **Open**. The Confirm Classes dialog box opens.
5. Select the classes you want to use and click **OK**.

Result: Microsoft Visual C++ creates the .h and .cpp files containing the wrapper classes.

Sample 1 - Creating an Automation Client

This topic discusses the goal and specification for Sample 1.

Goal

To introduce the basics of writing an Automation client application using the Sax Basic Engine, Microsoft Visual C++ 5.0 (with MFC), Microsoft Visual Basic 5.0, and Microsoft Excel 97. One sample is provided for each of these applications.

Disclaimer: The code samples in PADS Layout Automation Server Help are freeware. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.

Each version of this sample demonstrates, in the respective programming language, how to:

- Prepare the communication between your client and the Automation Server.
- Prepare a pointer to the top level Automation object.
- Connect and disconnect from PADS Layout, regardless of whether it is running or not.
- Access simple data from PADS Layout.

Tip: The source code for this sample is in the C:\PADS Projects\Samples\Scripts\samples\sample1 folder. The code is well commented, and you may use it when developing your own client application.

Specification

Sample 1 is a simple dialog box, which can connect and disconnect at run time from the server. It can connect to an existing running instance or a new instance of PADS Layout. Once connected, the client retrieves the name of the open design file.

An example of the Visual Basic version of Sample 1 appears below. The Visual C++ version of this sample looks almost identical. The Basic scripting version does not have the Connect and Disconnect buttons because the Sax Basic Engine is already connected to PADS Layout. The Excel 97 version is different because Excel is primarily an application, not a development environment. The Excel version of this sample outputs information in an Excel worksheet.

Requirement: To run Sample 1 clients, start PADS Layout and open a design file. If you don't start PADS Layout first, Sample 1 automatically starts the server, but it will not be visible and will not load a design file.

Make sure you copy the OLE samples from the distribution CD to your hard drive before running any of the samples.

Warning: Never exit a client application without first disconnecting from the server to which it is connected.

Sample 2 - Adding Notifications

This topic discusses the goal and specification for Sample 2.

Goal

To add client notifications, also called client callbacks, to Sample 1 so that Sample 1 can be notified when something changes in PADS Layout. Sample 2 is implemented in PADS Layout Sax Basic Engine, Microsoft Visual C++ 5.0 (and MFC), Microsoft Visual Basic 5.0, and Microsoft Excel 97.

Disclaimer: The code samples in PADS Layout Automation Server Help are freeware. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.

Tip: The source code for this sample is in the C:\PADS Projects\Samples\Scripts\Layout\samples\sample2 folder. The code is well commented, and you may use it when developing your own client application.

Specification

Sample 2 is based on Sample 1 source code and functionality. This sample adds the ability to retrieve two more data types from the PADS Layout server: the total number of components in the current design and the number of selected components. Moreover, Sample 2 automatically refreshes these values (name of the design file, component count, and selected component count) to match values in the PADS Layout server at all times by responding to notifications about design file and selection change.

To test Sample 2, run PADS Layout, open a design file, and connect the Sample 2 client to PADS Layout. If you load another file or change the selection, Sample 2 refreshes its values automatically.

Requirement: To run Sample 2 clients, start PADS Layout and open a design file. If you don't start PADS Layout first, Sample 2 automatically starts the server, but PADS Layout will not be visible and will not load a design file.

Make sure you copy the OLE samples from the distribution CD to your hard drive before running any of the samples.

Warning: Never exit a client application without first disconnecting from the server to which it is connected.

Sample 3 - Adding Database Access

This topic discusses the goal and specification for Sample 3.

Goal

To demonstrate access to the PADS Layout database.

Disclaimer: The code samples in PADS Layout Automation Server Help are freeware. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.

Tip: The source code for this sample is in the C:\PADS Projects\Samples\Scripts\Layout\samples\sample3 folder. The code is well commented, and you may use it when developing your own client application.

Specification

Sample 3 is based on Sample 2 source code. Sample 3 adds a list box that contains the list of all components in the current design. This list should automatically update when a new file is opened. This is the Basic version of Sample 3. To test Sample 3, run PADS Layout, open a design file, and connect the Sample 3 client to PADS Layout. If you open another file or change the selection, Sample 3 refreshes its values automatically.

Requirement: To run Sample 3 clients, start PADS Layout and open a design file. If you don't start PADS Layout first, Sample 3 automatically starts the server, but PADS Layout will not be visible and will not load a design file.

Make sure you copy the OLE samples from the distribution CD to your hard drive before running any of the samples.

Warning: Never exit a client application without first disconnecting from the server to which it is connected.

Sample 4 - Adding Cross-Probing

This topic discusses the goal and specification for Sample 4.

Goal

Sample 4 extends the access to the PADS Layout database components and retrieves detailed information about each component. It also adds full-duplex selection cross-probing and demonstrates the server locking mechanism to improve Automation performance.

Disclaimer: The code samples in PADS Layout Automation Server Help are freeware. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.

Tip: The source code for this sample is in the C:\PADS Projects\Samples\Scripts\Layout\samples\sample4 folder. The code is well commented, and you may use it when developing your own client application.

Specification

Sample 4 is based on Sample 3 source code. Sample 4 must allow selection and cross-probing between the client and the server. In other words, the list of selected objects must match the list of selected items in the Sample 4 list box at all times and vice-versa. Selection and cross-probing must support multiple selection. If a user double-clicks an item in the list box, a dialog box must show extended information about that selected object. The performance of the access of PADS Layout OLE methods will be improved using the server locking mechanism.

To test Sample 4, run PADS Layout, open a design file, and connect the Sample 4 client to PADS Layout. If you load another file or change the selection, Sample 4 refreshes its values automatically; matching the selected components at all times. Similarly, a user selection change in the Sample 4 client changes the selection list.

Requirement: To run Sample 3 clients, start PADS Layout and open a design file. If you don't start PADS Layout first, Sample 4 automatically starts the server, but PADS Layout will not be visible and will not load a design file.

Make sure you copy the OLE samples from the distribution CD to your hard drive before running any of the samples.

Warning: Never exit a client application without first disconnecting from the server to which it is connected.

Sample 5 - A PADS Layout Custom Reports Command

This topic discusses the goal, specification, additional features, and implementation for Sample 5.

Goal

This sample provides custom reports with output formats such as text files, Microsoft Excel spreadsheets, Microsoft Word documents, and HTML pages.

Disclaimer: The code samples in PADS Layout Automation Server Help are freeware. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.

Tip: The source code for this sample is in the C:\PADS Projects\Samples\Scripts\Layout\samples\sample5 folder. The code is well commented, and you may use it when developing your own client application. This sample includes Basic files, main source code, an Excel file, report generation macros, and a PADS Layout macro file that "plugs" the sample into PADS Layout.

Specification

This sample generates customizable part list reports of the open design and/or any Assembly Options in that file. You can choose the format of the reports: text files, CSV files, Microsoft Excel spreadsheets, Microsoft Word documents (as tables), or HTML pages (for any Web browser). Each format requires that you install the appropriate application on your system. In other words, to generate Excel reports, you must install Excel on your system.

The data that the report outputs is configurable from the dialog box in the sample. You can easily select the part properties to output one by one, configure the order in which to output the properties, and group the report by any part property. Component properties available include reference designator, part type, decal, part type logic, part type description, number of attributes, SMD flag, number of pins, location, placed flag, orientation, mounting layer, and glued flag.

You can dynamically link the Excel reports with PADS Layout; you can link each piece of data in the Excel report with its equivalent in PADS Layout. This allows selection and cross-probing between the Excel report and PADS Layout, and also allows changing the following part properties from the Excel report:

- Part decal. The list of compatible decals for that part appears in a list within the Excel cell.
- Location of the part.
- Orientation of the part.
- Mounting layer of the part. The list of valid layers appears in a list within the Excel cell.
- Glued flag.
- Installed flag, if the report is from an Assembly Option in the design.

Additional Features

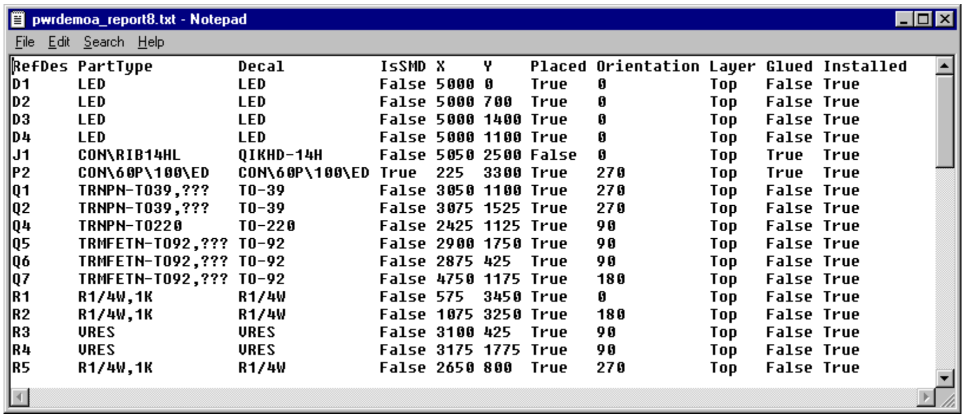
This sample includes many other useful features:

- Real-time refreshing of the dialog box when PADS Layout opens or saves the design file. This ensures that the dialog box is always in sync with the program.
- The ability to open a design file from the dialog box. This avoids you having to switch back and forth between PADS Layout and the dialog box when generating reports for many PCBs.
- The ability to save and load the report configurations to and from the system registry.
- The ability to reset default settings with a single button, to return to a “clean” configuration.
- A history of opened designs saved in the registry, up to ten, allowing quick report generation of a previously opened design file.

- Automatic suggestions for report file names.
- The ability to report all or only selected parts. This helps when generating a report on only one part in the PCB design.

The following examples are screen captures of custom reports, generated by this sample, in Notepad, Microsoft Excel (dynamically linked), Microsoft Word, and Microsoft Internet Explorer 4.0:

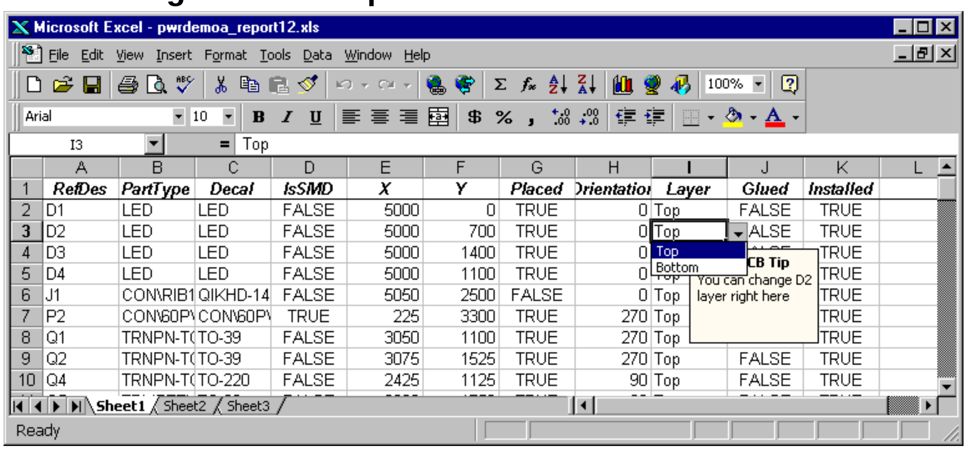
Figure 46-2. Report in Notepad Format



pwrdemoa_report8.txt - Notepad

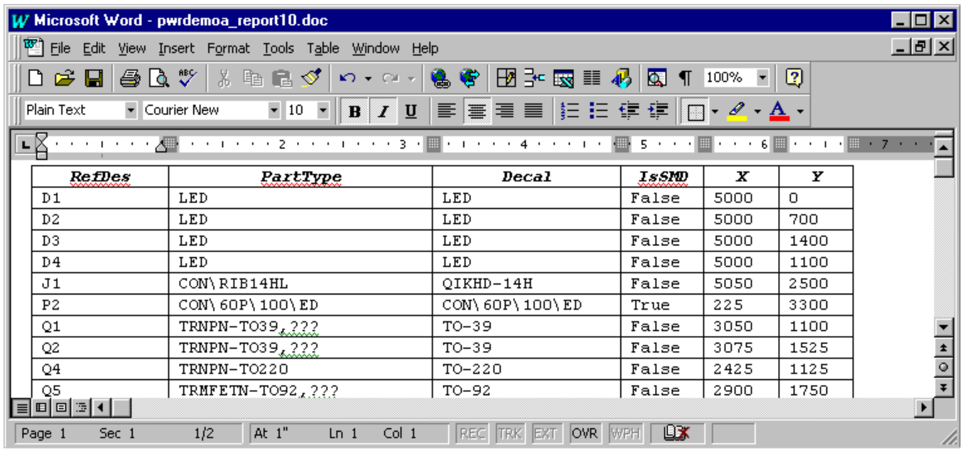
RefDes	PartType	Decal	IsSMD	X	Y	Placed	Orientation	Layer	Glued	Installed
D1	LED	LED	False	5000	0	True	0	Top	False	True
D2	LED	LED	False	5000	700	True	0	Top	False	True
D3	LED	LED	False	5000	1400	True	0	Top	False	True
D4	LED	LED	False	5000	1100	True	0	Top	False	True
J1	CON\RIB14HL	QIKHD-14H	False	5050	2500	False	0	Top	True	True
P2	CON\60P\100\ED	CON\60P\100\ED	True	225	3300	True	270	Top	True	True
Q1	TRNPN-TO39,???	TO-39	False	3050	1100	True	270	Top	False	True
Q2	TRNPN-TO39,???	TO-39	False	3075	1525	True	270	Top	False	True
Q4	TRNPN-TO220	TO-220	False	2425	1125	True	90	Top	False	True
Q5	TRMFETN-TO92,???	TO-92	False	2900	1750	True	90	Top	False	True
Q6	TRMFETN-TO92,???	TO-92	False	2875	425	True	90	Top	False	True
Q7	TRMFETN-TO92,???	TO-92	False	4750	1175	True	180	Top	False	True
R1	R1/4W,1K	R1/4W	False	575	3450	True	0	Top	False	True
R2	R1/4W,1K	R1/4W	False	1075	3250	True	180	Top	False	True
R3	VRES	VRES	False	3100	425	True	90	Top	False	True
R4	VRES	VRES	False	3175	1775	True	90	Top	False	True
R5	R1/4W,1K	R1/4W	False	2650	800	True	270	Top	False	True

Figure 46-3. Report in Microsoft Excel Format



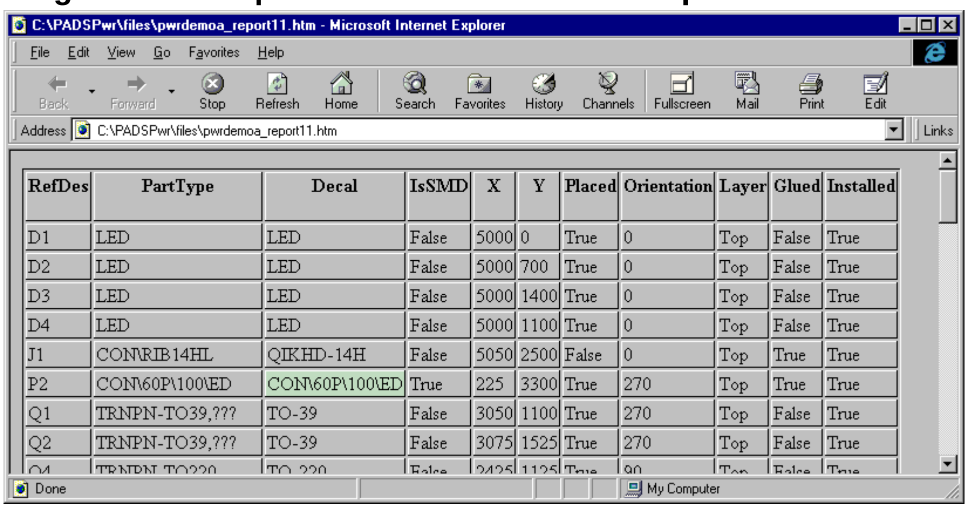
	RefDes	PartType	Decal	IsSMD	X	Y	Placed	Orientation	Layer	Glued	Installed
2	D1	LED	LED	FALSE	5000	0	TRUE	0	Top	FALSE	TRUE
3	D2	LED	LED	FALSE	5000	700	TRUE	0	Top	FALSE	TRUE
4	D3	LED	LED	FALSE	5000	1400	TRUE	0			TRUE
5	D4	LED	LED	FALSE	5000	1100	TRUE	0			TRUE
6	J1	CON\RIB1	QIKHD-14	FALSE	5050	2500	FALSE	0	Top		TRUE
7	P2	CON\60P\	CON\60P\	TRUE	225	3300	TRUE	270	Top		TRUE
8	Q1	TRNPN-T(	TO-39	FALSE	3050	1100	TRUE	270	Top		TRUE
9	Q2	TRNPN-T(	TO-39	FALSE	3075	1525	TRUE	270	Top	FALSE	TRUE
10	Q4	TRNPN-T(	TO-220	FALSE	2425	1125	TRUE	90	Top	FALSE	TRUE

Figure 46-4. Report in Microsoft Word Format



RefDes	PartType	Decal	IsSMD	X	Y
D1	LED	LED	False	5000	0
D2	LED	LED	False	5000	700
D3	LED	LED	False	5000	1400
D4	LED	LED	False	5000	1100
J1	CON\RIB14HL	QIKHD-14H	False	5050	2500
P2	CON\60P\100\ED	CON\60P\100\ED	True	225	3300
Q1	TRNPN-TO39,???	TO-39	False	3050	1100
Q2	TRNPN-TO39,???	TO-39	False	3075	1525
Q4	TRNPN-TO220	TO-220	False	2425	1125
Q5	TRMFETN-TO92,???	TO-92	False	2900	1750

Figure 46-5. Report in Microsoft Internet Explorer 4.0 Format



RefDes	PartType	Decal	IsSMD	X	Y	Placed	Orientation	Layer	Glued	Installed
D1	LED	LED	False	5000	0	True	0	Top	False	True
D2	LED	LED	False	5000	700	True	0	Top	False	True
D3	LED	LED	False	5000	1400	True	0	Top	False	True
D4	LED	LED	False	5000	1100	True	0	Top	False	True
J1	CON\RIB14HL	QIKHD-14H	False	5050	2500	False	0	Top	True	True
P2	CON\60P\100\ED	CON\60P\100\ED	True	225	3300	True	270	Top	True	True
Q1	TRNPN-TO39,???	TO-39	False	3050	1100	True	270	Top	False	True
Q2	TRNPN-TO39,???	TO-39	False	3075	1525	True	270	Top	False	True
Q4	TRNPN-TO220	TO-220	False	2425	1125	True	90	Top	False	True

Implementation

This sample was developed with Microsoft Visual Basic 5.0. Dynamic Excel reports include some Visual Basic for Application (VBA) code in the Excel report file itself. The code is modularized to make it easy to implement other report formats, such as Microsoft Access, Visual FoxPro, and so on.

Sample 6 - Accessing Database Objects through the Automation Interface

This topic discusses the goal and specification for Sample 6.

Goal

To demonstrate access to shapes in the PADS Layout database through the Automation interface.

Disclaimer: The code samples in PADS Layout Automation Server Help are freeware. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.

Tip: The source code for this sample is in the C:\PADS Projects\Samples\Scripts\Layout\samples\sample6 folder. The code is well commented, and you may use it when developing your own client application. This sample includes source code for Visual C++ v6.0.

Specification

Sample 6 is a Visual C++ application that displays all geometric objects, accessible through the Automation interface.

To test Sample 6, open a design in PADS Layout, then build and run a sample executable. The sample retrieves data from the design and draws it in its window. To refresh the view, on the View menu, click **Redraw**.

PADS Layout Automation Server Reference

The Automation Server Object Hierarchy strictly follows Microsoft standards. The Application Object, the root-level object, identifies the application and provides a way for Automation clients to bind to and navigate the application's exposed objects, methods, and properties. The Document Object handles all document-centered operations. The View Object handles all view-centered operations.

All PADS Layout database objects, including the Drawing, Label, NetClass, Text, Component, PartType, Net, Pin, Via, Jumper, Connection, and RouteSegment objects, are accessed either directly through the Document object or through the Objects collection object. The Objects collection object provides a convenient way of working with a set of data objects rather than on a per object basis.

All Automation objects implement their interface based on the Microsoft IDispatch interface.

Automation Objects

- The AntiPad Object
- The Application Object
- The AssemblyOptions Collection Object
- The Attribute Object
- The Attributes Collection Object
- The CBP Object
- The Circle Object
- The Component Object
- The Connection Object
- The Decal Object
- The Document Object
- The Drawing Object
- The Error Object
- The ErrorConflict Object
- The Jumper Object
- The Label Object
- The Layer Object
- The Library Object
- The LibraryItem Object
- The Measure Object
- The Net Object
- The NetClass Object
- The Objects Collection Object
- The Pad Object
- The PadStackLayer Object
- The PartType Object
- The Pin Object
- The Polyline Object
- The RouteSegment Object
- The SBP Object
- The Text Object
- The ThermalPad Object
- The Via Object
- The View Object
- The Wirebond Object

The AntiPad Object

This object represents an antipad in the padstack definition.

AntiPad Properties

AntiPad.Application

AntiPad.CornerRadius

AntiPad.CornerType

AntiPad.Length

AntiPad.Name

AntiPad.ObjectType

AntiPad.Offset

AntiPad.Orientation

AntiPad.PadStackLayer

AntiPad.Parent

AntiPad.Shape

AntiPad.Size

The Application Object

The Application object is the root-level object in the Automation Server object hierarchy, and represents the entire application. This object is usually the first object an Automation client connects to, before accessing an object, property, or method.

Application Properties

Application.ActiveDocument

Application.Application

Application.DefaultFilePath

Application.FullName

Application.Libraries

Application.Name

Application.ObjectType

Application.Parent

Application.Preference

Application.ProgressBar

Application.StatusBarText

Application.Version

Application.Visible

Application Methods

Application.CreateLibrary

Application.ExportLibraryItems

Application.GetConfigParamInt

Application.GetConfigParamString

Application.GetLibraryItems

Application.LockServer

Application.Measure

Application.OpenDocument Method

Application.OpenDocumentNoLock Method

Application.OpenTempDocument Method

Application.Quit Method

Application.RunMacro

Application.UnlockServer

Application Events

Application.OpenDocument Event

Application.ProgressChange

Application.Quit Event

The AssemblyOptions Collection Object

The AssemblyOptions collection object is the collection of all the assembly options in the open design. This object is usually retrieved using the Document.AssemblyOptions property.

AssemblyOptions Properties

AssemblyOptions.Application

AssemblyOptions.Count

AssemblyOptions.Item

AssemblyOptions.ItemType

AssemblyOptions.Next

AssemblyOptions.ObjectType

AssemblyOptions.Parent

AssemblyOptions.ParentObject

AssemblyOptions Methods

AssemblyOptions.Add

AssemblyOptions.Delete

AssemblyOptions.Merge

AssemblyOptions.Remove

AssemblyOptions.Reset

AssemblyOptions.Select

AssemblyOptions.Sort

The AssociatedNet Object

The AssociatedNet object represents an associated net existing in the PCB design currently open.

AssociatedNet Properties

AssociatedNet.Name

AssociatedNet.Nets

AssociatedNet.ObjectType

AssociatedNet.Parent

AssociatedNet.Selected

The Attribute Object

The Attribute object represents an attribute of a physical object, such as a Component or Net object. This object is usually retrieved from the The Attributes Collection Object.

Attribute Properties

Attribute.Application

Attribute.Name

Attribute.ObjectType

Attribute.Parent

Attribute.Value

Attribute.Measure

The Attributes Collection Object

The Attributes collection object is a collection of physical object attributes, such as attributes of a Document (using the Document.Attributes property), a Component (using the Component.Attributes property), or a Pin (using the Pin.Attributes property).

Attributes Collection Properties

Attributes.Application

Attributes.Count

Attributes.Item

Attributes.ItemType

Attributes.Next

Attributes.ObjectType

Attributes.Parent

Attributes.Application

Attributes Methods

Attributes.Add

Attributes.Delete

Attributes.Merge

Attributes.Remove

Attributes.Reset

Attributes.Select

Attributes.Sort

The CBP Object

The CBP object represents the physical component bond pad of a die part. You can retrieve the CBP object, or collection of CBP objects, only from The Component Object, which represents a die part (Component.IsDiePart returns True), or from other die part constituents (such as SBP or Wirebond).

CBP objects are not selectable.

You cannot retrieve the collection of all CBP objects from a document using Document.GetObjects (ppcbObjectTypeCBP). This method returns an empty collection. Also, Document.GetObjects (ppcbObjectTypeAll) does not return CBP, SBP, or Wirebond objects. You can get only collections of CBPs, SBPs, or Wirebonds using the Component object methods, if the component is a die parT (Component.IsDiePart).

CBP Properties

CBP.Application

CBP.Edge

CBP.Function

CBP.Layer

CBP.Length

CBP.Name

CBP.ObjectType

CBP.Parent

CBP.PositionX

CBP.PositionY

CBP.SBPs

CBP.Shape

CBP.Width

CBP.Wirebonds

CBP Methods

CBP.Component

The Circle Object

The Circle object represents a physical circle in the open design.

Circle Properties

Circle.Application

Circle.CenterX

Circle.CenterY

Circle.Geometry

Circle.Layer

Circle.LineWidth

Circle.ObjectType

Circle.OutlineType

Circle.Parent

Circle.Radius

Circle.ShapeType

The Component Object

The Component object represents a physical component, in the currently open design.

Component Properties

Component.Selected

Component.Substituted

Component.WireBondRulesAngleMaximum

Component.WireBondRulesClearanceWireToPad

Component.WireBondRulesClearanceWireToWire

Component.WireBondRulesLengthMaximum

Component.WireBondRulesLengthMinimum

Component.Wirebonds

Component Methods

Component.AddLabel

Component.Move

Component.MoveCenterr

The Connection Object

The Connection object represents a physical connection, also called a pin pair, in the open design.

Connection Properties

Connection.Application

Connection.Length

Connection.Name

Connection.Net

Connection.ObjectType

Connection.Parent

Connection.Pins

Connection.RouteSegments

Connection.Selected

Connection.Vias

The Decal Object

The Decal object represents a decal, in the currently open design.

Decal Properties

Decal.Application

Decal.Attributes

Decal.Components

Decal.LibraryTimeStamp

Decal.Name

Decal.ObjectType

Decal.Parent

Decal.Selected

Decal.TimeStamp

The Document Object

The Document object represents a currently open PCB design file. This object is usually retrieved using the Application.ActiveDocument property.

Document Properties

Document.ActiveView

Document.Application

Document.AssemblyOptions

Document.AssociatedNets

Document.Attributes

Document.BoardOutlineSurface

Document.Componentss

Document.Connections

Document.Drawings

Document.ElectricalLayerCount

Document.Errors

Document.FullName

Document.GridX

Document.GridY

Document.Jumpers

Document.LayerCount

Document Methods

Document.GetColor

Document.GetObjects

Document.GetVisibility

Document.ImportECOFile

Document.ImportNetList

Document.IntegrityTest

Document.Save Method

Document.SaveAs

Document.SaveAsNoLock

Document.SaveNoLock

Document.SaveAsTemp

Document.SaveTemp

Document.SelectObjects

Document.SetColor

Document.SetVisibility

Document Events

Document.PositionsChange

Document.Save Event

Document.SecurityLimit Event

Document.SelectionChange Event

The Drawing Object

The Drawing object represents a physical drawing object or a copper in the open design.

Drawing Properties

Drawing.Application

Drawing.DrawingType

Drawing.Geometry

Drawing.Name

Drawing.Net

Drawing.ObjectType

Drawing.Parent

Drawing.PositionX

Drawing.PositionY

Drawing.Selected

Drawing.Texts

The Error Object

The Error object represents a design error.

Error Properties

Error.ActualValue

Error.Application

Error.Conflicts

Error.Description

Error.ErrorClass

Error.ErrorType

Error.HasActualValue

Error.HasRequiredValue

Error.IsClearanceError

Error.IsConnectivityError

Error.IsHighSpeedError

Error.IsIgnoredFlag

Error.IsInvisibleFlag

Error.IsLatiumError

Error.IsMiscError

Error.IsTestPointError

Error.LayerNumber

Error.Name

Error.ObjectType

Error.Parent

Error.PositionX

Error.PositionY

Error.RequiredValueMax

Error.RequiredValueMin

The ErrorConflict Object

The Error object represents an error conflict in the Error Object.

ErrorConflict Properties

ErrorConflict.Application

ErrorConflict.ConflictObject

ErrorConflict.ConflictObjectDesc

ErrorConflict.ConflictObjectType

ErrorConflict.Name

ErrorConflict.ObjectType

ErrorConflict.Parent

The Jumper Object

The Jumper object represents a physical jumper that exists in the open design.

Jumper Properties

Jumper.Application

Jumper.Installed

Jumper.Length

Jumper.Name

Jumper.Net

Jumper.ObjectType

Jumper.Orientation

Jumper.Parent

Jumper.Points

Jumper.Selected

The Label Object

The Label object represents a physical label object in the open design.

Label Properties

Label.Application

Label.Attribute

Label.Component

Label.Display

Label.Name

Label.ObjectType

Label.Parent

Label.RightReading

Label.Selected

Label.Text

Label.Type

Label Methods

Label.Delete

The Layer Object

This object represents a design layer.

Layer Properties

Layer.Application

Layer.CopperThickness

Layer.Enabled

Layer.Name

Layer.Number

Layer.ObjectType

Layer.Parent

Layer.PlaneType

Layer.RoutingDirection

Layer.Type

Layer.Visible

Layer Methods

Layer.GetColor

Layer.GetDielectricConstant

Layer.GetDielectricThickness

Layer.GetDielectricType

Layer.SetColor

Layer.SetDielectricConstant

Layer.SetDielectricThickness

Layer.SetDielectricType

The Library Object

The Library object represents a library included in the PADS Layout library list. The library corresponds to the set of four library files, with extensions .pt07, .pd07, .ln07, and .ld07.

Library Properties

Library.Application

Library.FullName

Library.Name

Library.ObjectType

Library.Parent

Library.Path

Library Methods

Library.GetLibraryItems

Library.ImportLibraryItems

Library.ImportLibraryItems2

The LibraryItem Object

The LibraryItem object represents a physical item in a particular part library.

LibraryItem Properties

LibraryItem.Application

LibraryItem.Library

LibraryItem.Name

LibraryItem.ObjectType

LibraryItem.Parent

LibraryItem.Type

The Measure Object

The Measure object provides access to the internal PADS Layout unit parser. The Measure object can be either constructed from Application object (see Application.Measure method) or obtained from the object (see Attribute.Measure property). From the measure object, you can extract information about real value and unit.

Measure Properties

Measure.Application

Measure.Name

Measure.Number

Measure.Normalize

Measure.ObjectType

Measure.Parent

Measure.Prefix

Measure.Text

Measure.Unit

Measure.Value

The Net Object

The Net object represents a physical net in the open design.

Net Properties

Net.Application

Net.AssociatedNet

Net.Attributes

Net.Connections

Net.Drawings

Net.Length

Net.Name

Net.NetClass

Net.NetClassAttributes

Net.ObjectType

Net.Parent

Net.Pins

Net.Power

Net.Selected

Net.Vias

The NetClass Object

The NetClass object represents a netclass object in the open design.

NetClass Properties

NetClass.Application

NetClass.Attributes

NetClass.Name

NetClass.Name

NetClass.ObjectType

NetClass.Parent

The Objects Collection Object

The Objects collection object is a collection of homogeneous or heterogeneous database objects, such as Component objects, Jumper objects, Net objects, Pin objects, Via objects, Connection objects, and RouteSegment objects, in the open design. This object is usually retrieved using the Document.GetObjects method or using database object specific properties such as Document.Components, Document.Nets, Document.Vias, Document.Jumpers, Document.Connections, and Document.RouteSegments.

Objects Properties

Objects.Application

Objects.Count

Objects.Item

Objects.ItemType

Objects.Next

Objects.ObjectType

Objects.Parent

Objects.ParentObject

Objects Methods

Objects.Add

Objects.Merge

Objects.Remove

Objects.Reset

Objects.Select

Objects.Sort

The Pad Object

This object represents a pad in the padstack definition.

Pad Properties

Pad.Application

Pad.CornerRadius

Pad.CornerType

Pad.Diameter

Pad.InnerDiameter

Pad.Length

Pad.Name

Pad.ObjectType

Pad.Offset

Pad.Orientation

Pad.PadStackLayer

Pad.Parent

Pad.Shape

Pad.Width

The PadStackLayer Object

The PadStackLayer object represents a layer in the padstack definition.

PadStackLayer Properties

PadStackLayer.AntiPad

PadStackLayer.Application

PadStackLayer.Name

PadStackLayer.Number

PadStackLayer.ObjectType

PadStackLayer.Pad

PadStackLayer.Parent

PadStackLayer.Pin

PadStackLayer.ThermalPad

PadStackLayer.Via

The Pin Object

The Pin object represents a physical pin in the open design.

Pin Properties

Pin.Application

Pin.Attributes

Pin.Component

Pin.DrillSize

Pin.ElectricalType

Pin.FunctionName

Pin.Glued

Pin.Highlighted

Pin.IsSMD

Pin.Name

Pin.Net

Pin.Number

Pin.ObjectType

Pin.PadStackLayers

Pin.Parent

Pin.PlaneThermal

Pin.Plated

Pin.PositionX

Pin.PositionY

Pin.Selected

Pin.SlotLength

Pin.SlotOffset

Pin.SlotOrientation

Pin.Plated

Pin.TestPoint

The PartType Object

The PartType object represents a part type (package) of a physical component in the PCB design currently open in PADS Layout.

PartType Properties

PartType.Application

PartType.Attributes

PartType.Components

PartType.ECORegistered

PartType.Logic

PartType.Name

PartType.ObjectType

PartType.Parent

PartType.Selected

The Polyline Object

The Polyline object represents a physical polyline object in the open design.

Polyline Properties

Polyline.Application

Polyline.CenterX

Polyline.CenterY

Polyline.Geometry

Polyline.Layer

Polyline.LineWidthh

Polyline.ObjectType

Polyline.OutlineType

Polyline.Parent

Polyline.Points

Polyline.Radius

Polyline.ShapeType

The RouteSegment Object

The RouteSegment object represents a physical route segment, also called a route, in the open design.

RouteSegment Properties

RouteSegment.Application

RouteSegment.Layer

RouteSegment.Length

RouteSegment.Name

RouteSegment.Net

RouteSegment.ObjectType

RouteSegment.Parent

RouteSegment.Points

RouteSegment.SegmentType

RouteSegment.Selected

RouteSegment.Width

The SBP Object

The SBP object represents the physical substrate bond pad of a die part. You can retrieve the SBP object, or collection of SBP objects, only from the Component object, which represents a die part (Component.IsDiePart returns True), or from other die part constituents (such as CBP or Wirebond).

SBP objects are not selectable.

You cannot retrieve the collection of all SBP objects from a document using Document.GetObjects (ppcbObjectTypeSBP). This method returns an empty collection. Also, Document.GetObjects (ppcbObjectTypeAll) does not return CBP, SBP, or Wirebond objects. You can get only collections of CBPs, SBPs, or Wirebonds using the Component object methods, if the component is a die part (Component.IsDiePart).

SBP Properties

SBP.Application

SBP.CBPs

SBP.Component

SBP.Function

SBP.Layer

SBP.Length

SBP.Name

SBP.ObjectType

SBP.Orientation

SBP.Parent

SBP.Position X

SBP.PositionY

SBP.Shape

SBP.Tier

SBP.Width

SBP.Wirebonds

The Text Object

The Text object represents free text, or text associated with a 2D line in the open design.

Text Properties

Text.Application

Text.Drawing

Text.Height/Label.Height

Text.HorzJustification/Label.HorzJustification

Text.Layer/Label.Layer

Text.LineWidth/Label.LineWidth

Text Methods

The ThermalPad Object

This object represents a thermal pad in the padstack definition.

ThermalPad Properties

ThermalPad.SpokeWidth

The Via Object

The Via object represents a physical via in the open design.

Via Properties

Via.Application

Via.Attributes

Via.DrillSize

Via.EndLayer

Via.Glued

Via.Highlighted

Via.Name

Via.Net

Via.ObjectType

Via.PadStackLayers

Via.Parent

Via.PlaneThermal

Via.Plated

Via.PositionX

Via.PositionY

Via.Selected

Via.StartLayer

Via.TestPoint

Via.Type

The View Object

The View object represents the current view window, showing the open design. This object is usually retrieved using the Document.ActiveView property.

View Properties

View.Application

View.BottomRightX

View.BottomRightY

View.CenterX

View.CenterY

View.Name

View.ObjectType

View.Parent

View.PointerX

View.PointerY

View.TopLeftX

View.TopLeftY

View.Zoom

View Methods

View.Pan

View.Refresh

View.SetExtents

View.SetExtentsToAll

View.SetExtentsToBoard

View.SetExtentsToSelection

View.SetScale

View Events

View.Change

The Wirebond Object

The Wirebond object represents the physical bond wire of a die part. You can retrieve the Wirebond object, or collection of Wirebond objects, only from the Component object, which represents a die part (Component.IsDiePart), or from other die part constituents (such as CBP or SBP).

Wirebond objects are not selectable.

You cannot retrieve the collection of all Wirebond objects from a document using Document.GetObjects (ppcbObjectTypeWirebond). This method returns an empty collection. Also, Document.GetObjects (ppcbObjectTypeAll) does not return CBP, SBP, or Wirebond objects. You can get only collections of CBPs, SBPs, or Wirebonds using the Component object methods, if the component is a die part (Component.IsDiePart).

Wirebond Properties

Wirebond.Angle

Wirebond.Application

Wirebond.Component

Wirebond.EndOffsetX

Wirebond.EndOffsetY

Wirebond.EndPad

Wirebond.EndX

Wirebond.EndY

Wirebond.Name

Wirebond.ObjectType

Wirebond.Parent

Wirebond.StartOffsetX

Wirebond.StartOffsetY

Wirebond.StartPad

Wirebond.StartX

Wirebond.StartY

Constants

PPcbAntiPadShape

PPcbASCIISections

PPcbASCIIVersion

PPcbAttrFlags

PPcbBondPadEdge

PPcbBondPadShape

PPcbDesignObject

PPcbDielectricLayer

PPcbDielectricType

PPcbDocumentColor

PPcbDrawingType

PPcbDRCMode

PPcbGridType

PPcbHorizontalJustification

PPcbLabelType

PPcbLabelDisplayMode

PPcbLayerColor

PPcbLayerType

PPcbLibraryItemType

PPcbMeasureFormat

PPcbNudgeMode

PPcbObjectType

PPcbOriginType

PPcbOutlineType

PPcbPadCornerType

PPcbPadShape

PPcbPadStackLayerType

PPcbPinElectricalType

PPcbPlaneType

PPcbRightReadingStatus

PPcbRoutingDirection

PPcbSegmentType

PPcbShapeType

PPcbTestPointType

PPcbThermalPadShape

PPcbUnit

PPcbVerticalJustification

PPcbAntiPadShape

ppcbAntiPadShapeRound= 8

ppcbAntiPadShapeSquare= 9

PPcbASCIISections

Possible ASCII section values are:

ppcbASCIISectionPCB = &H00000001

ppcbASCIISectionReuse = &H00000002

ppcbASCIISectionText = &H00000004

ppcbASCIISectionLines = &H00000008

ppcbASCIISectionClusters = &H00000010

ppcbASCIISectionVias = &H00000020

ppcbASCIISectionDecals = &H00000040

ppcbASCIISectionPartTypes = &H00000080

ppcbASCIISectionParts = &H00000100

ppcbASCIISectionJumpers = &H00000200

ppcbASCIISectionConnections = &H00000400

ppcbASCIISectionRoutes = &H00000800

ppcbASCIISectionTeardrops = &H00001000

ppcbASCIISectionMisc = &H00002000

ppcbASCIISectionRules = &H00004000

ppcbASCIISectionCAM = &H00008000

ppcbASCIISectionPour = &H00010000

ppcbASCIISectionAssemblyOptions= &H00020000

ppcbASCIISectionTestPoints = &H00040000

ppcbASCIISectionAttributes = &H00080000

ppcbASCIISectionAll = &HFFFFFFFF

PPcbASCIIVersion

Possible ASCII version values are:

ppcbASCIIVerCurrent = 0

ppcbASCIIVer1_1 = 2

ppcbASCIIVer1_5 = 3

ppcbASCIIVer2_0 = 4

ppcbASCIIVer2_1 = 5

ppcbASCIIVer2_5 = 6

ppcbASCIIVer3_0 = 7

ppcbASCIIVer4_0 = 8

PPcbAttrFlags

Possible attribute flags values are:

ppcbAttrNone = &H00000001

ppcbAttrPart = &H00000002

ppcbAttrNet = &H00000004

ppcbAttrPin = &H00000008

ppcbAttrVia = &H00000010

ppcbAttrPCB = &H00000020

ppcbAttrPartType = &H00000040

ppcbAttrDecal = &H00000080

ppcbAttrNetClass = &H00000100

ppcbAttrAll = &HFFFFFFFF

PPcbBondPadEdge

Restriction: This information applies to only the BGA toolkit.

Possible bond pad edge values are:

ppcbBondPadEdgeUnknown = 0

ppcbBondPadEdgeLeft = 1

ppcbBondPadShapeTop = 2

ppcbBondPadShapeRight = 3

ppcbBondPadShapeBottom = 4

PPcbBondPadShape

Restriction: This information applies to only the BGA toolkit.

Possible bond pad shape values are:

ppcbBondPadShapeUnknown = 0

ppcbBondPadShapeRectangle = 1

ppcbBondPadShapeOval = 2

PPcbDesignObject

ppcbDesignObjectTrace = 0

ppcbDesignObjectVia = 1

ppcbDesignObjectPad = 2

ppcbDesignObjectCopper = 3

ppcbDesignObjectLine = 4

ppcbDesignObjectText = 5

ppcbDesignObjectError = 6

ppcbDesignObjectOutlineTop = 7

ppcbDesignObjectOutlineBottom = 8

ppcbDesignObjectRefDes = 9

ppcbDesignObjectPartType = 10

ppcbDesignObjectAttribute = 11

ppcbDesignObjectKeepout = 12

ppcbDesignObjectPinNumber = 13

PPcbDielectricLayer

ppcbDielectricLayerAbove = 0

ppcbDielectricLayerBelow = 1

PPcbDielectricType

ppcbDielectricTypeCoating = 0

ppcbDielectricTypeSubstrate = 1

ppcbDielectricTypePrepreg = 2

PPcbDocumentColor

ppcbDocumentColorBackground = 0

ppcbDocumentColorSelection = 1

ppcbDocumentColorHighlight = 2

ppcbDocumentColorBoardOutline = 3

ppcbDocumentColorConnection = 4

PPcbDrawingType

Possible drawing type values are:

ppcbDrw2Dline = 0

ppcbDrwBoard = 1

ppcbDrwCopper = 3

ppcbDrwCopperPour = 6

ppcbDrwCopperHatch = 7

ppcbDrwCopperThermal = 8

ppcbDrwKeepout = 9

PPcbDRCMode

Possible Design Rules Checking (DRC) mode values are:

ppcbDRCNone = 0

ppcbDRCOff = 1

ppcbDRCWarn = 2

ppcbDRCIgnoreClearance = 3

ppcbDRCPrevent = 4

PPcbErrorClass

Possible Error Class values are:

ppcbErrorClassClearanceError = 0

ppcbErrorClassConnectivityError = 0x20

ppcbErrorClassHighSpeedError = 0x40

ppcbErrorClassMiscError = 0x60

ppcbErrorClassTestPointError = 0x80

ppcbErrorClassDfmError = 0xa0

ppcbErrorClassBgaError = 0xc0

ppcbErrorClassViaCountError = 0xe0

PPcbErrorType

Possible Error Type values are:

ppcbErrorTypePadToPadError = 0 | 0

ppcbErrorTypePadToTrackError = 0 | 0x1

ppcbErrorTypeTrackToTrackError = 0 | 0x2

ppcbErrorTypeCopperPourError = 0 | 0x3

ppcbErrorTypeDrillHoleError = 0 | 0x4

ppcbErrorTypeDrillToDrillError = 0 | 0x5

ppcbErrorTypeTraceWidthError = 0 | 0x6

ppcbErrorTypePlacementKeepoutError = 0 | 0x7

ppcbErrorTypeHeightKeepoutError = 0 | 0x8

ppcbErrorTypeDrillKeepoutError = 0 | 0x9

ppcbErrorTypeTraceKeepoutError = 0 | 0xa

ppcbErrorTypePourKeepoutError = 0 | 0xb

ppcbErrorTypeViaKeepoutError = 0 | 0xc

ppcbErrorTypeTestPointKeepoutError = 0 | 0xd

ppcbErrorTypeBoardOutlineError = 0 | 0xe

ppcbErrorTypeSameNetPadToPadError = 0 | 0xf

ppcbErrorTypeSameNetPadToTrackError = 0 | 0x10

ppcbErrorTypeBodyToBodyError = 0 | 0x11

ppcbErrorTypeConnectivityError = 0x20 | 0

ppcbErrorTypeConnectivityDrillError = 0x20 | 0x1

ppcbErrorTypeConnectivityPlaneError = 0x20 | 0x2

ppcbErrorTypeDrillLayerPairingError = 0x20 | 0x3

ppcbErrorTypeDrillToPlaneShortError = 0x20 | 0x4

ppcbErrorTypeEDC_CapacitanceError = 0x40 | 0

ppcbErrorTypeEDC_LengthError = 0x40 | 0x1

ppcbErrorTypeEDC_DelayError = 0x40 | 0x2

ppcbErrorTypeEDC_MinImpedanceError = 0x40 | 0x3

ppcbErrorTypeEDC_MaxImpedanceError = 0x40 | 0x4

ppcbErrorTypeEDC_LoopError = 0x40 | 0x5

ppcbErrorTypeEDC_StubError = 0x40 | 0x6

ppcbErrorTypeEDC_ParallelismError = 0x40 | 0x7

ppcbErrorTypeTiePlaneError = 0x60 | 0

ppcbErrorTypeTearDropGenError = 0x60 | 0x1

ppcbErrorTypeLatiumMarkerError = 0x60 | 0x2

ppcbErrorTypeBLZ_ViaAtSMDFitInsideError = 0x60 | 0x3

ppcbErrorTypeBLZ_ViaAtSMDCenterError = 0x60 | 0x4

ppcbErrorTypeBLZ_ViaAtSMDEndError = 0x60 | 0x5

ppcbErrorTypeBLZ_ViaAtSMDCenterOutError = 0x60 | 0x6

ppcbErrorTypeBLZ_ViaAtSMDTooManyError = 0x60 | 0x7

ppcbErrorTypeBLZ_GapBadError = 0x60 | 0x8

ppcbErrorTypeBLZ_GapViolationError = 0x60 | 0x9

ppcbErrorTypeBLZ_GapObstacleSizeError = 0x60 | 0xa

ppcbErrorTypeBLZ_GapObstacleCountError = 0x60 | 0xb

ppcbErrorTypeBLZ_GapIrregularLengthError = 0x60 | 0xc

ppcbErrorTypeBLZ_GapControlledPercentError = 0x60 | 0xd

ppcbErrorTypeBLZ_LengthToleranceError = 0x60 | 0xe

ppcbErrorTypeBLZ_ProfileCornerError = 0x60 | 0xf

ppcbErrorTypeTestPointGenError = 0x80 | 0

ppcbErrorTypeTestPointProbeToProbeError = 0x80 | 0x1

ppcbErrorTypeTestPointProbeToComponentError = 0x80 | 0x2

ppcbErrorTypeTestPointProbeToBoardError = 0x80 | 0x3

ppcbErrorTypeTestPointProbeToKeepoutError = 0x80 | 0x4

ppcbErrorTypeTestPointNailCountError = 0x80 | 0x5

ppcbErrorTypeBLZ_ProbeNotComponentError = 0x80 | 0x6

ppcbErrorTypeBLZ_ProbeOnSMDPinError = 0x80 | 0x7

ppcbErrorTypeBLZ_ProbeGridError = 0x80 | 0x8

ppcbErrorTypeBLZ_ProbeNailDiameterError = 0x80 | 0x9

ppcbErrorTypeBLZ_ProbeToPadError = 0x80 | 0xa

ppcbErrorTypeBLZ_ProbeToTraceError = 0x80 | 0xb

ppcbErrorTypeDFMGenError = 0xa0

ppcbErrorTypeDFMAcidTrapError = 0xa0 | 0x1

ppcbErrorTypeDFMSliverError = 0xa0 | 0x2

ppcbErrorTypeDFMSolderBridgeError = 0xa0 | 0x3

ppcbErrorTypeDFMStarvedThermalError = 0xa0 | 0x4 ppcbErrorTypeDFMLayerCompareError = 0xa0 | 0x5

ppcbErrorTypeDFMDrillError = 0xa0 | 0x6

ppcbErrorTypeDFFGenError = 0xa0 | 0x7

ppcbErrorTypeDFFAcidTrapError = 0xa0 | 0x8

ppcbErrorTypeDFFSliverError = 0xa0 | 0x9

ppcbErrorTypeDFFSolderBridgeError = 0xa0 | 0xa

ppcbErrorTypeDFFStarvedThermalError = 0xa0 | 0xb

ppcbErrorTypeDFFLayerCompareError = 0xa0 | 0xc

ppcbErrorTypeDFFDrillError = 0xa0 | 0xd

ppcbErrorTypeDFFDrillToMaskError = 0xa0 | 0xe

ppcbErrorTypeDFFPadToMaskError = 0xa0 | 0xf

ppcbErrorTypeDFFDrillToPadError = 0xa0 | 0x10

ppcbErrorTypeDFFTraceWidthError = 0xa0 | 0x11

ppcbErrorTypeDFFPadSizeError = 0xa0 | 0x12

ppcbErrorTypeBGA_GenError = 0xc0

ppcbErrorTypeBGA_WBRMinLengthError = 0xc0 | 0x1

ppcbErrorTypeBGA_WBRMaxLengthError = 0xc0 | 0x2

ppcbErrorTypeBGA_WBRMaxAngleError = 0xc0 | 0x3

ppcbErrorTypeBGA_WBRWBToWBError = 0xc0 | 0x4

ppcbErrorTypeBGA_WBRWBToSBPError = 0xc0 | 0x5

ppcbErrorTypeViaCountError = 0xe0

PPcbErrorValueType

Possible Error Value Type values are:

ppcbErrorValueTypeMeasure = 0

ppcbErrorValueTypeInt = 1

ppcbErrorValueTypeDouble = 2

PPcbGridType

Possible grid type values are:

ppcbGridNone = 0 None.

ppcbGridDesign = 1 PADS Layout design grid.

ppcbGridVia = 2 PADS Layout via grid.

ppcbGridDisplay = 3 PADS Layout display grid.

ppcbGridAll = 9999 All PADS Layout grids.

PPcbHorizontalJustification

Possible horizontal justification values are:

ppcbJustifyLeft = 0

ppcbJustifyHCenter = 1

ppcbJustifyRight = 2

PPcbLabelType

Possible label type values are:

ppcbLabelTypeRefDesignator = 0

ppcbLabelTypePartType = 1

ppcbLabelTypeAttribute = 2

PPcbLabelDisplayMode

Possible label display mode values are:

ppcbLabelDisplayNone = 0

ppcbLabelDisplayValue = 1

ppcbLabelDisplayNameAndValue = 2

ppcbLabelDisplayFullNameAndValue = 3

PPcbLayerColor

ppcbLayerColorTrace = 0

ppcbLayerColorVia = 1

ppcbLayerColorPad = 2

ppcbLayerColorCopper = 3

ppcbLayerColorLine = 4

ppcbLayerColorText = 5

ppcbLayerColorError = 6

ppcbLayerColorOutlineTop = 7

ppcbLayerColorOutlineBottom = 8

ppcbLayerColorRefDes = 9

ppcbLayerColorPartType = 10

ppcbLayerColorAttribute = 11

ppcbLayerColorKeepout = 12,

ppcbLayerColorPinNumber = 13

PPcbLayerType

Possible layer type values are:

ppcbLayerUnknown = 0

ppcbLayerComponent = 1

ppcbLayerRouting = 2

ppcbLayerDrill = 3

ppcbLayerSilkscreen = 4

ppcbLayerPasteMask = 5

ppcbLayerSolderMask = 6

ppcbLayerAssembly = 7

ppcbLayerGeneral = 8

ppcbLayerAll = 9999

PPcbLibraryItemType

Possible library item type values are:

ppcbLibraryItemTypePartType = 0

ppcbLibraryItemTypeDecal = 1

ppcbLibraryItemTypeLogicDrawing = 2

ppcbLibraryItemTypeDrawing = 3

ppcbLibraryItemTypeAll = 9999

PPcbMeasureFormat

Possible PADS Layout measure format values are:

ppcbMeasureFormatStandard = 0

ppcbMeasureFormatCurrent = 1

ppcbMeasureFormatShort = 2

ppcbMeasureFormatLong = 3

PPcbNudgeMode

Possible nudge mode values are:

ppcbNudgeNone = 0

ppcbNudgeOff = 1

ppcbNudgeWarn = 2

ppcbNudgeAuto = 3

PPcbObjectType

Possible database object type values are:

Tip: ppcbObjectTypeCBP, ppcbObjectTypeSBP, and ppcbObjectTypeWirebond are available in the Advanced Packaging Toolkit only.

ppcbObjectTypeUnknown = 0
The server may return this value to indicate an invalid object. The client may use this value when trying to work with empty object collections.

ppcbObjectTypeComponent = 1

ppcbObjectTypeNet = 2

ppcbObjectTypePin = 3

ppcbObjectTypeVia = 4

ppcbObjectTypeConnection = 5

ppcbObjectTypeRouteSegment = 6

ppcbObjectTypeJumper = 7

ppcbObjectTypePartType = 8

ppcbObjectTypeCBP = 9

ppcbObjectTypeSBP = 10

ppcbObjectTypeWirebond = 11

ppcbObjectTypeNetClass = 12

ppcbObjectTypeDrawing = 13

ppcbObjectTypeText = 14

ppcbObjectTypeLabel = 15

ppcbObjectTypePolyline = 16

ppcbObjectTypeCircle = 17

ppcbObjectTypeLibrary = 18

ppcbObjectTypeLibraryItem = 19

ppcbObjectTypeApplication = 20

ppcbObjectTypeAttribute = 21

ppcbObjectTypeAttributeType = 22

ppcbObjectTypeDocument = 23

ppcbObjectTypeMeasure = 24

ppcbObjectTypeView = 25

ppcbObjectTypeAssemblyOptions = 26

ppcbObjectTypeAttributes = 27

ppcbObjectTypeAttributeTypes = 28

ppcbObjectTypeObjects = 29

ppcbObjectTypePadStackLayer = 31

ppcbObjectTypePad = 32

ppcbObjectTypeThermalPad = 33

ppcbObjectTypeAntiPad = 34

ppcbObjectTypeLayer = 35

ppcbObjectTypeError = 36

ppcbObjectTypeErrorConflict = 37

ppcbObjectTypeAssociatedNet = 38

ppcbObjectTypeAll = 9999

All PADS Layout Automation database object types, including the Component, Net, Pin, Via, Connections, and RouteSegment objects, but not the CBP, SBP, and Wirebond objects.

This specifically means that if you use the Document.GetObjects (ppcbObjectTypeAll) method, the collection of objects returned does not include any CBP, SBP, or Wirebond objects.

PPcbOriginType

Possible origin type values are:

ppcbOriginTypeDesign = 0

ppcbOriginTypeParent = 1

PPcbOutlineType

Possible outline type values are:

ppcbOutlineTypeCenter = 0

ppcbOutlineTypeOuter = 1

ppcbOutlineTypeInner = 2

PPcbPadCornerType

ppcbPadCornerType90Degree = 0

ppcbPadCornerTypeChamfered = 1

ppcbPadCornerTypeRounded = 2

PPcbPadShape

ppcbPadShapeOvalFinger = 0

ppcbPadShapeRectangularFinger = 1

ppcbPadShapeRound = 2

ppcbPadShapeSquare = 3

ppcbPadShapeAnnular = 4

ppcbPadShapeOdd = 5

PPcbPadStackLayerType

ppcbPadStackLayerTypeMounted = -2

ppcbPadStackLayerTypeInner = -1

ppcbPadStackLayerTypeOpposite = 0

PPcbPinElectricalType

Possible gate electrical type values are:

ppcbElectricalTypeUnknown = 0

ppcbElectricalTypeSource = 1

ppcbElectricalTypeBidirectional = 2

ppcbElectricalTypeOpenCollector = 3

ppcbElectricalTypeOrTieableSource = 4

ppcbElectricalTypeTristate = 5

ppcbElectricalTypeLoad = 6

ppcbElectricalTypeTerminator = 7

ppcbElectricalTypePower = 8

ppcbElectricalTypeGround = 9

PPcbPlaneType

ppcbPlaneTypeNoPlane = 0

ppcbPlaneTypeCAMPlane = 1

ppcbPlaneTypeSplitMixedPlane = 2

PPcbRightReadingStatus

Possible right-reading status values are:

ppcbRightReadingNone = 0

ppcbRightReadingOrthogonal = 1

ppcbRightReadingAngled = 2

PPcbRoutingDirection

ppcbRoutingDirectionHorizontal = 0

ppcbRoutingDirectionVertical = 1

ppcbRoutingDirectionAny = 2

ppcbRoutingDirectionDiagonal45 = 3

ppcbRoutingDirectionDiagonal135 = 4

PPcbSegmentType

Possible segment type values are:

ppcbSegmentUnknown = 0

ppcbSegmentLine = 1

ppcbSegmentArc = 2

PPcbShapeType

Possible shape type values are:

ppcbShapeTypeOpen = 0

ppcbShapeTypeHollow = 1

ppcbShapeTypeFilled = 2

ppcbShapeTypeVoid = 3

PPcbTestPointType

Possible test point type values are:

ppcbTestPointNone = 0 No test point.

ppcbTestPointTopLayer = 1 Test point on top layer.

ppcbTestPointBottomLayer = 2 Test point on bottom layer.

PPcbThermalPadShape

ppcbThermalPadShapeRound = 6

ppcbThermalPadShapeSquare = 7

PPcbUnit

Possible unit type values are:

ppcbUnitCurrent = 0 Current unit in use in PADS Layout.

ppcbUnitDatabase = 1 Internal PADS Layout database unit.

ppcbUnitMils = 2 Mils unit (1/1000 of an inch).

ppcbUnitInch = 3 Inch unit.

ppcbUnitMetric = 4 Metric unit (1/1000 of a meter).

PPcbVerticalJustification

Possible vertical justification values are:

ppcbJustifyTop = 0

ppcbJustifyVCenter = 1

ppcbJustifyBottom = 2

AntiPad.Application

This property returns the application object.

Prototype

Application as Application

Argument

None

AntiPad.CornerRadius

This property returns the Anti Pad corner radius.

Prototype

```
CornerRadius as Double
      CornerRadius (unit as PPcbUnit)  as Double
```

Argument

unit - [Optional]	Unit in which the value is returned.

AntiPad.CornerType

This property returns the Anti Pad corner type.

Prototype

```
CornerType as PPcbPadCornerType
```

Argument

None

AntiPad.Length

This property returns the Anti Pad length.

Prototype

```
Length as Double
      Length (unit as PPcbUnit)  as Double
```

Argument

unit - [Optional]   Unit in which the value is returned.

AntiPad.Name

This property returns the name of this anti pad.

Prototype

```
Name as String
```

Argument

None

AntiPad.ObjectType

This property returns the type of the object - ppcbObjectTypeAntiPad.

Prototype

```
ObjectType as PPcbObjectType
```

Argument

None

AntiPad.Offset

This property returns the Anti Pad offset.

Prototype

```
Offset as Double
      Offset  (unit as PPcbUnit)  as Double
```

Argument

unit - [Optional]   Unit in which the value is returned.

AntiPad.Orientation

This property returns the Anti Pad orientation.

Prototype

```
Orientation as Double
```

Argument

None

AntiPad.PadStackLayer

This property returns the PadStackLayer Object to which this anti pad belongs to.

Prototype

```
PadStackLayer as PadStackLayer
```

Argument

None

AntiPad.Parent

This property returns the parent of the object.

Prototype

```
Parent as Document
```

Argument

None

AntiPad.Shape

This property returns the anti pad shape.

Prototype

```
Shape as PPcbAntiPadShape
```

Argument

None

AntiPad.Size

This property returns the anti pad's size. For shape ppcbAntiPadShapeRound it returns diameter.

Prototype

```
Size (unit as PPcbUnit) as Double
```

Argument

unit	[Optional] Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

Application.ActiveDocument

This property returns the active document.

Prototype

ActiveDocument As Document

Arguments

None

Comments

The active document represents the open design.

Sample

The following sample code retrieves the name of the open design using the Document.Name property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "You are working with " & ActiveDocument.Name
End Sub
```

See Also

Application.OpenDocument Method, Document.Name

Application.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an Automation object. All Automation server applications have an Application object and all Automation objects have an application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

Application.DefaultFilePath

This property sets or returns the path that the program uses to open design files.

Prototype

DefaultFilePath As String

Arguments

None

Comments

This property checks the FileDir folder entry in the powerpcb.ini file. When you set this property to a new value, the .ini file entry also changes.

For example, “C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs” is the default path when you install using the default installation settings.

Sample

The following sample code changes the default file path and notifies the client of this change. See “Running Code Samples” for more information on running this sample.

```
Sub Main
oldPath = DefaultFilePath
DefaultFilePath = "C:\TEMP"
MsgBox "The default file path used to be " & oldPath & " and it was just
changed to " & DefaultFilePath
End Sub
```

Application.FullName

This property returns the file name of the application, including its path.

Prototype

FullName As String

Arguments

None

Comments

For example, this function can return the string ”
C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\powerpcb.exe.”

Sample

The following sample code displays the common name and the actual .exe name of the program. See “Running Code Samples” for more information on running this sample.

```
Sub Main
MsgBox "Hi, my name is " & Name & " and I am located in " & FullName
End Sub
```

See Also

Application.Name, Application.Version

Application.Libraries

This property returns a collection of available libraries, or a specific library.

Prototype

Libraries as Collection

Libraries(*Name* as String) as Library

Arguments

name Name of the library to retrieve. Should not include wildcards.

Comments

None

Sample

The following sample code displays the number of available libraries. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "Number of libraries: " & Libraries.Count
End Sub
```

Application.Name

This property returns the name of the application.

Prototype

Name As String

Arguments

None

Comments

In PADS Layout this property returns the string “PowerPCB.”

This property is the default property for the Application object.

Sample

The following sample code displays the common name, the version, and the actual .exe name for the program, depending on which application you are running. See “Running Code Samples” for more information on running this sample.

```
Sub Main
MsgBox "Hi, my name is " & Name & " version " & Version & " and I am
located in " & FullName
End Sub
```

See Also

Application.FullName, Application.Version

Application.ObjectType

This property returns the type of this object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Application.Parent

This property returns the parent of the object.

Prototype

Parent As Application

Arguments

None

Comments

None

Application.Preference

This property sets or returns a preference.

Prototype

Preference(*name* As String) As Variant

Argument

name Name of the preference.

Comments

The following are possible *name* argument values:

DRC	Sets or returns the DRC mode of the open design. The DRC mode values are of type PPcbDRCMode.
Nudge	Sets or returns the Nudge mode of the open design. The Nudge mode values are of type .
ModifyUnionMember	Sets or returns the ability (True/False) to move an individual component when that component belongs to a union.

This property generates an exception if the name argument is not a valid document preference.

Sample

The following sample code demonstrates how to use this property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
' Set the DRC mode to "Prevent"
ActiveDocument.Preference("DRC") = ppcbDRCPrevent
' Set the Nudge mode to "Automatic"
ActiveDocument.Preference("Nudge") = ppcbNudgeAuto
End Sub
```

Application.ProgressBar

This property sets or returns current value of the status bar, as a percentage.

Prototype

ProgressBar As Integer

Arguments

None

Comments

This property allows you to retrieve the current status of long batch processes that are running, or to display the status of long Basic scripts.

Set the value greater than 0 or less than 100 to deactivate the status bar.

Use this property with Application.StatusBarText.

Sample

The following sample demonstrates this property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
 StatusBarText = "My Batch Process ..." 'show progress text
 For i = 0 to 100
  ProgressBar = i
 Next
 ProgressBar = -1 'deactivate progress bar
 StatusBarText = "" 'hide progress text
End Sub
```

See Also

Application.ProgressChange

Application.StatusBarText

This property sets or returns the text in the status bar.

Prototype

StatusBarText As String

Arguments

None

Comments

To set the status bar text to nothing, set this property to an empty string (" ").

Sample

The following sample displays a message on the status bar. See "Running Code Samples" for more information on running this sample.

```
Sub Main
StatusBarText = "Wow! I can even print my own messages in here!"
End Sub
```

Application.Version

This property returns the version.

Prototype

Version As String

Arguments

None

Comments

This property returns the application version as a string with the following format: <major>.<minor>, for example “4.0”.

Sample

The following sample displays the application name and version. See “Running Code Samples” for more information on running this sample.

```
Sub Main
MsgBox "Hi, my name is " & Name & " version " & Version
End Sub
```

See Also

Application.Name, Application.FullName

Application.Visible

This property sets or returns whether the program is visible.

Prototype

Visible As Boolean

Arguments

None

Comments

This property is usually used in the following cases:

- When an Automation client starts the Automation server using an asynchronous OLE Automation call, such as the Visual Basic function CreateObject. The Automation server always starts as invisible (this is a client/server rule). You can use this property to make the program visible.
- When a client attempts to shut down the program (see Application.Quit Method) by making PADS Layout invisible, disconnecting from it, and letting the server shut down appropriately.
- When a client needs to make the server window the active window, so it appears on top of other application windows.

Sample

The following sample makes the program invisible, waits a second, and then makes it visible again. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Visible = False
Wait 1
Visible = True
End Sub
```

AssemblyOptions.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This is a Microsoft-required property.

This property identifies the object as an Automation object. All Automation server applications have an Application object and all Automation objects have an Application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

AssemblyOptions.Count

This property returns the number of Assembly Options.

Prototype

Count As Long

Arguments

None

Comments

None

AssemblyOptions.Item

This property returns an Assembly Option, given its index or its name.

Prototype

Item(*index* As Long) As Document

Item(*name* As String) As Document

Arguments

index	Index (in the collection) of the Assembly Option to retrieve.
name	Name of the Assembly Option to retrieve.

Comments

This is the default member of the AssemblyOptions collection object.

This property generates an exception if the *index* or *name* argument is not valid.

AssemblyOptions.ItemType

This property returns the type of an object given its index.

Prototype

ItemType(*index* As long) As PPcbObjectType

ItemType(*name* As String) As PPcbObjectType

Arguments

index	Index of the object in the collection to query.
name	Name of the object to retrieve.

Comments

This property generates an exception if the *index* argument is not valid.

See Also

AssemblyOptions.Item

AssemblyOptions.Next

This property returns the index of the next object of a specified type after the specified index.

Prototype

Next(*index* As long, *type* As PPcbObjectType) As Long

Arguments

index	Index of the object in the collection to query.
type	Type of the object to query.

Comments

This property generates an exception if the *index* argument is not valid.

If Index = zero (0), then this function returns the index of the first item of the given type. If an item is not found, then the return value is zero (0).

AssemblyOptions.ObjectType

This property returns the type of this object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

AssemblyOptions.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

This is a Microsoft-required property.

AssemblyOptions.ParentObject

This property returns the parent object of the collection.

Prototype

ParentObject as Object

Arguments

None

Comments

If a collection has ParentObject, it means the collection is "active." Addition or deletion of an item to or from the collection modifies the parent object as well. Otherwise, only the collection is affected.

AssociatedNet.Name

This property returns the name of the associated net. For example, this property returns the string ?$$$1*$$$2? for associated net $$$1*$$$2.

This property is the default property for the AssociatedNet object.

Prototype

```
Name as String
```

Arguments

None

AssociatedNet.Nets

This property returns the collection of all nets belonging to the associated net.

Prototype

```
Nets as Objects
Nets (name as String) as Net
```

Arguments

name Optional
Name of an existing net

Return Values

When an existing net name is passed to this property, it returns the net object with that name. If the net name is not specified, this property returns the collection of all nets in Objects.

AssociatedNet.ObjectType

This property returns the type of the object. All application database objects in the Automation server implement this property to compensate for the lack of a Visual Basic equivalent for the Visual C++ QueryInterface function.

Prototype

```
ObjectType as BlazeObjectType
```

Arguments

None

Return Values

This property always returns blazeObjectTypeAssociatedNet.

AssociatedNet.Parent

This property returns the parent of the object.

Prototype

```
Parent as Document
```

Arguments

None

AssociatedNet.Selected

This property sets or determines whether the associated net is selected or not. You can also select an application database object using the Document.SelectObjects and Objects.Select methods.

Prototype

```
Selected as Boolean
```

Arguments

None

Sample

The following sample code selects associated net $$$1*$$$2 only, assuming it exists in the open design. For more information about running this sample code, see Running Code Samples.

```
Sub Main
   ActiveDocument.SelectObjects(,,False)
   ActiveDocument.AssociatedNets("$$$1*$$$2").Selected = True
End Sub
```

See also: Document.SelectionChange Event, Document.SelectObjects, Objects.Select

Attribute.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This is a Microsoft-required property.

This property identifies the object as a PADS Layout Automation object. All Automation server applications have an Application object and all Automation objects have an Application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

Attribute.Name

This property returns the name of the attribute.

Prototype

Name As String

Arguments

None

Comments

None

Sample

The following sample code lists all attributes in the open design and places that list in a custom dialog box. This sample uses the UserDialog Editor in the Sax Basic Engine in PADS Layout. See the Sax Basic Editor On-line Help for more information. See “Running Code Samples” for more information on running this sample.

```
Dim ListAttrs$(10000)
Sub Main
index = 0
For Each nextAttr In ActiveDocument.Attributes
ListAttrs$(index) = nextAttr.Name
index = index + 1
Next nextAttr
' This piece of code is automatically generated by the Basic Dialog Editor
in PADS Layout.
 Begin Dialog UserDialog 180,238,"Attributes" ' %GRID:10,7,1,1
  ListBox 10,7,160,203,ListAttrs(),.ListBox1
  OKButton 10,210,160,21
 End Dialog
Dim dlg As UserDialog
Dialog dlg
End Sub
```

See Also

Attribute.Value

Attribute.ObjectType

This property returns the type of this object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Attribute.Parent

This property returns the parent of the object.

Prototype

Parent As Object

Arguments

None

Comments

This is a Microsoft-required property.

Attribute.Value

This property sets or returns the value of the attribute.

Prototype

Value As Variant

Arguments

None

Comments

The value of an attribute may be of type Boolean, Byte, Single, Integer, PortInt, Long, Double, String, or Measure object.

This property generates an exception if a type mismatch exists between the *value* type and the type of the attribute.

If a value is not defined, an empty string is returned.

Sample 1

The following sample code changes the Cost attribute of all LED components to US$3.99, in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each nextComp In ActiveDocument.Components
If nextComp.PartType = "LED" Then
' Ignore exceptions generated when that attribute does not exist
On Error Resume Next
nextComp.Attributes("PRICE").Value = 3.99
End If
Next nextComp
End Sub
```

Sample 2

The following sample code adds attributes of different types to the level of PCB design and then changes their values. See "Running Code Samples" for more information on running this sample.

```
Sub Main
'Create new file to access some predefined attribute types such as List or
Measure OpenDocument""
'Add attributes of different types to PCB design
With Active Document
'Free text type
.Attributes.Add "Description","Free Text Value"
'Yes/No type
.Attributes.Add "PowerGround", "True"
'Number type
.Attributes.Add "Some Number", 100
```

```
'Decimal Number type
.Attributes.Add "Some Decimal Number", 0.45
'Measure type (in volts)
.Attributes.Add "Voltage", Measure("5V)"
'list type (only values from predefined set allowed)
.Attributes.Add "HyperLynx.Signal Type", "Clock"
End With
'Modify PCB attributes of different types
With ActiveDocument
'Free text type
.Attributes("Description").Value = "Another Free Text Value"
'Yes/No type
.Attributes("PowerGround").Value = False
'Number type
.Attributes("Some Number").Value = 200
'Decimal Number type
.Attributes("Some Decimal Number").Value = 0.25
'Measure type (in volts)
.Attributes("Voltage").Value = Measure("2.5V")
'list type (only values from predefined set allowed)
.Attributes("HyperLynx.Signal Type").Value = "Data"
End With
End Sub
```

See Also

Attribute.Name

Attributes.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This is a Microsoft-required property.

This property identifies the object as a PADS Layout Automation object. All Automation server applications have an Application object and all Automation objects have an Application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

Attributes.Count

This property returns the number of attributes.

Prototype

Count As Long

Arguments

None

Comments

None

Attributes.Item

This property returns an attribute, given its index or its name.

Prototype

Item(*index* As Long) As Attribute

Item(*name* As String) As Attribute

Arguments

index	Index (in the collection) of the attribute to retrieve.
name	Name of the attribute to retrieve.

Comments

This is the default member of the Attributes collection object.

This property generates an exception if the *index* or *name* argument is not valid.

Attributes.ItemType

This property returns the type of an object given its index.

Prototype

ItemType(*index* As long) As PPcbObjectType

ItemType(*name* As String) As PPcbObjectType

Arguments

index	Index of the object in the collection to query.
name	Name of the object to retrieve.

Comments

This property generates an exception if the *index* argument is not valid.

See Also

Attributes.Item

Attributes.Next

This property returns the index of the next object of a specified type after the specified index.

Prototype

Next(*index* As long, type As PPcbObjectType) As Long

Arguments

index	Index of the object in the collection to query.
type	Type of the object to query.

Comments

This property generates an exception if the *index* argument is not valid.

If Index = zero (0), then this function returns the index of the first item of the given type. If an item is not found, then the return value is zero (0).

Attributes.ObjectType

This property returns the type of this object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Attributes.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

This is a Microsoft-required property.

Attributes.ParentObject

This property returns the parent object of the collection.

Prototype

ParentObject as Object

Arguments

None

Comments

If a collection has ParentObject, it means the collection is “active.” Addition or deletion of an item to or from the collection modifies the parent object as well. Otherwise, only the collection is affected.

CBP.Application

This property returns the Application object.

Restriction: This information applies to only the BGA toolkit.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an Automation object. All Automation server applications have an Application object and all Automation objects have an application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

This is a Microsoft-required property.

CBP.Component

This method returns the component object of the CBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Component As Component

Arguments

None

Comments

None

Sample

The following sample code represents a subroutine that depends on the CBP object passed as an argument. The code retrieves the name of the component and the name of the decal to which the CBP belongs. See "Running Code Samples" for more information on running this sample.

```
Sub QueryCBP(aCBP As CBP)
MsgBox "Component Bond Pad" & aCBP.Name &
"belongs to component" & aCBP.Component.Name &
"(" & aCBP.Component.Decal &")"
End Sub
```

CBP.Edge

This property returns the edge of the CBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Edge As PPcbBondPadEdge

Arguments

None

Comments

None

Sample

The following sample code displays information about the edge of each CBP in each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
Select Case aCBP.Edge
Case ppcbBondPadEdgeUnknown
MsgBox aCBP.Name & " Edge: Unknown"
Case ppcbBondPadEdgeLeft
MsgBox aCBP.Name & " Edge: Left"
Case ppcbBondPadEdgeTop
MsgBox aCBP.Name & " Edge: Top"
Case ppcbBondPadEdgeRight
MsgBox aCBP.Name & " Edge: Right"
Case ppcbBondPadEdgeBottom
MsgBox aCBP.Name & " Edge: Bottom"
Case Else
MsgBox aCBP.Name & " Edge: Unknown"
End Select
Next
End If
Next
End Sub
```

CBP.Function

This property returns the signal name of the CBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Function As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the signal name of each CBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
MsgBox aCBP.Name & " Function: " & aCBP.Function
Next
End If
Next
End Sub
```

CBP.Layer

This property returns the LIQ layer of the CBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Layer As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the layer on which each CBP in each die in the open design is mounted. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
MsgBox aCBP.Name & " Layer: " & aCBP.Layer
Next
End If
Next
End Sub
```

CBP.Length

This property returns the length of the CBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Length([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the length value is returned.

Comments

None

Sample

The following sample code retrieves the length of each CBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
MsgBox aCBP.Name & " Length: " & Format (aCBP.Length, "#.###")
Next
End If
Next
End Sub
```

See Also

CBP.Width

CBP.Name

This default property returns the name of the CBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Name As String

Arguments

None

Comments

This is a Microsoft-required property.

Sample

The following sample code retrieves the name of each CBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
MsgBox aCBP.Name
Next
End If
Next
End Sub
```

CBP.ObjectType

This property returns the type of the object.

Restriction: This information applies to only the BGA toolkit.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property always returns ppcbObjectTypeCBP.

This property is generally used to identify the kind of database objects in a heterogeneous Objects collection or when implementing a generic routine that depends on the type of the database object passed as the argument.

All database objects in the Automation server implement this property to compensate for the lack of a Visual Basic equivalent for the Visual C++ QueryInterface function.

Sample

See "Running Code Samples" for more information on running this sample.

```
Sub DoSomethingToDieObject(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeCBP
' Do something specific to CBP objects
Case ppcbObjectTypeSBP
' Do something specific to SBP objects
Case ppcbObjectTypeWirebond
' Do something specific to Wirebond objects
Case Else
MsgBox "Not a Die object"
End Select
End Sub
```

CBP.Parent

This property returns the parent of the object.

Restriction: This information applies to only the BGA toolkit.

Prototype

Parent As Document

Arguments

None

Comments

This is a Microsoft-required property.

CBP.PositionX

This property returns the x-coordinate of the CBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

PositionX([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the x-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the location of each CBP in each die in the open design, in current units. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
MsgBox aCBP.Name & ": (" & aCBP.PositionX & ", " & aCBP.PositionY & ")"
Next
End If
Next
End Main
```

See Also

CBP.PositionY

CBP.PositionY

This property returns the y-coordinate of the CBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

PositionY([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Arguments

unit	[Optional] Unit in which the y-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the location of each CBP in each die in the open design, in current units. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
MsgBox aCBP.Name & ": (" & aCBP.PositionX & ", " & aCBP.PositionY & ")"
Next
End If
Next
End Sub
```

See Also

CBP.PositionX

CBP.SBPs

This property returns the collection of SBPs linked to this CBP object.

Restriction: This information applies to only the BGA toolkit.

Prototype

SBPs As Objects

Arguments

None

Comments

None

Sample

The following sample retrieves the number of SBPs linked to each CBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
MsgBox "Number of SBPs linked to " & aCBP.Name & ": " & aCBP.SBPs.Count
Next
End If
Next
End Main
```

See Also

CBP.Wirebonds

CBP.Shape

This property returns the shape of the CBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Shape As PPcbBondPadShape

Arguments

None

Comments

None

Sample

The following sample retrieves the shape of each CBP on each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
Select Case aCBP.Shape
Case ppcbBondPadShapeUnknown
MsgBox aCBP.Name & " Shape: Unknown"
Case ppcbBondPadShapeRectangle
MsgBox aCBP.Name & " Shape: Rect"
Case ppcbBondPadShapeOval
MsgBox aCBP.Name & " Shape: Oval"
Case Else
MsgBox aCBP.Name & " Shape: Unknown"
End Select
Next
End If
Next
End Sub
```

CBP.Width

This property returns the width of the CBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Width([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the width value is returned.

Comments

None

Sample

The following sample retrieves the width of each CBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
MsgBox aCBP.Name & " Width: " & Format (aCBP.Width, "#.###")
Next
End If
Next
End Sub
```

See Also

CBP.Length

CBP.Wirebonds

This property returns the collection of bond wires attached to the CBP object.

Restriction: This information applies to only the BGA toolkit.

Prototype

Wirebonds As Objects

Arguments

None

Comments

None

Sample

The following sample retrieves the number of bond wires connected to each CBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP in comp.CBPs
MsgBox "Number of WBs connected to " & aCBP.Name & ":" &
aCBP.Wirebonds.Count
Next
End If
Next
End Sub
```

See Also

CBP.SBPs

Circle.Application

This property returns the Application object.

Prototype

Application as Application

Arguments

None

Comments

This is a Microsoft-required property.

Circle.CenterX

This property returns the x-coordinate of the circle's center.

Prototype

CenterX (*Unit* as PPcbUnit, *Origin* as PPcbOriginType) as Double

Arguments

unit	[Optional] Unit in which the center X value is returned. This optional argument is ppcbUnitCurrent by default.
origin	[Optional]Type of reference point from which the result is counted. The default value is ppcbOriginTypeDesign.

Comments

None

Sample

The following sample code shows the position of the selected circle. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
If geom.ObjectType = ppcbObjectTypeCircle Then
MsgBox "Position: (" & geom.CenterX & ", " & geom.CenterY & ")"
End If
Next geom
Next drw
End Sub
```

Circle.CenterY

This property returns the y-coordinate of the circle's center.

Prototype

CenterY (*Unit* as PPcbUnit, *Origin* as PPcbOriginType) as Double

Arguments

unit	[Optional] Unit in which the center Y value is returned. This optional argument is ppcbUnitCurrent by default.
origin	[Optional]Type of reference point from which the result is counted. The default value is ppcbOriginTypeDesign.

Comments

None

Sample

The following sample code shows the position of the selected circle. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
If geom.ObjectType = ppcbObjectTypeCircle Then
MsgBox "Position: (" & geom.CenterX & ", " & geom.CenterY & ")"
End If
Next geom
Next drw
End Sub
```

Circle.Geometry

This property returns a collection of objects, currently polylines, texts, or circles, representing this object's child geometry objects.

Prototype

Geometry as Collection

Arguments

None

Comments

None

Sample

The following sample code shows the number of child objects. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
MsgBox "Child object count: " & geom.Geometry.Count
Next geom
Next drw
End Sub
```

Circle.Layer

This property returns the layer number of the object.

Prototype

Layer as Long

Arguments

None

Comments

None

Sample

The following sample code shows the layer number of the selected circle. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
MsgBox "Layer number: " & geom.Layer
Next geom
Next drw
End Sub
```

Circle.LineWidth

This property returns the line width of the circle.

Prototype

LineWidth (*Unit* as PPcbUnit) as Double

Argument

unit [Optional] Unit in which the line width value is returned. This optional argument is ppcbUnitCurrent by default.

Comments

None

Sample

The following sample code shows the line width of the selected circle. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
MsgBox "LineWidth: " & geom.LineWidth
Next geom
Next drw
End Sub
```

Circle.ObjectType

This property returns the type of object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Sample

The following sample code tests the ObjectType property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each drw In ActiveDocument.Drawings
For Each geom In drw.Geometry
type = geom.ObjectType
If type <> ppcbObjectTypePolyline And type <> ppcbObjectTypeCircle Then
MsgBox "Test failed"
End If
Next geom
Next drw
End Sub
```

Circle.OutlineType

This property returns the outline type of the circle.

Prototype

OutlineType as PPcbOutlineType

Arguments

None

Comments

None

Sample

The following sample code shows the outline type of the selected circle. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
Select Case geom.OutlineType
Case ppcbOutLineTypeCenter
s = "Center line"
Case ppcbOutLineTypeOuter
s = "Outer line"
Case ppcbOutLineTypeInner
s = "Inner line"
End Select
MsgBox "Outline type: " & s
Next geom
Next drw
End Sub
```

Circle.Parent

This property returns the parent of the object.

Prototype

Parent as Document

Arguments

None

Comments

This is a Microsoft-required property.

Circle.Radius

This property returns the value of the radius of the circle.

Prototype

Radius (*Unit* as PPcbUnit) as Double

Argument

Unit	[Optional] Unit in which the radius value is returned. This optional argument is ppcbUnitCurrent by default.

Comments

None

Sample

The following sample code shows the radius of the selected circle. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
If geom.ObjectType = ppcbObjectTypeCircle Then
MsgBox "Radius: " & geom.Radius
End If
Next geom
Next drw
End Sub
```

Circle.ShapeType

This property returns the shape type of the circle. The value of PPcbShapeOpen is not applicable.

Prototype

ShapeType as PPcbShapeType

Arguments

None

Comments

None

Sample

The following sample code shows the shape type of the selected circle. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
Select Case geom.ShapeType
Case ppcbShapeTypeOpen
s = "Open"
Case ppcbShapeTypeHollow
s = "Hollow"
Case ppcbShapeTypeFilled
s = "Filled"
Case ppcbShapeTypeVoid
s = "Void"
End Select
MsgBox "Shape type: " & s & " shape"
Next geom
Next drw
End Sub
```

Component.Application

This property returns the application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an automation object. All Automation server applications have an application object and all automation objects have an application property. This property is usually used in automation client applications that handle large volumes of objects from different sources, such as different automation server applications, to quickly identify the application to which the object belongs.

Component.Attributes

This property returns the collection of all attributes of the component.

Prototype

Attributes As Attributes

Attributes(*name* As String) As Attribute

Argument

name	Name of an existing component attribute.

Comments

When an existing attribute *name* is passed to this property, it returns that component Attribute object. Otherwise, it returns the collection of all component attributes in an Attributes collection object.

Sample

The following sample code retrieves the number of attributes on component U1, assuming it exists in the open design, using the Attributes.Count property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set attrs = ActiveDocument.Components("U1").Attributes
MsgBox "There are " & attrs.Count & " attribute(s) in component U1."
End Sub
```

Component.CBPs

This property returns the CBP collection of the die.

Restriction: This information applies to only the BGA toolkit.

Prototype

CBPs As Objects

Arguments

None

Comments

This property generates an exception if Component.IsDiePart is False.

Sample

The following sample code iterates through all CBPs in each die in the open design and prints the names of the CBPs. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aCBP In comp.CBPs
Debug.Print aCBP.Name
Next aCBP
End If
Next
End Sub
```

Component.CenterX

This property returns the x-coordinate of the center of the component.

Prototype

CenterX (*Unit* as PPcbUnit) as Double

Argument

unit [Optional] Unit in which the CenterX value is returned. This optional argument is ppcbUnitCurrent by default.

Comments

None

Sample

The following sample code displays coordinates of the center of a component. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeComponent,,True)
For Each comp In selected
Dim msg as String
Msg = "Component center: (" & comp.CenterX & ", " & comp.CenterY & ")"
MsgBox msg
Exit For
Next comp
End Sub
```

Component.CenterY

This property returns the y-coordinate of the center of the component.

Prototype

CenterY (*Unit* as PPcbUnit) as Double

Argument

unit [Optional] Unit in which the CenterY value is returned. This optional argument is ppcbUnitCurrent by default.

Comments

None

Sample

The following sample code displays coordinates of the center of the selected component. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeComponent,,True)
For Each comp In selected
Dim msg as String
Msg = "Component center: (" & comp.CenterX & ", " & comp.CenterY & ")"
MsgBox msg
Exit For
Next comp
End Sub
```

Component.Decal

This property sets or returns the decal of the component.

Prototype

Decal As String

Arguments

None

Comments

This property can only set a decal that is in the list of component-compatible decals, which is returned by the Component.DecalCompatibleList property.

This property generates an exception if the decal to set is invalid, or if the change cannot occur because of violations to the current DRC setting, the Nudge setting, locked test point status, or existing rules.

Sample

The following sample code retrieves the current decal of component C1, assuming it exists in the open design, along with the list of compatible decals to which it can be changed. See "Running Code Samples" for more information on running this sample.

```
Sub Main
decalList = ActiveDocument.Components("C1").DecalCompatibleList
decalListString = ""
For index = LBound(decalList) To UBound(decalList)
If index = 1 Then
decalListString = decalList(index)
Else
decalListString = decalListString & " or " & decalList(index)
End If
Next index
MsgBox "C1 decal is " & ActiveDocument.Components("C1").Decal & " and can
be changed to : " & decalListString
End Sub
```

See Also

Component.DecalCompatibleList

Component.DecalAttributes

This property returns the collection of all attributes assigned to the component decal.

Prototype

DecalAttributes As Attributes

DecalAttributes(*name* As String) As Attribute

Argument

name Name of an existing decal attribute.

Comments

When an existing attribute *name* is passed to this property, it returns that decal Attribute object. Otherwise, it returns the collection of all decal attributes in an Attributes collection object.

Sample

The following sample code retrieves the number of attributes assigned to the decal for component U1, assuming it exists in the open design, using the Attributes.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set attrs = ActiveDocument.Components("U1").DecalAttributes
MsgBox "There are " & attrs.Count & " attribute(s) in decal " &
ActiveDocument.Components("U1").Decal
End Sub
```

Component.DecalCompatibleList

This property returns the list of compatible decals of the component.

Prototype

DecalCompatibleList As Variant

Arguments

None

Comments

Use this property to identify all compatible decals for the component before changing the component decal using the Component.Decal property.

Sample

The following sample code retrieves the current decal for component C1, assuming it exists in the open design, along with the list of compatible decals to which it can be changed. See “Running Code Samples” for more information on running this sample.

```
Sub Main
decalList = ActiveDocument.Components("C1").DecalCompatibleList
decalListString = ""
For index = LBound(decalList) To UBound(decalList)
If index = 1 Then
decalListString = decalList(index)
Else
decalListString = decalListString & " or " & decalList(index)
End If
Next index
MsgBox "C1 decal is " & ActiveDocument.Components("C1").Decal & " and can
be changed to : " & decalListString
End Sub
```

See Also

Component.Decal

Component.DieHeight

This property returns the height of the die.

Restriction: This information applies to only the BGA toolkit.

Prototype

DieHeight([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the die height value is returned. This optional argument is ppcbUnitCurrent by default.

Comments

This property generates an exception if Component.IsDiePart is False.

Sample

The following sample code retrieves the height of each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
MsgBox "DieHeight (" & comp.Name & "): " & Format (comp.DieHeight,
"#.###")
End If
Next
End Sub
```

Component.DieLength

This property returns the length of the die.

Restriction: This information applies to only the BGA toolkit.

Prototype

DieLength([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the die length value is returned. This optional argument is ppcbUnitCurrent by default.

Comments

This property generates an exception if Component.IsDiePart is False.

Sample

The following sample code retrieves the length of each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
MsgBox "DieLength (" & comp.Name & "): " & Format (comp.DieLength,
"#.###")
End If
Next
End Sub
```

Component.DieWidth

This property returns the width of the die.

Restriction: This information applies to only the BGA toolkit.

Prototype

DieWidth([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the die width value is returned. This optional argument is ppcbUnitCurrent by default.

Comments

This property generates an exception if Component.IsDiePart is False.

Sample

The following sample code retrieves the width of each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
MsgBox "DieWidth (" & comp.Name & "): " & Format (comp.DieWidth, "#.###")
End If
Next
End Sub
```

Component.Glued

This property sets or returns whether the component is glued.

Prototype

Glued As Boolean

Arguments

None

Comments

None

Sample

The following sample code glues all components in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each nextComp In ActiveDocument.Components
nextComp.Glued = True
Next nextComp
End Sub
```

Component.Installed

This property sets or returns whether the component is installed in the current assembly option.

Prototype

Installed As Boolean

Arguments

None

Comments

This property is useful only when the parent of the component is an Assembly Option. If the parent of the component is not an Assembly Option, this property always returns True and cannot be set to a different value.

Sample

The following sample code creates a new Assembly Option named EmptyAssOpt in the open design, and then uninstalls all components in that newly created Assembly Option. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set newAssOpt = ActiveDocument.AssemblyOptions.Add("EmptyAssOpt")
For Each nextComp In newAssOpt.Components
nextComp.Installed = False
Next nextComp
End Sub
```

See Also

Component.Substituted

Component.IsDiePart

This property returns whether the component is a die.

Prototype

IsDiePart As Boolean

Arguments

None

Comments

In PADS Layout, this property always returns FALSE.

Sample

The following sample code retrieves the name of each die presented in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
MsgBox comp.Name
End If
Next
End Sub
```

Component.IsSMD

This property returns whether the component is SMD.

Prototype

IsSMD As Boolean

Arguments

None

Comments

This function returns whether this component is 100% SMD, meaning that all its pins are SMD pins.

Sample

The following sample code retrieves the number of SMD components in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
nbSMDComponents = 0
For Each nextComp In ActiveDocument.Components
If nextComp.IsSMD = True Then nbSMDComponents = nbSMDComponents + 1
Next nextComp
MsgBox "There are " & nbSMDComponents & " SMD components, out of " &
ActiveDocument.Components.Count
End Sub
```

Component.Labels

This property returns a collection of all labels associated with the component.

Prototype

Labels as Collection

Labels (*Name* as String) as Label

Argument

name Name of the label to retrieve (optional).

Comments

None

Sample

The following sample code displays a message box showing the name of the first label in the selected component. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(pcbObjectTypeComponent,,True)
For Each comp In selected
For Each label In comp.Labels
MsgBox "The first component label is " & label.Text
Exit For
Next label
Exit For
Next comp
End Sub
```

Component.Layer

This property sets or returns the mounting layer of the component.

Prototype

Layer As Long

Arguments

None

Comments

This property can only be set to a component layer(PPcbLayerType). Use the Document.LayerType property to identify all valid component layers in the open design.

Use the Document.LayerName property to identify the layer name that matches the returned layer number.

This property generates an exception if the layer to set is invalid or is not a component layer.

Sample

The following sample code mounts component U1, assuming it exists in the open design, on the opposite component layer. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set compU1 = ActiveDocument.Components("U1")
For index=1 To Document.LayerCount
If Document.LayerType(index) = ppcbLayerComponent And index <>
compU1.layer Then
compU1.layer = index
Exit For
End If
Next index
End Sub
```

See Also

Document.LayerCount, Document.LayerName, Document.LayerType

Component.Name

This property returns the name of the component.

Prototype

Name As String

Arguments

None

Comments

For example, this property returns the string “U1” for component U1.

This property is the default property for the Component object.

Sample

The following sample code lists all components in the open design and places that list in a custom dialog box. When a component is selected in the list box, the sample selects that component in PADS Layout.

This sample uses the UserDialog Editor in the Sax Basic Engine in PADS Layout. See “Running Code Samples” for more information on running this sample.

See also: Sax Basic Editor On Line Help (C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\sbe5_000.hlp)

```
Dim ListComps$(10000)
Sub Main
index = 0
For Each nextComp In ActiveDocument.Components
ListComps$(index) = nextComp.Name
index = index + 1
Next nextComp
' This piece of code is automatically generated by the Basic Dialog Editor
in PADS Layout.
Begin Dialog UserDialog 180,238,"Components",.CallbackFunc '
%GRID:10,7,1,1
ListBox 10,7,160,203,ListComps(),.ListBox1
OKButton 10,210,160,21
End Dialog
Dim dlg As UserDialog
Dialog dlg
End Sub
' The following function is automatically called by the system when
something has happened in the dialog; it is used to easily process user
actions.
Function CallbackFunc%(DlgItem$, Action%, SuppValue%)
Select Case Action%
Case 2 ' Value changing or button pressed
If DlgItem$ = "ListBox1" Then
```

```
ActiveDocument.SelectObjects(ppcbObjectTypeAll, , False)
ActiveDocument.SelectObjects(ppcbObjectTypeComponent,
ListComps(SuppValue%))
End If
End Select
End Function
```

Component.ObjectType

This property returns the type of the object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property returns ppcbObjectTypeComponent.

All PADS Layout database objects in the PADS Layout Automation server implement this property to compensate for the lack of a Basic equivalent for the Visual C++ QueryInterface function.

This property is generally used:

- To identify the kind of PADS Layout database objects in a heterogeneous Objects collection.
- When implementing a generic routine that depends on the type of the PADS Layout database object passed as argument. For example:

```
Sub DoSomething(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeComponent
' Do something specific to component objects
Case ppcbObjectTypeNet
' Do something specific to net objects
Case ppcbObjectTypePin
' Do something specific to pin objects
Case ppcbObjectTypeVia
' Do something specific to via objects
Case ppcbObjectTypeConnection
' Do something specific to connection objects
Case ppcbObjectTypeRouteSegment
' Do something specific to route segment objects
Case ppcbObjectTypeJumper
' Do something specific to jumper objects
Case Else
MsgBox "Not a PADS Layout database object"
End Select
End Sub
```

Component.Orientation

This property sets or returns the orientation of the component.

Prototype

Orientation As Double

Arguments

None

Comments

The proper completion of this method depends on the items listed below. To force a component rotation, you must first disable the following:

- Glued status, which is set using the Component.Glued property.
- DRC mode, which is set using the Document.Preference property.
- Nudge mode, which is set using the Document.Preference property.
- ModifyUnionMember mode, which is set using the Document.Preference property.

This method generates an exception if a PADS Layout security failure occurs during processing.

Sample

The following sample code rotates U1 to 45 degrees, assuming U1 exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.Components("U1").Orientation = 45
End Sub
```

See Also

Document.PositionsChange

Component.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

None

Component.PartType

This property sets or returns the part type of the component.

Prototype

PartType As String

Arguments

None

Comments

This property can be set to a new value only when the parent of the component is an Assembly Option. If the parent is an Assembly Option, this property substitutes the component in the current Assembly Option with a new part type. The new part type must have a decal compatible with the default decal.

This property generates an exception if the parent of the component is not an Assembly Option or if the part type is invalid.

Sample

The following sample code retrieves the part type of component U1, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "The part type of U1 is " &
ActiveDocument.Components("U1").PartType
End Sub
```

See Also

Component.Substituted, Component.PartTypeObject

Component.PartTypeAttributes

This property returns the collection of all attributes for the component part type.

Prototype

PartTypeAttributes As Attributes

PartTypeAttributes(*name* As String) As Attribute

Argument

name Name of an existing part type attribute.

Comments

When an existing attribute *name* is passed to this property, it returns that part type Attribute object. If the attribute *name* does not exist, this property returns the collection of all part type attributes in an Attributes collection object.

Sample

The following sample code retrieves the number of attributes assigned to the part type for component U1, assuming it exists in the open design, using the Attributes.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set attrs = ActiveDocument.Components("U1").PartTypeAttributes
MsgBox "There are " & attrs.Count & " attribute(s) in decal " &
ActiveDocument.Components("U1").PartType
End Sub
```

Component.PartTypeECORegistered

This property sets or returns the part type ECO registration status.

Prototype

PartTypeECORegistered As Boolean

Arguments

None

Comments

Beginning with PowerPCB v3.5 this property becomes obsolete. Use PartType.ECORegistered instead.

See Also

PartType.ECORegistered

Component.PartTypeLogic

This property returns the logic type of the part type for the component.

Prototype

PartTypeLogic As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the logic type of the part type for component U1, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "The Logic type of U1 is " &
ActiveDocument.Components("U1").PartTypeLogic
End Sub
```

Component.PartTypeObject

This property returns the part type object of this component.

Prototype

PartTypeObject As PartType

Arguments

None

Comments

None

Sample

The following sample code retrieves the part type object for component U1, assuming it exists in the open schematic. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "The part type of U1 is " &
ActiveDocument.Components("U1").PartTypeObject.Name
End Sub
```

See Also

Component.PartType

Component.Pins

This property returns the collection of all pins in the component.

Prototype

Pins As Objects

Pins(*name* As String) As Pin

Argument

name Name of an existing pin.

Comments

When an existing pin *name* is passed to this property, it returns that Pin object. If the pin *name* does not exist, this property returns the collection of all pins of the component in an Objects collection object.

Sample

The following sample code retrieves the number of pins in component U1, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "U1 has " & ActiveDocument.Components("U1").Pins.Count & " pins."
End Sub
```

Component.Placed

This property returns whether the component is placed.

Prototype

Placed As Boolean

Arguments

None

Comments

A component is considered *placed* if all its pins are inside the board outline, but outside a board cut out.

Sample

The following sample code determines whether component U1 is placed or not, assuming U1 exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Select Case ActiveDocument.Components("U1").Placed
Case True
MsgBox "Component U1 is placed"
Case False
MsgBox "Component U1 is not placed"
End Select
End Sub
```

Component.PositionX

This property returns the x-coordinate of the component.

Prototype

PositionX([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the x-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the location of component U1, assuming it exists in the open design, in current design units. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set compU1 = ActiveDocument.Components("U1")
MsgBox "U1 position is = (" & compU1.PositionX & ", " & compU1.PositionY &
")"
End Sub
```

See Also

Component.PositionY, Component.Move

Component.PositionY

This property returns the y-coordinate of the component.

Prototype

PositionY([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the y-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the location of component U1, assuming it exists in the open design, in current design units. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set compU1 = ActiveDocument.Components("U1")
MsgBox "U1 position is = (" & compU1.PositionX & ", " & compU1.PositionY &
")"
End Sub
```

See Also

Component.PositionX, Component.Move

Component.SBPs

This property returns the die's SBP collection.

Restriction: This information applies to only the BGA toolkit.

Prototype

SBPs As Objects

Arguments

None

Comments

This property generates an exception if Component.IsDiePart is False.

Sample

The following sample code iterates through SBPs for each die component in the open design and prints the names of the SBPs. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP In comp.SBPs
Debug.Print aSBP.Name
Next
End If
Next
End Sub
```

Component.Selected

This property sets or returns whether the component is selected.

Prototype

Selected As Boolean

Arguments

None

Comments

You can also select a PADS Layout database object using the Document.SelectObjects and Objects.Select methods.

Sample

The following sample code selects component U1 only, assuming it exists in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
ActiveDocument.SelectObjects(,,False)
ActiveDocument.Components("U1").Selected = True
End Sub
```

See Also

Document.SelectionChange Event

Component.Substituted

This property returns whether the component is substituted in the current Assembly Option.

Prototype

Substituted As String

Arguments

None

Comments

This property is useful only when the parent of the component is an Assembly Option. If the parent of the component is not an Assembly Option, this property always returns False.

To substitute a component in an Assembly Option, use the Component.PartType property.

Sample

The following sample code retrieves the number of substituted components in the MyAssOpt Assembly Option, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set myAssOpt = ActiveDocument.AssemblyOptions("MyAssOpt")
nbSubstComp = 0
For Each nextComp In myAssOpt.Components
 If nextComp.Substituted = True Then nbSubstComp = nbSubstComp + 1
Next nextComp
MsgBox "There are " & nbSubstComp & " substituted components in " &
myAssOpt.Name
End Sub
```

See Also

Component.PartType, Component.Installed

Component.WireBondRulesAngleMaximum

This property returns the bond wire maximum angle rule of the die, in degrees.

Restriction: This information applies to only the BGA toolkit.

Prototype

WireBondRulesAngleMaximum As Double

Arguments

None

Comments

This property generates an exception if Component.IsDiePart is False.

Sample

The following sample code retrieves the maximum angle rule for each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
MsgBox "MaxAngleRule: " & Format (comp.WireBondRulesAngleMaximum,
"#.###")
End If
Next
End Sub
```

Component.WireBondRulesClearanceWireToPad

This property returns the bond wire-to-pad clearance rule of the die.

Restriction: This information applies to only the BGA toolkit.

Prototype

WireBondRulesClearanceWireToPad([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the clearance value is returned.

Comments

This property generates an exception if Component.IsDiePart is False.

Sample

The following sample code retrieves the wire-to-pad clearance rule for each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
MsgBox "WireToPad: " & Format (comp.WireBondRulesClearanceWireToPad,
"#.###")
End If
Next
End Sub
```

Component.WireBondRulesClearanceWireToWire

This property returns the bond wire-to-wire clearance rule of the die.

Restriction: This information applies to only the BGA toolkit.

Prototype

WireBondRulesClearanceWireToWire([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the clearance value is returned.

Comments

This property generates an exception if Component.IsDiePart is False.

Sample

The following sample code retrieves the wire-to-wire clearance rule for each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
'For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
MsgBox "WireToWire: " & Format
(comp.WireBondRulesClearanceWireToWire,"#.###")
End If
Next
End Sub
```

Component.WireBondRulesLengthMaximum

This property returns the bond wire maximum length rule of the die.

Restriction: This information applies to only the BGA toolkit.

Prototype

WireBondRulesLengthMaximum([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the length value is returned.

Comments

This property generates an exception if Component.IsDiePart is False.

Sample

The following sample code retrieves the maximum length rule for each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
MsgBox "MaxLengthRule: " & Format (comp.WireBondRulesLengthMaximum,
"#.###")
End If
Next
End Sub
```

Component.WireBondRulesLengthMinimum

This property returns the bond wire minimum length rule of the die.

Restriction: This information applies to only the BGA toolkit.

Prototype

WireBondRulesLengthMinimum([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the length value is returned.

Comments

This property generates an exception if Component.IsDiePart is False.

Sample

The following sample code retrieves the minimum length rule for each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
MsgBox "MinLengthRule: " & Format (comp.WireBondRulesLengthMinimum,
"#.###")
End If
Next
End Sub
```

Component.Wirebonds

This property returns the bond wire collection of the die.

Restriction: This information applies to only the BGA toolkit.

Prototype

Wirebonds As Objects

Arguments

None

Comments

None

Sample

The following sample code iterates through all bond wires in each die component in the open design and prints the bond wire names. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each nWB In comp.Wirebonds
Debug.Print nWB.Name
Next
End If
Next
End Sub
```

Connection.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an Automation object. All Automation server applications have an Application object and all Automation objects have an Application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

Connection.Length

This property returns the length of the connection.

Prototype

Length([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the length value is returned.

Comments

None

Sample

The following sample code retrieves the length of the first connection found in the open design, assuming at least one connection exists. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set firstConn = ActiveDocument.Connections(1)
MsgBox "Connection " & firstConn.Name & " has a length of " &
firstConn.Net.Length
End Sub
```

Connection.Name

This property returns the name of the connection.

Prototype

Name As String

Arguments

None

Comments

This property is the default property for the Connection object.

Sample

The following sample code lists all connections in the open design by name and then places that list in a custom dialog box. When a connection is selected in the list box, the sample selects that connection.

This sample uses the UserDialog Editor in the Sax Basic Engine. See "Running Code Samples" for more information on running this sample.

See also: Sax Basic Editor On Line Help (C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\sbe5_000.hlp)

```
Dim ListConns$(10000)
Sub Main
index = 0
For Each nextConn In ActiveDocument.Connections
ListConns$(index) = nextConn.Name
index = index + 1
Next nextConn
' This piece of code is automatically generated by the Basic Dialog Editor
in PADS Layout.
Begin Dialog UserDialog 180,238,"Connections",.CallbackFunc '
%GRID:10,7,1,1
ListBox 10,7,160,203,ListConns(),.ListBox1
OKButton 10,210,160,21
End Dialog
Dim dlg As UserDialog
Dialog dlg
End Sub
' The following function is automatically called by the system when
something has happened
' in the dialog; it is used to easily process user actions.
Function CallbackFunc%(DlgItem$, Action%, SuppValue%)
Select Case Action%
Case 2 ' Value changing or button pressed
If DlgItem$ = "ListBox1" Then
ActiveDocument.SelectObjects(ppcbObjectTypeAll, , False)
```

```
ActiveDocument.SelectObjects(ppcbObjectTypeConnection,
ListConns(SuppValue%))
End If
End Select
End Function
```

Connection.Net

This property returns the net connected to the connection.

Prototype

Net As Net

Arguments

None

Comments

None

Sample

The following sample code retrieves the net connected to the first connection found in the open design, assuming at least one connection exists. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set firstConn = ActiveDocument.Connections(1)
MsgBox "Connection " & firstConn.Name & " is connected to net " &
firstConn.Net.Name
End Sub
```

Connection.ObjectType

This property returns the type of the object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property returns ppcbObjectTypeConnection.

All database objects in the PADS Layout Automation server implement this property to compensate for the lack of a Visual Basic equivalent for the Visual C++ QueryInterface function.

This property is generally used:

- To identify the kind of database objects in a heterogeneous Objects collection.
- When implementing a generic routine that depends on the type of the database object passed as argument. For example:

```
Sub DoSomething(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeComponent
' Do something specific to component objects
Case ppcbObjectTypeNet
' Do something specific to net objects
Case ppcbObjectTypePin
' Do something specific to pin objects
Case ppcbObjectTypeVia
' Do something specific to via objects
Case ppcbObjectTypeConnection
' Do something specific to connection objects
Case ppcbObjectTypeRouteSegment
' Do something specific to route segment objects
Case ppcbObjectTypeJumper
' Do something specific to jumper objects
Case Else
MsgBox "Not a PADS Layout database object"
End Select
End Sub
```

Connection.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

None

Connection.Pins

This property returns the collection of all pins in the connection.

Prototype

Pins As Objects

Pins(*name* As String) As Pin

Argument

name	Name of an existing pin.

Comments

When an existing pin name is passed to this property, it returns that Pin object. If the pin name does not exist, this property returns the collection of all pins of the connection in an Objects collection object.

Sample

The following sample code retrieves the number of pins in the first connection found in the open design, assuming at least one connection exists. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set firstConn = ActiveDocument.Connections(1)
MsgBox "Connection " & firstConn.Name & " connects " &
firstConn.Pins.Count & " pins."
End Sub
```

Connection.RouteSegments

This property returns the collection of all trace segments in the connection.

Prototype

RouteSegments As Objects

RouteSegments(*name* As String) As RouteSegment

Argument

name	Name of an existing trace segment.

Comments

When an existing trace segment *name* is passed to this property, it returns that RouteSegment object. If the trace segment *name* does not exist, this property returns the collection of all route segments in the connection in an Objects collection object.

Sample

The following sample code retrieves the number of trace segments in the first connection found in the open design, assuming at least one connection exists. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set firstConn = ActiveDocument.Connections(1)
MsgBox "Connection " & firstConn.Name & " includes " &
firstConn.RouteSegments.Count & " route segments."
End Sub
```

Connection.Selected

This property sets or returns whether the connection is selected.

Prototype

Selected As Boolean

Arguments

None

Comments

None

Sample

The following sample code selects the first connection found in the open design, assuming at least one connection exists. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.SelectObjects(,,False)
ActiveDocument.Connections(1).Selected = True
End Sub
```

See Also

Document.SelectionChange Event

Connection.Vias

This property returns the collection of all vias in the connection.

Prototype

Vias As Objects

Vias(*name* As String) As Via

Argument

name	Name of an existing via.

Comments

When an existing via *name* is passed to this property, it returns that Via object. If the via *name* does not exist, this property returns the collection of all vias in the collection in an Objects collection object.

Sample

The following sample code retrieves the number of vias in the first connection found in the open design, assuming at least one connection exists. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set firstConn = ActiveDocument.Connections(1)
MsgBox "Connection " & firstConn.Name & " includes " &
firstConn.Vias.Count & " vias."
End Sub
```

Decal.Application

This property returns the application object.

Prototype

Application as Application

Argument

None

Decal.Attributes

This property returns the Decals's collection of attributes.

Prototype

Attributes (*name* As String) As Attribute

Argument

name	**Name of an existing decal attribute.**

Decal.Components

This property returns the collection of Components objects of this decal.

Prototype

Components (*name* As String) As Component

Argument

name	Name of an existing component.

Decal.LibraryTimeStamp

This property returns the Decal's library timestamp.

Prototype

TimeStamp as Date

Argument

None

Decal.Name

This property returns the Decal's name.

Prototype

Name as String

Argument

None

Decal.ObjectType

This property returns the type of this database object: a decal.

Prototype

ObjectType as PPcbObjectType

Argument

None

Decal.Parent

This property returns the parent of the object.

Prototype

Parent as Document

Argument

None

Decal.Selected

This property sets or returns the Decal's selection status.

Prototype

Selected as Boolean

Argument

None

Decal.TimeStamp

This property returns the Decal's timestamp.

Prototype

TimeStamp as Date

Argument

None

Document.ActiveView

This property returns the active view.

Prototype

ActiveView As View

Arguments

None

Comments

The active view represents the main view of the open design.

Sample

The following sample code pans the current view to the origin of the open design using the View.Pan method. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.ActiveView.Pan(0,0)
End Sub
```

Document.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as a PADS Layout Automation object. All Automation server applications have an Application object and all Automation objects have an Application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

Document.AssemblyOptions

This property returns the collection of all assembly options.

Prototype

AssemblyOptions As AssemblyOptions

AssemblyOptions(*name* As String) As Document

Argument

name	Name of an existing assembly option.

Comments

When an existing assembly option *name* is passed to this property, it returns that assembly option object, packaged as a Document object. If the Assembly Option *name* does not exist, this property returns the collection of all assembly options existing in the open design, packaged as an AssemblyOptions collection object.

The Document.Name property of an assembly option document is in the following format: Assembly Option Name:PCB Design File Name, for example WithoutU1:PWRDEMOA.PCB.

Tip: Assembly options are always packaged as Document objects.

Sample 1

The following sample code retrieves the number of assembly options in the open design using the AssemblyOptions.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set assopts = ActiveDocument.AssemblyOptions
MsgBox "There are " & assopts.Count & " assembly option(s) in " &
ActiveDocument.Name
End Sub
```

Sample 2

The following sample code retrieves the number of uninstalled and substituted components in the assembly option named MyAssOpt, assuming it exists in the open design, using the Component.Installed and Component.Substituted properties. See "Running Code Samples" for more information on running this sample.

```
Sub Main
nbUninstalled = 0
nbSubstituted = 0
For Each nextComp In ActiveDocument.AssemblyOptions("MyAssOpt")
If nextComp.Installed = False Then nbUninstalled = nbUninstalled + 1
If nextComp.Substituted = True Then nbSubstituted = nbSubstituted + 1
Next nextComp
```

```
MsgBox "There are " & nbUninstalled & " uninstalled and " & nbSubstituted
& " substituted components in assembly option MyAssOpt."
End Sub
```

Document.AssociatedNets

This property returns the collection of all associated nets.

Prototype

```
AssociatedNets as Objects
AssociatedNets (name as String) as AssociatedNet
```

Arguments

Name Optional
Name of an existing associated net

Return Values

When an existing associated net name is passed to this property, it returns the associated net object with that name. If the associated net name is not specified, this property returns the collection of all associated nets in Objects.

Document.Attributes

This property returns the collection of all attributes of the document.

Prototype

Attributes As Attributes

Attributes(*name* As String) As Attribute

Argument

name Name of an existing document attribute.

Comments

When an existing attribute *name* is passed to this property, it returns that document Attribute object. If the attribute *name* does not exist, this property returns the collection of all document attributes in an Attributes collection object.

Sample 1

The following sample code retrieves the number of document attributes in the open design using the Attributes.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set attrs = ActiveDocument.Attributes
MsgBox "There are " & attrs.Count & " document attribute(s) in " &
ActiveDocument.Name
End Sub
```

Sample 2

The following sample code retrieves the value of the document attribute DEADLINE, assuming it exists in the open design, using the Attribute.Value property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "DEADLINE = " & ActiveDocument.Attributes("DEADLINE").Value
End Sub
```

Document.BoardOutlineSurface

This property returns the surface of the document board outline.

Prototype

BoardOutlineSurface([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the board outline surface value is returned.

Comments

The board outline surface is calculated disregarding board outline cut outs or holes.

Sample

The following sample code retrieves the board outline surface of the open design. See “Running Code Samples” for more information on running this sample.

```
Function UnitName(unit As Long) As String
Select Case unit
Case ppcbUnitMils
UnitName = "mils"
Case ppcbUnitInch
UnitName = "inches"
Case ppcbUnitMetric
UnitName = "mm"
Case Else
UnitName = "unknown"
End Select
End Function

Sub Main
MsgBox "The board outline surface is " &
ActiveDocument.BoardOutlineSurface & " square " &
UnitName(ActiveDocument.unit)
End Sub
```

Document.Components

This property returns the collection of all components.

Prototype

Components As Objects

Components(*name* As String) As Component

Argument

name Name of an existing component.

Comments

When an existing component *name* is passed to this property, it returns that Component object. If the component *name* does not exist, this property returns the collection of all components in an Objects collection object.

Sample 1

The following sample code retrieves the number of components in the open design using the Objects.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set comps = ActiveDocument.Components
MsgBox "There are " & comps.Count & " component(s) in " &
ActiveDocument.Name
End Sub
```

Sample 2

The following sample code retrieves the number of pins in component U1, assuming it exists in the open design, using the Component.Pins and Objects.Count properties. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set compU1 = ActiveDocument.Components("U1")
MsgBox "Component " & compU1.Name & " has " & compU1.Pins.Count & "
pin(s)."
End Sub
```

See Also

Document.GetObjects

Document.Connections

This property returns the collection of all connections.

Prototype

Connections As Objects

Connections(*name* As String) As Connection

Argument

name	Name of an existing connection.

Comments

When an existing connection name is passed to this property, it returns that Connection object. If the connection name does not exist, this property returns the collection of all connections in an Objects collection object.

Sample 1

The following sample code retrieves the number of connections in the open design using the Objects.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set conns = ActiveDocument.Connections
MsgBox "There are " & conns.Count & " connection(s) in " &
ActiveDocument.Name
End Sub
```

Sample 2

The following sample code retrieves the sum of the length (routed and unrouted) of all connections in the open design using the Connection.Length property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
totalLength = 0.0
For Each nextConn In ActiveDocument.Connections
totalLength = totalLength + nextConn.Length
Next nextConn
MsgBox "The sum of all connection length of " & ActiveDocument.Name & " is
" & totalLength
End Sub
```

See Also

Document.GetObjects

Document.Drawings

This property returns the collection of drawings in the document or a specific drawing.

Prototype

Drawings as Collection

Drawings (*Name* as String) as Drawing

Argument

name Name of the drawing to retrieve (optional).

Comments

None

Sample

See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "Number of Drawings: " & ActiveDocument.Drawings.Count
End Sub
```

Document.ElectricalLayerCount

This property returns the number of electrical layers in the design.

Prototype

ElectricalLayerCount as Boolean

Arguments

None

Comments

None

Sample

This sample shows the number of electrical layers in the design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "El. layer count: " & ActiveDocument.ElectricalLayerCount
End Sub
```

Document.Errors

This property returns the number of electrical layers in the design.

Prototype

Errors as Objects

Errors (errorNumber as Integer) as Error

ArgumentsArguments

errorNumber error number

Comments

When the error number is passed to this property, it returns that Error Object. If the number is not specified, this property returns the collection of all errors in an Objects collection object.

Document.FullName

This property returns the file name of the document, including its path.

Prototype

FullName As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the name and location of the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "Hi, you are using " & ActiveDocument.FullName & " located in " &
ActiveDocument.Path
End Sub
```

See Also

Document.Name, Document.Path

Document.GridX

This property sets or returns the X grid of the document.

Prototype

GridX([*type* As PPcbGridType = ppcbGridDesign], [*unit* As PPcbUnit]) As Double

Arguments

type	[Optional] Type of grid to set or return.
unit	[Optional] Unit in which the grid value is set or returned.

Comments

None

Sample

The following sample code sets the display grid of the open design to the same value as the routing grid. See "Running Code Samples" for more information on running this sample.

```
Sub Main
curGridX = ActiveDocument.GridX
curGridY = ActiveDocument.GridY
ActiveDocument.GridX(ppcbGridDisplay) = curGridX
ActiveDocument.GridY(ppcbGridDisplay) = curGridY
End Sub
```

See Also

Document.GridY

Document.GridY

This property sets or returns the Y grid of the document.

Prototype

GridY([*type* As PPcbGridType = ppcbGridDesign], [*unit* As PPcbUnit]) As Double

Arguments

type	[Optional] Type of grid to set or return.
unit	[Optional] Unit in which the grid value is set or returned.

Comments

None

Sample

The following sample code sets the display grid of the open design to the same value as the routing grid. See "Running Code Samples" for more information on running this sample.

```
Sub Main
curGridX = ActiveDocument.GridX
curGridY = ActiveDocument.GridY
ActiveDocument.GridX(ppcbGridDisplay) = curGridX
ActiveDocument.GridY(ppcbGridDisplay) = curGridY
End Sub
```

See Also

Document.GridX

Document.Jumpers

This property returns the collection of all jumpers.

Prototype

Jumpers As Objects

Jumpers(*name* As String) As Jumper

Argument

name Name of an existing jumper.

Comments

When an existing jumper *name* is passed to this property, it returns that Jumper object. If the jumper *name* does not exist, this property returns the collection of all jumpers in an Objects collection object.

Sample 1

The following sample code retrieves the number of jumpers in the open design using the Objects.Count property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set jmps = ActiveDocument.Jumpers
MsgBox "There are " & jmps.Count & " jumpers(s) in " &
ActiveDocument.Name
End Sub
```

Sample 2

The following sample code retrieves the sum of the length of all jumpers in the open design using the Jumper.Length property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
totalLength = 0.0
For Each nextJmp In ActiveDocument.Jumpers
totalLength = totalLength + nextJmp.Length
Next nextJmp
MsgBox "The sum of all jumper length of " & ActiveDocument.Name & " is " &
totalLength
End Sub
```

See Also

Document.GetObjects

Document.LayerCount

This property returns the total number of layers in the document.

Prototype

LayerCount As Long

Arguments

None

Comments

Layer 0 is not considered a valid layer in the Automation server.

Sample

The following sample code retrieves the total number of layers in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox ActiveDocument.LayerCount & " layers in this PCB design."
End Sub
```

See Also

Document.LayerType

Document.LayerEnabled

This property returns the enabled status of a layer.

Prototype

LayerEnabled(*layer* as Long) as Boolean

Argument

name	Number of layer (required).

Comments

None

Sample

This sample shows the enabled status of layer 17 in the design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "Is layer 17 enabled? " & ActiveDocument.LayerEnabled(17)
End Sub
```

Document.LayerName

This property returns the layer name of a layer number.

Prototype

LayerName(*layer* As Long) As String

Argument

layer	Layer

Comments

Layer 0 is not considered a valid layer in the Automation server.

This property generates an exception if the layer argument is not a valid layer number.

Sample

The following sample code retrieves the name of layer 2 in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
MsgBox ActiveDocument.LayerName(2) & " is layer 2."
End Sub
```

Document.Layers

This property returns the collection of all layers in the design.

Prototype

Layers as Objects

Layers(*layerNumber* as Integer) as Layer

Layers(*layerName* as String) as Layer

Argument

layerNumber	Number of the layer
layerName	Name of the layer

Return Values

When a layer number/name is passed to this property, it returns that Layer Object. If the number/name is not specified, this property returns the collection of all layers in an Objects collection object.

Sample

```
doc = Application.ActiveDocument

msg = ""
For Each layer In doc.Layers

  msg = msg & layer.Number & ", " & layer.Name & ", "
          & layer.Type & ", " & layer.PlaneType & ", "
          & layer.RoutingDirection & ", " & layer.Visible & ", "
          & layer.Enabled & ", "
          & layer.GetColor(ppcbLayerColorPad) & ", "
          & layer.CopperThickness & ", "
          & layer.GetDielectricThickness(ppcbDielectricLayerAbove)
          & ", "
          & layer.GetDielectricConstant(ppcbDielectricLayerBelow)
  msg = msg & chr(13)

Next layer

MsgBox msg

layer = doc.Layers(1)
curr_pad_color = layer.GetColor(ppcbLayerColorPad)
layer.SetColor(ppcbLayerColorPad, 10)

MsgBox "Press any key"

layer.Visible = false

MsgBox "Press any key"
```

```
layer.Visible = true

MsgBox "Press any key"

layer.SetColor(ppcbLayerColorPad, curr_pad_color)
```

Document.LayerType

This property returns the type of a given layer number.

Prototype

LayerType(*layer* As Long) As PPcbLayerType

Argument

layer Layer

Comments

Layer 0 is not considered a valid layer in the Automation server.

This property generates an exception if the *layer* argument is not a valid layer number.

Sample

The following sample code retrieves the type of layer 2 in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Dim strType As String
Select Case ActiveDocument.LayerType(2)
Case ppcbLayerComponent
strType = "Component layer"
Case ppcbLayerRouting
strType = "Routing layer"
Case ppcbLayerDrill
strType = "Drill layer"
Case ppcbLayerSilkscreen
strType = "Silkscreen layer"
Case ppcbLayerPasteMask
strType = "Paste mask layer"
Case ppcbLayerSolderMask
strType = "Solder mask layer"
Case ppcbLayerAssembly
strType = "Assembly layer"
Case ppcbLayerGeneral
strType = "General layer"
Case Else
strType = "Unknown layer type"
End Select
MsgBox "Layer 2 is a " & strType
End Sub
```

See Also

Document.LayerCount

Document.MaxRealValue

This property Returns the maximum real value in the system.

Prototype

MaxRealValue as double

Arguments

None

Document.MinRealValue

This property Returns the minimum real value in the system.

Prototype

MinRealValue as double

Arguments

None

Document.Name

This property returns the name of the document. For example, if the current design file is \PADS Projects\Samples\pwrdemoa.pcb, this function returns the string “pwrdemoa.pcb”.

Prototype

Name As String

Arguments

None

Comments

This property is the default property for the Document object.

Sample

The following sample code retrieves the name and location of the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
MsgBox "Hi, you are using " & ActiveDocument.Name & " located in " &
ActiveDocument.Path
End Sub
```

See Also

Document.FullName, Document.Path

Document.NetClasses

This property returns the collection of net classes in the document or in a specific net class.

Prototype

NetClasses as Collection

NetClasses (*Name* as String) as NetClass

Argument

name Name of the net class to retrieve.

Comments

None

Sample

This sample shows the total number of net classes in the design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "Number of Net Classes: " & ActiveDocument.NetClasses.Count
End Sub
```

Document.Nets

This property returns the collection of all nets.

Prototype

Nets As Objects

Nets(*name* As String) As Net

Argument

name	Name of an existing net.

Comments

When an existing net *name* is passed to this property, it returns that Net object. If the net *name* does not exist, this property returns the collection of all nets in an Objects collection object.

Sample 1

The following sample code retrieves the number of nets in the open design using the Objects.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set nets = ActiveDocument.Nets
MsgBox "There are " & nets.Count & " net(s) in " & ActiveDocument.Name
End Sub
```

Sample 2

The following sample code retrieves the number of pins connected to net VCC, assuming it exists in the open design, using the Net.Pins and Objects.Count properties. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set netVCC = ActiveDocument.Nets("VCC")
MsgBox "Net " & netVCC.Name & " connects " & netVCC.Pins.Count & "
pin(s)."
End Sub
```

See Also

Document.GetObjects

Document.ObjectType

This property returns the type of this object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Document.OriginX

This property returns the x origin of the document.

Prototype

OriginX([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the x origin is returned.

Comments

If the unit passed as an argument is different from the server database unit type, the return value is 0.0. The origin of the document in mils, millimeters, or inches is 0.0 by definition.

Sample

The following sample code retrieves the origin of the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set doc = ActiveDocument
MsgBox "Origin = (" & doc.OriginX(ppcbUnitDatabase) & ", " &
doc.OriginY(ppcbUnitDatabase) & ")"
End Sub
```

See Also

Document.OriginY

Document.OriginY

This property returns the y origin of the document.

Prototype

OriginY([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the y origin is returned.

Comments

If the unit passed as an argument is different from the server database unit type, the return value is 0.0. The origin of the document in mils, millimeters, or inches is 0.0 by definition.

Sample

The following sample code retrieves the origin of the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set doc = ActiveDocument
MsgBox "Origin = (" & doc.OriginX(ppcbUnitDatabase) & ", " &
doc.OriginY(ppcbUnitDatabase) & ")"
End Sub
```

See Also

Document.OriginX

Document.Parent

This property returns the parent of the object.

Prototype

Parent As Application

Arguments

None

Comments

None

Document.PartTypes

This property returns the collection of all part types.

Prototype

PartTypes As Objects

PartTypes(*name* As String) As PartType

Argument

name	Name of an existing part type.

Comments

When an existing part type name is passed to this property, it returns that PartType object. Otherwise, this property returns the collection of all part types in an Objects collection object.

Sample

The following sample code retrieves the number of part types in the open schematic using the Objects.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set pkgs = ActiveDocument.PartTypes
MsgBox "There are " & pkgs.Count & " part type(s) in " &
ActiveDocument.Name
End Sub
```

See Also

Document.GetObjects

Document.Path

This property returns the path of the document.

Prototype

Path As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the name and path of the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "Hi, you are using " & ActiveDocument.Name & " located in " &
ActiveDocument.Path
End Sub
```

See Also

Document.FullName, Document.Name

Document.Pins

This property returns the collection of all pins.

Prototype

Pins As Objects

Pins(*name* As String) As Pin

Argument

name	Name of an existing pin.

Comments

When an existing pin *name* is passed to this property, it returns that Pin object. If the pin *name* does not exist, this property returns the collection of all pins in an Objects collection object.

Sample 1

The following sample code retrieves the number of pins in the open design using the Objects.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set pins = ActiveDocument.Pins
MsgBox "There are " & pins.Count & " pin(s) in " & ActiveDocument.Name
End Sub
```

Sample 2

The following sample code retrieves the drill size of pin U1.1, assuming it exists in the open design, using the Pin.DrillSize property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set pinU1_1= ActiveDocument.Pins("U1.1")
MsgBox "Pin " & pinU1_1.Name & "'s drill size is " & pinU1_1.DrillSize
End Sub
```

See Also

Document.GetObjects

Document.Preference

This property sets or returns a document preference.

Prototype

Preference(*name* As String) AsVariant

Argument

name Name of the preference.

Comments

The following are possible *name* argument values:

DRC	Sets or returns the DRC mode of the open design. The DRC mode values are of type PPcbDRCMode.
Nudge	Sets or returns the Nudge mode of the open design. The Nudge mode values are of type PPcbNudgeMode.
ModifyUnionMember	Sets or returns the ability (True/False) to move an individual component when that component belongs to a union.

This property generates an exception if the name argument is not a valid document preference.

Sample

The following sample code demonstrates how to use this property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
' Set the DRC mode to "Prevent"
ActiveDocument.Preference("DRC") = ppcbDRCPrevent
' Set the Nudge mode to "Automatic"
ActiveDocument.Preference("Nudge") = ppcbNudgeAuto
End Sub
```

Document.RouteSegments

This property returns the collection of all trace segments.

Prototype

RouteSegments As Objects

RouteSegments(*name* As String) As RouteSegment

Argument

name	Name of an existing trace segment.

When an existing trace segment *name* is passed to this property, it returns that RouteSegment or Net. If the trace segment *name* does not exist, this property returns the collection of all trace segments in an Objects collection object.

Comments

None

Sample

The following sample code retrieves the number of trace segments in the open design using the Objects.Count property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set rtesegs = ActiveDocument.RouteSegments
MsgBox "There are " & rtesegs.Count & " route segment(s) in " &
ActiveDocument.Name
End Sub
```

See Also

Document.GetObjects

Document.Saved

This property sets or returns whether the document is saved.

Prototype

Saved As Boolean

Arguments

None

Comments

This property is usually used before opening a new design, using the Application.OpenDocument Method method, to ensure that the message Save old file before reloading? does not appear.

Sample

The following sample code sets the saved status of the open design to True and then opens PWRDEMOA.PCB, assuming that it exists in the folder specified by the Application.DefaultFilePath property. It then retrieves the name of the opened design file. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.Saved = True
OpenDocument(DefaultFilePath & "\PWRDEMOA.PCB")
MsgBox ActiveDocument.FullName & " has just been opened."
End Sub
```

See Also

Document.Save Method, Document.SaveAs

Document.Texts

This property returns the collection of free texts in the document.

Prototype

Texts as Collection

Texts (*Name* as String) as Text

Argument

name	Name of the text to retrieve (optional).

Comments

None

Sample

This sample shows the total number of texts in the design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
MsgBox "Number of Texts: " & ActiveDocument.Texts.Count
End Sub
```

Document.Unit

This property sets or returns the system unit used in the document.

Prototype

Unit As PPcbUnit

Arguments

None

Sample

The following sample code sets the unit in the open design to metric. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.Unit = ppcbUnitMetric
End Sub
```

Document.Vias

This property returns the collection of all vias.

Prototype

Vias As Objects

Vias(*name* As String) As Via

Argument

name Name of an existing via.

Comments

When an existing via name is passed to this property, it returns that Via object. If the via name does not exist, this property returns the collection of all vias in an Objects collection object.

Sample

The following sample code retrieves the number of vias in the open design using the Objects.Count property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set vias = ActiveDocument.Vias
MsgBox "There are " & vias.Count & " via(s) in " & ActiveDocument.Name
End Sub
```

See Also

Document.GetObjects

Drawing.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This is a Microsoft-required property.

Sample

Not required.

Drawing.DrawingType

This property returns the type of the open drawing.

Prototype

DrawingType as PPcbDrawingType

Arguments

None

Comments

None

Sample

The following sample function displays the name and type of the drawing. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,, TRUE)
For Each drw In selected
Select Case drw.DrawingType
Case ppcbDrw2Dline
Msgbox "Type: 2D Line"
Case ppcbDrwBoard
Msgbox "Type: Board Outline"
Case ppcbDrwCopper
Msgbox "Type: Copper"
Case ppcbDrwCopperPour
Msgbox "Type: CopperPour"
Case ppcbDrwCopperHatch
Msgbox "Type: CopperHatch"
Case ppcbDrwCopperThermal
Msgbox "Type: CopperThermal"
Case ppcbDrwKeepout
Msgbox "Type: Keepout"
End Select
Next drw
End Sub
```

Drawing.Geometry

This property returns a collection of objects, currently polylines, text, or circles, representing the geometry of the drawing.

Prototype

Geometry as Collection

Arguments

None

Comments

None

Sample

The following sample shows the number of polylines in the selected drawing. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
I = 0
Dim geom as Object
For Each geom in drw.Geometry
If geom.ObjectType = ppcbObjectTypePolyline Then I = I + 1
Next geom
MsgBox "Drawing contains " & I & " polylines"
Exit For
Next drw
End Sub
```

Drawing.Name

This default property returns the name of the drawing.

Prototype

Name as String

Arguments

None

Comments

None

Sample

The following example displays a message box showing the name of the first drawing in the active document. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each drw In ActiveDocument.Drawings
MsgBox "The first drawing's name is " & drw.Name
Exit For
Next drw
End Sub
```

Drawing.Net

This property returns a net with which the drawing is associated.

Prototype

Net as Net

Arguments

None

Comments

None

Sample

The following sample function displays the name of the parent object in the selected drawing. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
If drw.Net Is Nothing Then
Msgbox "No Net"
Else
Msgbox "Net: " & drw.Net.Name
End If
Next drw
End Sub
```

Drawing.ObjectType

This property returns the type of object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Sample

The following example shows the number of selected drawings. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set sel = ActiveDocument.GetObjects(,,TRUE)
n = 0
For Each obj In sel
If obj.ObjectType = ppcbObjectTypeDrawing Then n = n + 1
Next obj
MsgBox n & " drawing(s) selected"
End Sub
```

Drawing.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

This is a Microsoft-required property.

Drawing.PositionX

This property returns the X coordinate of the origin of the drawing.

Prototype

PositionX(*Unit* as PPcbUnit, *Origin* as PPcbOriginType) as Double

Arguments

unit [Optional] Unit in which the result is to be expressed. This optional argument is ppcbUnitCurrent by default.

origin [Optional]Type of reference point from which the result is counted. The default value is ppcbOriginTypeDesign.

Comments

None

Sample

The following sample function displays selected drawing positions. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing, ,TRUE)
For Each drw In selected
Msgbox "Position: (" & drw.PositionX & ", " & drw.PositionY & ") "
Next drw
End Sub
```

Drawing.PositionY

This property returns the Y coordinate of the origin of the drawing.

Prototype

PositionY(*Unit* as PPcbUnit, *Origin* as PPcbOriginType) as Double

Arguments

unit [Optional] Unit in which the result is to be expressed. This optional argument is ppcbUnitCurrent by default.

origin [Optional]Type of reference point from which the result is counted. The default value is ppcbOriginTypeDesign.

Comments

None

Sample

The following sample function displays the positions of the selected drawing objects. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing, ,TRUE)
For Each drw In selected
Msgbox "Position: (" & drw.PositionX & ", " & drw.Position.Y & ") "
Next drw
End Sub
```

Drawing.Selected

This property sets or returns whether the component is selected.

Prototype

Selected as Boolean

Arguments

None

Comments

None

Sample

The following sample function displays a message showing whether or not the first drawing is selected. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each drw In ActiveDocument.Drawings
Msgbox "Is Drawing " & drw.Name & " selected? " & drw.Selected
Exit For
Next drw
End Sub
```

Drawing.Texts

This property returns a collection of text objects associated with the drawing, or a specific text object, referenced by its name.

Prototype

Texts as Collection

Texts (*Name* as String) as Text

Argument

name Text object name.

Comments

None

Sample

The following sample function returns the number of texts associated with the selected drawing. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
MsgBox "Number of associated texts: " & drw.Texts.Count
Next drw
End Sub
```

Error.ActualValue

This property returns the actual value of the Error Object.

Prototype

Conflicts as Objects

Conflicts(conflictNumber as Integer) as ErrorConflict

Arguments

unit [Optional] Unit in which the actual value is returned.

Sample

The following sample code displays information about all of the errors' actual value in PADS Layout. See "Running Code Samples" for more information on running this sample.

```
' Display information about all errors' actual value in Layout
doc = Application.ActiveDocument
MsgBox "Total number of errors: " & doc.Errors.Count

For Each theErr In doc.Errors
   ' Error actual value information text
   info = "Actual value N/A"
   If theErr.HasActualValue Then
      info = " Actual value: " & theErr.ActualValue
      info = info & " [default unit]; "
      If (theErr.ValueType = ppcbErrorValueTypeMeasure) Then
         info = info & theErr.ActualValue(ppcbUnitCurrent)
         info = info & " [current unit]; "
         info = info & theErr.ActualValue(ppcbUnitDatabase)
         info = info & " [database unit]; "
         info = info & theErr.ActualValue(ppcbUnitMils)
         info = info & " [mils]; "
         info = info & theErr.ActualValue(ppcbUnitInch)
         info = info & " [inch]; "
         info = info & theErr.ActualValue(ppcbUnitMetric)
         info = info & " [millimeter]"
      End If
   End If
   MsgBox info
   ' Next error
Next
```

Error.Application

This property returns the Application object.

Prototype

Application as Application

Arguments

None

Error.Conflicts

This property returns the collection of all error conflicts in the Error Object.

Prototype

Conflicts as Objects

Conflicts(conflictNumber as Integer) as ErrorConflict

Arguments

conflictNumber	Conflict number

Comments

When the conflict number is passed to this property, it returns that ErrorConflict Object. If the number is not specified, this property returns the collection of all error conflicts in an Objects collection object.

Error.Description

This property returns the description of the Error Object.

Prototype

Description as String

Arguments

None

Error.ErrorClass

This property returns the error class of the Error Object.

Prototype

ErrorClass as PPcbErrorClass

Arguments

None

Error.ErrorType

This property returns the error type of the Error Object.

Prototype

ErrorType as PPcbErrorType

Arguments

None

Error.HasActualValue

This property returns if the actual value of the Error Object exists.

Prototype

HasActualValue as Boolean

Arguments

None

Error.HasRequiredValue

This property returns if the required value of the Error Object exists.

Prototype

HasRequiredValue as Boolean

Arguments

None

Error.IsClearanceError

This property returns if the error is a clearance error.

Prototype

IsClearanceError as Boolean

Arguments

None

Error.IsConnectivityError

This property returns if the error is a connectivity error.

Prototype

IsConnectivityError as Boolean

Arguments

None

Error.IsHighSpeedError

This property returns if the error is a high speed error.

Prototype

IsHighSpeedError as Boolean

Arguments

None

Error.IsIgnoredFlag

This property returns if the error is flagged as ignored.

Prototype

IsIgnoredFlag as Boolean

Arguments

None

Error.IsInvisibleFlag

This property returns if the error is flagged as invisible.

Prototype

IsInvisibleFlag as Boolean

Arguments

None

Error.IsLatiumError

This property returns if the error is a latium error.

Prototype

IsLatiumError as Boolean

Arguments

None

Error.IsMiscError

This property returns if the error is a miscellaneous error.

Prototype

IsMiscError as Boolean

Arguments

None

Error.IsTestPointError

This property returns if the error is a Test Point error.

Prototype

IsTestPointError as Boolean

Arguments

None

Error.LayerNumber

This property returns the layer number of this Error Object.

Prototype

LayerNumber as int

Arguments

None

Error.Name

This property returns the name of this error.

Prototype

Name As String

Arguments

None

Error.ObjectType

This property returns the type of this object - ppcbObjectTypeError.

Prototype

ObjectType as PPcbObjectType

Arguments

None

Error.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Error.PositionX

This property returns the X-coordinate of the Error Object.

Prototype

PositionX as Double

PositionX(unit as PPcbUnit) as Double

Arguments

unit	[Optional Argument] Unit in which the X-coordinate is returned.

Error.PositionY

This property returns the Y-coordinate of the Error Object.

Prototype

PositionY as Double

PositionY(unit as PPcbUnit) as Double

Arguments

unit	[Optional Argument] Unit in which the Y-coordinate is returned.

Error.RequiredValueMax

Returns the required maximum value of the Error Object. If this value equals the maximum real value (see Document.MaxRealValue) it means infinity.

Prototype

RequiredValueMax as Double

RequiredValueMax (unit as PPcbUnit) as Double

Arguments

unit	[Optional Argument] Unit in which the required maximum value is returned.

Error.RequiredValueMin

Returns the required minimum value of the Error Object. If this value equals the minimum real value (see Document.MinRealValue) it means minus infinity.

Prototype

RequiredValueMin as Double

RequiredValueMin (unit as PPcbUnit) as Double

Arguments

unit	[Optional Argument] Unit in which the required minimum value is returned.

ErrorConflict.Application

This property returns the Application object.

Prototype

Application as Application

Arguments

None

ErrorConflict.ConflictObject

Returns the conflicting object.

Prototype

ConflictObject as Object

Argument

None

ErrorConflict.ConflictObjectDesc

Returns the description of the conflicting object.

Prototype

ConflictObjectDesc as String

Argument

None

ErrorConflict.ConflictObjectType

Returns the type of the conflicting object.

Prototype

ConflictObjectType as PPcbObjectType

Argument

None

ErrorConflict.Name

This property returns the name of this error conflict.

Prototype

Name as String

Argument

None

ErrorConflict.ObjectType

Returns the type of the object - ppcbObjectTypeErrorConflict.

Prototype

ObjectType as PPcbObjectType

Argument

None

ErrorConflict.Parent

This property returns the parent of the object.

Prototype

Parent as Document

Arguments

None

Jumper.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an automation object. All automation server applications have an application object and all automation objects have an application property. This property is usually used in automation client applications that handle large volumes of objects from different sources, such as different automation server applications, to quickly identify the application to which the object belongs.

Jumper.Installed

This property sets or returns whether the component is installed in the current assembly option.

Prototype

Installed As Boolean

Arguments

None

Comments

This property is useful only when the parent of the component is an Assembly Option. If the parent of the component is not an Assembly Option, this property always returns True and cannot be set to a different value.

Sample

The following sample code creates a new Assembly Option named EmptyAssOpt in the open design, and then uninstalls all components in that newly created Assembly Option. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set newAssOpt = ActiveDocument.AssemblyOptions.Add("EmptyAssOpt")
For Each nextComp In newAssOpt.Components
nextComp.Installed = False
Next nextComp
End Sub
```

Jumper.Length

This property returns the length of the jumper.

Prototype

Length([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the length value is returned.

Comments

None

Sample

The following sample code retrieves the length of jumper JMP1, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set jmp1 = ActiveDocument.Jumpers("JMP1")
MsgBox "Jumper " & jmp1.name & " has a length of " & jmp1.Length
End Sub
```

Jumper.Name

This property returns the name of the jumper.

Prototype

Name As String

Arguments

None

Comments

This property is the default property for the Jumper object.

Sample

The following sample code lists all jumpers in the open design and then places that list in a custom dialog box. When a jumper is selected in the list box, the sample selects that jumper.

This sample uses the User Dialog Editor in the Sax Basic Engine in PADS Layout. See "Running Code Samples" for more information on running this sample.

See also: Sax Basic Editor On Line Help (C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\sbe5_000.hlp)

```
Dim ListJmps$(10000)
Sub Main
index = 0
For Each nextJmp In ActiveDocument.Jumpers
ListJmps$(index) = nextJmp.Name
index = index + 1
Next nextJmp
' This piece of code is automatically generated by the Basic Dialog Editor
in PADS Layout.
Begin Dialog UserDialog 180,238,"Jumpers",.CallbackFunc ' %GRID:10,7,1,1
ListBox 10,7,160,203,ListJmps(),.ListBox1
OKButton 10,210,160,21
End Dialog
Dim dlg As UserDialog
Dialog dlg
End Sub
' The following function is automatically called by the system when
something has happened
' in the dialog; it is used to easily process user actions.
Function CallbackFunc%(DlgItem$, Action%, SuppValue%)
Select Case Action%
Case 2 ' Value changing or button pressed
If DlgItem$ = "ListBox1" Then
ActiveDocument.SelectObjects(ppcbObjectTypeAll, , False)
ActiveDocument.SelectObjects(ppcbObjectTypeJumper, ListJmps(SuppValue%))
End If
End Select
End Function
```

Jumper.Net

This property returns the net connected to the jumper.

Prototype

Net As Net

Arguments

None

Comments

None

Sample

The following sample code identifies the net connected to jumper JMP1, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set jmp1 = ActiveDocument.Jumpers("JMP1")
MsgBox "Jumper " & jmp1.name & " is connected to net " & jmp1.Net.Name
End Sub
```

Jumper.ObjectType

This property returns the type of the object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property returns ppcbObjectTypeJumper.

All PADS Layout database objects in the PADS Layout Automation server implement this property to compensate for the lack of a Visual Basic equivalent for the Visual C++ QueryInterface function.

This property is generally used:

- To identify the kind of PADS Layout database objects in a heterogeneous Objects collection.
- When implementing a generic routine that depends on the type of the PADS Layout database object passed as argument. For example:

```
Sub DoSomething(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeComponent
' Do something specific to component objects
Case ppcbObjectTypeNet
' Do something specific to net objects
Case ppcbObjectTypePin
' Do something specific to pin objects
Case ppcbObjectTypeVia
' Do something specific to via objects
Case ppcbObjectTypeConnection
' Do something specific to connection objects
Case ppcbObjectTypeRouteSegment
' Do something specific to route segment objects
Case ppcbObjectTypeJumper
' Do something specific to jumper objects
Case Else
MsgBox "Not a PADS Layout database object"
End Select
End Sub
```

Jumper.Orientation

This property returns the orientation of the jumper.

Prototype

Orientation As Double

Arguments

None

Comments

None

Sample

The following sample code retrieves the orientation of jumper JMP1, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set jmp1 = ActiveDocument.Jumpers("JMP1")
MsgBox "Jumper " & jmp1.name & " has an orientation of " &
jmp1.Orientation & " degrees."
End Sub
```

Jumper.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

None

Jumper.Points

This property returns the array of points defining the jumper.

Prototype

Points([*unit* As PPcbUnit]) AsVariant

Argument

unit	[Optional] Unit in which the point coordinate values are returned.

Comments

None

Jumper.Selected

This property sets or returns whether the jumper is selected.

Prototype

Selected As Boolean

Arguments

None

Comments

You can also select a PADS Layout database object using the Document.SelectObjects and Objects.Select methods.

Sample

The following sample code selects jumper JMP1, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.SelectObjects(,,False)
ActiveDocument.Jumpers("JMP1").Selected = True
End Sub
```

See Also

Document.SelectionChange Event

Label.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This is a Microsoft-required property.

Sample

Not required.

Label.Attribute

This property returns an attribute with which the label is associated.

Prototype

Attribute as Attributes

Arguments

None

Comments

None

Sample

The following sample routine displays the name of an attribute with which the label is associated, if any. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,,TRUE)
For Each lab In selected
If Not lab.Attribute Is Nothing Then
MsgBox "The label is associated with attribute " & lab. Attribute.Name
Else
MsgBox "The label is not associated with any attribute"
End If
Next lab
End Sub
```

Label.Component

This property returns a component with which the label is associated.

Prototype

Component as Component

Arguments

None

Comments

None

Sample

The following sample routine displays the name of a component with which the selected label is associated, if any. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,,TRUE)
For Each lab In selected
If Not lab.Component Is Nothing Then
MsgBox "The label is associated with component " & lab.Component.Name
Else
MsgBox "The label is not associated with any component"
End If
Next lab
End Sub
```

Label.Display

This property returns or sets the label display mode.

Prototype

Display as PPcbLabelDisplayMode

Arguments

None

Comments

None

Sample

The following sample shows the display type of the selected label. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,,TRUE)
For Each label In selected
Select Case label.Display
Case ppcbLabelDisplayNone
MsgBox "Label shows nothing"
Case ppcbLabelDisplayValue
MsgBox "Label shows attribute value"
Case ppcbLabelDisplayNameAndValue
MsgBox "Label shows attribute name and value"
Case ppcbLabelDisplayFullNameAndValue
MsgBox "Label shows attribute full name and value"
End Select
Next label
End Sub
```

Label.Name

This default property returns the name of the Label object.

Prototype

Name as String

Arguments

None

Comments

None

Sample

The following example displays a message box showing the name of the selected label in the active document. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,,TRUE)
For Each label In selected
MsgBox "Selected label name: " & label.Name
Next label
End Sub
```

Label.ObjectType

This property returns the type of object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Sample

The following example shows the number of selected labels. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(,,TRUE)
n = 0
For Each obj In selected
If obj.ObjectType = ppcbObjectTypeLabel Then n = n + 1
Next obj
MsgBox n & " label(s) selected"
End Sub
```

Label.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

This is a Microsoft-required property.

Label.RightReading

This property returns or sets the right-to-left reading status of the label.

Prototype

RightReading as PPcbRightReadingStatus

Arguments

None

Comments

None

Sample

This sample switches the selected label to Orthogonal Right-Reading mode. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,,TRUE)
For Each lab In selected
Select Case lab.RightReading
Case ppcbRightReadingNone
s = "None"
Case ppcbRightReadingOrthogonal
s = "Orthogonal"
Case ppcbRightReadingAngled
s = "Angled"
End Select
MsgBox "Selected label Right Reading mode: " & s
lab.RightReading = ppcbRightReadingOrthogonal
Next lab
End Sub
```

Label.Selected

This property sets or returns whether the Label object is selected.

Prototype

Selected as Boolean

Arguments

None

Comments

None

Sample

The following sample displays messages indicating whether the label is selected for all labels of the first component, then selects the labels. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
For Each label In comp.Labels
Msgbox "Is Label " & label.Text & " selected ?" & label.Selected
label.Selected = True
Next label
Exit For
Next comp
End Sub
```

Label.Text

This property returns the displayed string for the Label object.

Prototype

Text as String

Arguments

None

Comments

None

Sample

This sample shows the text of the selected labels. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,, TRUE)
For Each label In selected
MsgBox "Selected label: " & label.Text
Next label
End Sub
```

Label.Type

This property returns the type of label.

Prototype

LabelType as PPcbLabelType

Arguments

None

Comments

None

Sample

The following sample displays the type of the selected label. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,,TRUE)
For Each label In selected
Select Case label.LabelType
Case ppcbLabelTypeRefDesignator
MsgBox "Label type: Ref Designator"
Case ppcbLabelTypePartType
MsgBox "Label type: Part Type"
Case ppcbLabelTypeAttribute
MsgBox "Label type: Attribute"
End Select
Next label
End Sub
```

Layer.Application

This property returns the application object.

Prototype

Application as Application

Argument

None

Layer.CopperThickness

This property sets or returns the layer copper thickness.

Prototype

CopperThickness (*unit* as PPcbUnit) as Double

Argument

unit	[Optional] Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

Layer.Enabled

This property sets or returns the layer enabled property.

Prototype

Enabled as Boolean

Argument

None

Layer.Name

This property sets or returns the name of this layer.

Prototype

Name as String

Argument

None

Layer.Number

This property returns the layer number.

Prototype

Number as Integer

Argument

None

Layer.ObjectType

This property returns the type of the object - ppcbObjectTypeLayer.

Prototype

ObjectType as PPcbObjectType

Argument

None

Layer.Parent

This property returns the parent of the object.

Prototype

Parent as Document

Argument

None

Layer.PlaneType

This property sets or returns the layer plane type.

Prototype

PlaneType as PPcbPlaneType

Argument

None

Layer.RoutingDirection

This property sets or returns the layer routing direction.

Prototype

RoutingDirection as PPcbRoutingDirection

Argument

None

Layer.Type

This property sets or returns the layer type.

Prototype

Type as PPcbLayerType

Argument

None

Layer.Visible

This property sets or returns the layer visibility.

Prototype

Visible as Boolean

Argument

None

Library.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This is a Microsoft-required property.

Sample

Not required.

Library.FullName

This property returns the full name of library files, including path and name.

Prototype

FullName As String

Arguments

None

Comments

None

Sample

The following example displays a message box showing the name of the first available library. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each lib In Libraries
MsgBox "The full name of the first available library is " & lib.FullName
Exit For
Next lib
End Sub
```

Library.Name

This default property returns the name of the library.

Prototype

Name As String

Arguments

None

Comments

This is a Microsoft-required property.

Sample

The following example displays a message box showing the name of the first available library. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each lib In Libraries
MsgBox "The name of the first available library is " & lib.Name
Exit For
Next lib
End Sub
```

Library.ObjectType

This property returns the type of object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Sample

The following example tests the ObjectType property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each lib In Libraries
If lib.ObjectType <> ppcbObjectTypeLibrary Then
MsgBox "Test failed"
End If
Next lib
End Sub
```

Library.Parent

This property returns the parent of this object.

Prototype

Parent As Application

Arguments

None

Comments

This is a Microsoft-required property.

Library.Path

This property returns the path to the library.

Prototype

Path As String

Arguments

None

Comments

None

Sample

The following example displays a message box showing the name of the first available library. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each lib In Libraries
MsgBox "The path to the first available library is " & lib.Path
Exit For
Next lib
End Sub
```

LibraryItem.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This is a Microsoft-required property.

LibraryItem.Library

This property returns the library the library item belongs to.

Prototype

Library As Object

Arguments

None

Comments

None

Sample

The following sample code displays the name of the library an item belongs to. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set items = GetLibraryItems()
For Each item In items
MsgBox "The item " & item & " belongs to " & item.Library & " library"
Exit For
Next item
End Sub
```

LibraryItem.Name

This default property returns the name of the library item.

Prototype

Name As String

Arguments

None

Comments

This is a Microsoft-required property.

Sample

The following example displays a message box showing the name of the first item in the available libraries. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set items = GetLibraryItems()
For Each item In items
MsgBox "The first item name is " & item.Name
Exit For
Next item
End Sub
```

LibraryItem.ObjectType

This property returns the type of object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Sample

The following example tests the ObjectType property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set items = GetLibraryItems()
For Each item In items
If item.ObjectType <> ppcbObjectTypeLibraryItem Then
MsgBox "Test failed"
End If
Next item
End Sub
```

LibraryItem.Parent

This property returns the parent of this object.

Prototype

Parent As Application

Arguments

None

Comments

This is a Microsoft-required property. The parent property of a LibraryItem object always returns the application.

LibraryItem.Type

This property returns the type of the library item object.

Prototype

Type as PPcbLibraryItemType

Arguments

None

Comments

None

Sample

This sample displays the type of a library item. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set items = GetLibraryItems()
For Each item In items
Select Case item.Type
Case ppcbLibraryItemTypePartType
MsgBox "Item type: PartType"
Case ppcbLibraryItemType
MsgBox "Item type: Decal"
Case ppcbLibraryItemType
MsgBox "Item type: LogicDrawing"
Case ppcbLibraryItemType
MsgBox "Item type: Drawing"
End Select
Exit For
Next item
End Sub
```

Measure.Application

This property returns the Measure object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an automation object. All automation server applications have an application object and all automation objects have an application property. This property is usually used in large automation client applications that handle large volumes of objects from different sources, such as different automation server applications, to quickly identify the application to which the object belongs.

Measure.Name

This property returns the name of the quantity represented by the measure. For example "Capacitance" for measure "10pF", "Voltage" for "5V".

Prototype

Name As String

Arguments

None

Sample 1

The following sample code retrieves the quantity name of "10pF" measure (the Capacitance). See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox Measure("10pF").Quantity ' displays Capacitance
End Sub
```

Sample 2

The following sample code retrieves the quantity name of "10k" measure (the Resistance).

```
Sub Main
MsgBox Measure("10k").Name ' displays Resistance
End Sub
```

Sample 3

The following sample code retrieves the quantity name of "12V" measure (the Voltage).

```
Sub Main
MsgBox Measure("12V").Name ' displays Voltage
End Sub
```

See Also

Measure.Value

Measure.Number

This property returns the number to combine with a prefix and unit.

Prototype

Number As Double

Arguments

None

Comments

This property returns the left numeric part of the measurement. You can format this number before output using standard methods available in the programming language you use for creating a client. Use this property with Measure.Prefix and Measure.Unit.

Sample

The following sample code retrieves a formatted measurement (to a hundredth of a decimal point) using standard Basic formatting. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set Cap = Measure (1.12345e-12, "F")
MsgBox Format(Cap.Number, "#.###") & Cap.Prefix & Cap.Unit
End Sub
```

See Also

Measure.Unit, Measure.Prefix

Measure.Normalize

This property normalizes the text value of the measure and returns a new text.

Prototype

Normalize As String

Arguments

None

Comments

This property selects the appropriate unit prefix and appends a unit if it is missing. For example, it converts a time measurement from “5e-9” to “5ns.”

Sample

The following sample code normalizes a capacitance measure. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set Cap = Measure(2e-10, "F")
MsgBox Cap.Normalize ' shows 200pF
End Sub
```

See Also

Measure.Text

Measure.ObjectType

This property returns the type of this object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Measure.Parent

This property returns the PADS Layout Application object.

Prototype

Parent As Application

Arguments

None

Comments

None

Measure.Prefix

This property returns the unit prefix to combine with a number and unit.

Prototype

Prefix([*format* As PPcbMeasureFormat = ppcbMeasureFormatStandard]) As String

Argument

format	[Optional] Indicates the format of the prefix.

Comments

This property returns the prefix currently used in the measurement. Use this property with Measure.Number and Measure.Unit.

Possible format values:

ppcbMeasureFormatStandard	To return a standard prefix representation (for example, p for Pico or k for Kilo).
ppcbMeasureFormatCurrent	To return a prefix in the format currently used in the Measure value.
ppcbMeasureFormatShort	To return a short prefix (for example, p for Pico or k for Kilo).
ppcbMeasureFormatLong	To return a long unit name (for example, Pico, Kilo, or Mega).

Sample

See the example for Measure.Number

See Also

Measure.Number, Measure.Unit

Measure.Text

This property sets or returns the exact text value of the measure.

Prototype

Text As String

Arguments

None

Comments

This property defines a custom format of a measurement. The text value consists of a number followed by an optional prefix and unit. The text value always matches the Attribute.Value property if an attribute represents a measure.

Sample

The following sample code demonstrates the difference between Measure.Text, Measure.Value, and Attribute.Value properties. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set C1 = ActiveDocument.Components("C1")
Set Cap = Measure("500pF")
C1.Attributes.Add "Capacitance", Cap
MsgBox Cap 'displays 0.0000000005 (the default Value property)
MsgBox Cap.Value 'displays 0.0000000005
MsgBox Cap.Text 'displays 500pF
MsgBox C1.Attributes("Capacitance") 'displays 500pF
MsgBox C1.Attributes("Capacitance").Value 'displays 500pF (long form)
End Sub
```

See Also

Measure.Value

Measure.Unit

This property returns the unit name of the measure.

Prototype

Unit([*format* As PPcbMeasureFormat = ppcbMeasureFormatStandard]) As String

Argument

format	[Optional] Indicates the format of the unit representation.

Comments

This property returns the name of the physical unit without a prefix.

Possible Format values:

ppcbMeasureFormatStandard	To return a standard unit representation (for example, F for Capacitance).
ppcbMeasureFormatCurrent	To return the unit in the format currently used in this Measure value.
ppcbMeasureFormatShort	To return a short unit name (for example F for Capacitance).
ppcbMeasureFormatLong	To return a long unit name (for example Farad for Capacitance).

Sample

See the example for Measure.Number.

See Also

Measure.Number, Measure.Prefix

Measure.Value

This property sets or returns the value of the measure.

Prototype

Value As Double

Arguments

None

Comments

This property returns a floating-point number for a measure, taking into account the unit prefix. For example, 10000 for 10K or 2e-10 for 200pF.

This is a default property, so it can be omitted in Basic scripts.

If you set a new Value, then Measure is automatically normalized. See also: Measure.Normalize

Sample

The following sample code demonstrates how to work with measures. See "Running Code Samples" for more information on running this sample.

```
Sub Main
'1 - Compare two existing measure attributes (.Value call may be omitted)
Set U1_Val =
ActiveDocument.Components("U1").Attributes("Value").Measure.Value
Set U2_Val =
ActiveDocument.Components("U2").Attributes("Value").Measure.Value
Set U1_Quantity =
ActiveDocument.Components("U1").Attributes("Value").Measure.Name
Set U2_Quantity =
ActiveDocument.Components("U2").Attributes("Value").Measure.Name
If U1_Quantity <> U2_Quantity then
MsgBox "Cannot compare values with different physical units"
ElseIf U1_Val < U2_Val then
MsgBox "U1 value is less than U2 value"
ElseIf U1_Val > U2_Val then
MsgBox "U1 value is greater than U2 value"
Else
MsgBox "U1 value is equal to U2 value"
End If
'2 - Check that the Resistor's Value is in range between 100K and 10M
Set R1_Val = ActiveDocument.Components("R1").Attributes("Value").Measure
If R1_Val >= Measure("100k") And R1_Val <= Measure("10M") And R1_Val.Name
= "Resistence" Then
MsgBox "Resistor Is In Range [100k, 10M]
End If
'3 - Calculate total thermal dessipation for all parts
'make sure attribute exists for all parts
For Each part In ActiveDocument.Components
If part.Attributes("Thermal.Dissipation") Is Nothing Then
```

```
Part.Attributes.Add "Thermal.Dissipation", Measure("10mW")
End If
Next
Dim Total As Measure 'declare Total as Measure object explicitly!
Set Total = Measure("0mW") 'create Measure to accamulate the total
For Each part In ActiveDocument.Components
Total = Total + part.Attributes("Thermal.Dissipation").Measure
Next
MsgBox "Total Thermal Dissipation = " & Total.Text
End Sub
```

See Also

Measure.Text, Measure.Name

Net.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an Automation object. All automation server applications have an application object and all automation objects have an application property. This property is usually used in automation client applications that handle large volumes of objects from different sources, such as different automation server applications, to quickly identify the application to which the object belongs.

Net.AssociatedNet

This property returns the associated net which the net belongs to.

Prototype

```
AssociatedNet as AssociatedNet
```

Arguments

None

Returns

This property returns Nothing if the net doesn't belong to any associated net.

Net.Attributes

This property returns the collection of all attributes assigned to the net.

Prototype

Attributes As Attributes

Attributes(*name* As String) As Attribute

Argument

name	Name of an existing net attribute.

Comments

When an existing attribute *name* is passed to this property, it returns that net Attribute object. If the attribute *name* does not exist, this property returns the collection of all net attributes in an Attributes collection object.

Sample

The following sample code retrieves the number of attributes assigned to net VCC, assuming it exists in the open design, using the Attributes.Count property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set attrs = ActiveDocument.Nets("VCC").Attributes
MsgBox "There are " & attrs.Count & " attribute(s) in net VCC."
End Sub
```

Net.Connections

This property returns the collection of all connections on the net.

Prototype

Connections As Objects

Connections(*name* As String) As Connection

Argument

name Name of an existing connection.

Comments

When an existing connection *name* is passed to this property, it returns that Connection object. If the connection *name* does not exist, this property returns the collection of all connections of the net in an Objects collection object.

Sample

The following sample code retrieves the number of connections in net VCC, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "VCC has " & ActiveDocument.Nets("VCC").Connections.Count & "
connections."
End Sub
```

Net.Drawings

This property returns the collection of coppers and copper pours belonging to the net object, or a specific copper or copper pour.

Prototype

Drawings as Collection

Drawings (*Name* as String) as Drawing

Argument

name Name of the drawing.

Comments

None

Sample

The following sample code displays the number of drawings associated with a net. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each net In ActiveDocument.Nets
If net.Drawings.Count > 0 Then
MsgBox "Net " & net.Name & " has " & net.Drawings.Count & " drawings"
Exit For
End If
Next net
End Sub
```

Net.Length

This property returns the length of the net.

Prototype

Length([*bRouted* As Boolean = False], [*unit* As PPcbUnit]) As Double

Arguments

bRouted	[Optional] True to retrieve the routed length only. False to retrieve the total length, including unrouted parts approximated using the Manhattan method.
unit	[Optional] Unit in which the length is returned.

Comments

None

Sample

The following sample code retrieves the routed, unrouted, and total length of net VCC, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
lenTotal = Format$(ActiveDocument.Nets("VCC").Length, "Fixed")
lenRouted = Format$(ActiveDocument.Nets("VCC").Length(True), "Fixed")
MsgBox "VCC's total length is (in current unit) " & lenTotal & " including
" & lenRouted & " routed and " & lenTotal-lenRouted & " unrouted."
End Sub
```

Net.Name

This property returns the name of the net. For example, this property returns the string “GND” for net GND.

This property is the default property for the Net object.

Prototype

Name As String

Arguments

None

Sample

The following sample code lists all nets in the open design and then places that list in a custom dialog box. When a net is selected in the list box, the sample selects that net.

This sample uses the UserDialog Editor in the Sax Basic Engine in PADS Layout. See “Running Code Samples” for more information on running this sample.

See also: Sax Basic Editor On Line Help (C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\sbe5_000.hlp)

```
Dim ListNets$(10000)
Sub Main
index = 0
For Each nextNet In ActiveDocument.Nets
ListNets$(index) = nextNet.Name
index = index + 1
Next nextNet
' This piece of code is automatically generated by the Basic Dialog Editor
in PADS Layout.
Begin Dialog UserDialog 180,238,"Nets",.CallbackFunc ' %GRID:10,7,1,1
ListBox 10,7,160,203,ListNets(),.ListBox1
OKButton 10,210,160,21
End Dialog
Dim dlg As UserDialog
Dialog dlg
End Sub
' The following function is automatically called by the system when
something has happened
' in the dialog; it is used to easily process user actions.
Function CallbackFunc%(DlgItem$, Action%, SuppValue%)
Select Case Action%
Case 2 ' Value changing or button pressed
If DlgItem$ = "ListBox1" Then
ActiveDocument.SelectObjects(ppcbObjectTypeAll, , False)
ActiveDocument.SelectObjects(ppcbObjectTypeNet, ListNets(SuppValue%))
End If
End Select
End Function
```

Net.NetClass

This property returns the net object's net class.

Prototype

NetClass as NetClass

Arguments

None

Comments

None

Sample

The following sample code displays the name of the net class a net belongs to, if any. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each net In ActiveDocument.Nets
If Not net.NetClass Is Nothing Then
MsgBox "Net " & net.Name & " belongs to class " & net.NetClass.Name
Exit For
End If
Next net
End Sub
```

Net.NetClassAttributes

This property returns the collection of all attributes assigned to the net class.

Prototype

NetClassAttributes As Attributes

NetClassAttributes(*name* As String) As Attribute

Argument

name	Name of an existing net class attribute.

Comments

When an existing attribute *name* is passed to this property, it returns that net class Attribute object. If the attribute *name* does not exist, this property returns the collection of all net class attributes in an Attributes collection object.

Sample

The following sample code retrieves the number of attributes assigned to the net class of net VCC, assuming it exists in the open design, using the Attributes.Count property. See NetClass Attributes for more information on running this sample.

```
Sub Main
Set attrs = ActiveDocument.Nets("VCC").NetClassAttributes
MsgBox "There are " & attrs.Count & " attribute(s) in net class " &
ActiveDocument.Nets("VCC").Name
End Sub
```

Net.ObjectType

This property returns the type of the object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property returns ppcbObjectTypeNet.

All database objects in the Automation server implement this property to compensate for the lack of a Visual Basic equivalent for the Visual C++ QueryInterface function.

This property is generally used:

- To identify the kind of database objects in a heterogeneous Objects collection.
- When implementing a generic routine that depends on the type of the database object passed as argument. For example:

```
Sub DoSomething(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeComponent
' Do something specific to component objects
Case ppcbObjectTypeNet
' Do something specific to net objects
Case ppcbObjectTypePin
' Do something specific to pin objects
Case ppcbObjectTypeVia
' Do something specific to via objects
Case ppcbObjectTypeConnection
' Do something specific to connection objects
Case ppcbObjectTypeRouteSegment
' Do something specific to route segment objects
Case ppcbObjectTypeJumper
' Do something specific to jumper objects
Case Else
MsgBox "Not a PADS Layout database object"
End Select
End Sub
```

Net.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

None

Net.Pins

This property returns the collection of all pins connected to the net.

Prototype

Pins As Objects

Pins(*name* As String) As Pin

Argument

name	Name of an existing pin.

Comments

When an existing pin *name* is passed to this property, it returns that Pin object. If the pin *name* does not exist, this property returns the collection of all pins connected to the net in an Objects collection object.

Sample

The following sample code retrieves the number of pins connected to net VCC, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "VCC connects " & ActiveDocument.Nets("VCC").Pins.Count & " pins."
End Sub
```

Net.Power

This property returns whether the net is a power net.

Prototype

Power As Boolean

Arguments

None

Comments

None

Sample

The following sample code retrieves the number of power nets in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
nbPowerNets = 0
For Each nextNet In ActiveDocument.Nets
If nextNet.Power = True Then nbPowerNets = nbPowerNets +1
Next nextNet
MsgBox "There are " & nbPowerNets & " power nets (out of " &
ActiveDocument.Nets.Count & ") in " & ActiveDocument.Name
End Sub
```

Net.Selected

This property sets or returns whether the net is selected.

Prototype

Selected As Boolean

Arguments

None

Comments

You can also select a database object using the Document.SelectObjects and Objects.Select methods.

Sample

The following sample code selects net VCC only, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.SelectObjects(,,False)
ActiveDocument.Nets("VCC").Selected = True
End Sub
```

See Also

Document.SelectionChange Event

Net.Vias

This property returns the collection of all vias connected to the net.

Prototype

Vias As Objects

Vias(*name* As String) As Via

Argument

name	Name of an existing via.

Comments

When an existing via name is passed to this property, it returns that Via object. If the via name does not exist, this property returns the collection of all vias connected to the net in an Objects collection object.

Sample

The following sample code retrieves the number of vias connected to net VCC, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "VCC connects " & ActiveDocument.Nets("VCC").Vias.Count & " vias."
End Sub
```

NetClass.Application

This property returns the Application object.

Prototype

Application as Application

Arguments

None

Comments

This is a Microsoft-required property.

NetClass.Attributes

This property returns a collection of attributes of the net class, or a specific attribute, referenced by its name.

Prototype

Attributes as Attributes

Attributes (*Name* as String) as Attribute

Argument

name   Name of the attribute to retrieve.

Comments

None

Sample

The following sample code displays a message box showing the number of the attribute for a net class. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each class In ActiveDocument.NetClasses
Set attrs = class.Attributes
MsgBox "There are " & attrs.Count & " attribute(s) for net class " &
class.Name
Exit For
Next class
End Sub
```

NetClass.Name

This default property returns the name of the net class.

Prototype

Name as String

Arguments

None

Comments

This is a Microsoft-required property.

Sample

The following sample code displays a message box showing the number of nets in the first net class in the active document. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each class In ActiveDocument.NetClasses
MsgBox "Net Class " & class.Name & " includes " & class.Nets.Count & "
nets"
Exit For
Next class
End Sub
```

NetClass.Nets

This property returns the collection of all nets of the net class, or a specific net of the net class, referenced by its name.

Prototype

Nets as Collection

Nets (*Name* as String) as Net

Argument

name	Name of the net to retrieve.

Comments

None

Sample

The following sample code retrieves the number of nets in a net class, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each class In ActiveDocument.NetClasses
MsgBox "Net Class " & class.Name & " includes " & class.Nets.Count & "
nets"
Exit For
Next class
End Sub
```

NetClass.ObjectType

This property returns the type of object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Sample

The following sample code finds a net class starting with “A.” See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set objs = ActiveDocument.GetObjects(, "A*")
For Each obj In objs
If obj.ObjectType = ppcbObjectTypeNetClass Then
MsgBox obj.Name & " is a net class"
Exit For
End If
Next obj
End Sub
```

NetClass.Parent

This property returns the parent of the object.

Prototype

Parent as Document

Arguments

None

Comments

This is a Microsoft-required property.

Objects.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This is a Microsoft-required property.

This property identifies the object as an Automation object. All Automation server applications have an Application object and all Automation objects have an Application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

Objects.Count

This property returns the number of objects in the collection.

Prototype

Count As Long

Arguments

None

Comments

None

Objects.Item

This property returns the object given its index or its name.

Prototype

Item(*index* As Long) As Object

Item(*name* As String) As Object

Arguments

index	Index (in the collection) of the object to retrieve.
name	Name of the object to retrieve.

Comments

This is the default member of the Objects collection object.

This property generates an exception if the *index* passed to this function is negative or out of range, or if the *index* or *name* argument is not valid.

Sample

The following sample code shows two different methods for iterating through all database objects in the open design, using Component objects here as an example. The second method is preferred because it is cleaner and faster. See “Running Code Samples” for more information on running this sample.

```
' Method 1: Use the Object.Item property
Sub Main
Set comps = ActiveDocument.Components
For I=1 To comps.Count
Set thisComp = comps.Item(I)
' Do something with the component thisComp
Next I
End Sub
' Method 2: Do not use the Object.Item property (preferred method)
Sub Main
For Each nextComp in ActiveDocument.Components
' Do something with the component nextComp
Next nextComp
End Sub
```

See Also

Objects.ItemType

Objects.ItemType

This property returns the type of an object given its index.

Prototype

ItemType(*index* As long) As PPcbObjectType

ItemType(*name* As String) As PPcbObjectType

Arguments

index	Index of the object in the collection to query.
name	Name of the object to retrieve.

Comments

This property generates an exception if the index argument is not valid.

See Also

Objects.Item

Objects.Next

This property returns the index of the next object of a specified type after the specified index.

Prototype

Next(*index* As long, *type* As PPcbObjectType) As Long

Arguments

index	Index of the object in the collection to query.
type	Type of the object to query.

Comments

This property generates an exception if the *index* argument is not valid.

If Index = zero (0), then this function returns the index of the first item of the given type. If an item is not found, then the return value is zero (0).

Objects.ObjectType

This property returns the type of this object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Objects.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

This is a Microsoft-required property.

Objects.ParentObject

This property returns the parent object of the collection.

Prototype

ParentObject as Object

Arguments

None

Comments

If a collection has ParentObject, it means the collection is “active.” Addition or deletion of an item to or from the collection modifies the parent object as well. Otherwise, only the collection is affected.

Pad.Application

This property returns the application object.

Prototype

Application as Application

Argument

None

Pad.CornerRadius

This property returns the pad's corner radius. This property should be used for the following pad shapes: ppcbPadShapeSquare, ppcbPadShapeRectangularFinger.

Prototype

CornerRadius (*unit* as PPcbUnit) as Double

Argument

unit - [Optional]	Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default

Pad.CornerType

This property returns the pad's corner type. This property should be used for the following pad shapes: ppcbPadShapeSquare, ppcbPadShapeRectangularFinger.

Prototype

CornerType as PPcbPadCornerType

Argument

None

Pad.Diameter

This property returns the pad's diameter. This property should be used for the following pad shapes: ppcbPadShapeRound, ppcbPadShapeAnnular, ppcbPadShapeOdd.

Prototype

Diameter (*unit* as PPcbUnit) as Double

Argument

unit - [Optional]	Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

Pad.InnerDiameter

This property returns the pad's inner diameter. This property should be used for the following pad shapes: ppcbPadShapeAnnular.

Prototype

InnerDiameter (*unit* as PPcbUnit) as Double

Argument

unit - [Optional]	Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

Pad.Length

This property returns the pad's length. This property should be used for the following pad shapes: ppcbPadShapeOvalFinger, ppcbPadShapeRectangularFinger.

Prototype

Length (*unit* as PPcbUnit) as Double

Argument

unit - [Optional]	Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

Pad.Name

This property returns the name of this pad.

Prototype

Name as String

Argument

None

Pad.ObjectType

This propertry returns the type of the object - ppcbObjectTypePad.

Prototype

ObjectType as PPcbObjectType

Argument

None

Pad.Offset

This property returns the pad's offset. This property should be used for the following pad shapes: ppcbPadShapeOvalFinger, ppcbPadShapeRectangularFinger.

Prototype

Offset (unit as PPcbUnit) as Double

Argument

unit - [Optional]	Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

Pad.Orientation

This property returns the pad's orientation. This property should be used for the following pad shapes: ppcbPadShapeOvalFinger, ppcbPadShapeRectangularFinger.

Prototype

Orientation as Double

Argument

None

Pad.PadStackLayer

This property returns the PadStackLayer Object to which this pad belongs to.

Prototype

PadStackLayer as PadStackLayer

Argument

None

Pad.Parent

This property returns the parent of the object.

Prototype

Parent as Document

Argument

None

Pad.Shape

This property returns the pad shape.

Prototype

Shape as PPcbPadShape

Argument

None

Pad.Width

This property returns the pad's width. This property should be used for the following pad shapes: ppcbPadShapeOvalFinger, ppcbPadShapeRectangularFinger, ppcbPadShapeSquare.

Prototype

Width (*unit* as PPcbUnit) as Double

Argument

unit - [Optional]	Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

PadStackLayer.AntiPad

This property returns the AntiPad Object assigned to this padstack layer or Nothing if the antipad is not defined.

Prototype

AntiPad as AntiPad

Argument

None

PadStackLayer.Application

This property returns the application object.

Prototype

Application as Application

Argument

None

PadStackLayer.Name

This property returns the name of this padstack layer.

Prototype

Name as String

Argument

None

PadStackLayer.Number

This property returns the number of this padstack layer. Values -2, -1, 0 correspond to PPcbPadStackLayerType.

Prototype

Number as Integer

Argument

None

PadStackLayer.ObjectType

This property returns the type of the object - ppcbObjectTypePadStackLayer.

Prototype

ObjectType as PPcbObjectType

Argument

None

PadStackLayer.Pad

This property returns the Pad Object assigned to this padstack layer.

Prototype

Pad as Pad

Argument

None

PadStackLayer.Parent

This property returns the parent of the object.

Prototype

Parent as Document

Argument

None

PadStackLayer.Pin

This property returns the Pin Object to which this padstack layer belongs to. If padstack layer belongs to via then Nothing is returned.

Prototype

Pin as Pin

Argument

None

PadStackLayer.ThermalPad

This property returns the ThermalPad Object assigned to this padstack layer or Nothing if thermal pad is not defined.

Prototype

ThermalPad as ThermalPad

Argument

None

PadStackLayer.Via

This property returns the Via Object to which this padstack layer belongs to. If padstack layer belongs to pin then Nothing is returned.

Prototype

Via as Via

Argument

None

PartType.Application

This property returns the PADS Layout Application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as a PADS Layout automation object. All automation server applications have an application object and all automation objects have an application property. This property is usually used in large automation client applications that handle large volumes of objects from different sources, such as different automation server applications, to quickly identify the application to which the object belongs.

PartType.Attributes

This property returns the collection of all attributes of the part type.

Prototype

Attributes As Attributes

Attributes(*name* As String) As Attribute

Argument

name	Name of an existing part type attribute.

Comments

When an existing attribute *name* is passed to this property, it returns that part type Attribute object. Otherwise, it returns the collection of all part type attributes in an Attributes collection object.

Sample

The following sample code retrieves the number of attributes on part type 7400, assuming it exists in the open design, using the Attributes.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set attrs = ActiveDocument.PartTypes("7400").Attributes
MsgBox "There are " & attrs.Count & " attribute(s) in part type 7400."
End Sub
```

PartType.Components

Returns object collection of all components of this part type.

Prototype

Components As Objects

Components(*name* As String) As Component

Argument

name	Name of an existing component.

Comments

When an existing pin *name* is passed to this property, it returns that Component object. Otherwise, it returns the collection of all components of the part type as an Objects collection object.

Sample

The following sample displays total number of 7400 parts.

```
Sub Main
MsgBox Str(ActiveDocument.PartTypes("7400").Components.Count)
End Sub
```

PartType.ECORegistered

This property sets or returns the part type ECO registration status.

Prototype

ECORegistered As Boolean

Arguments

None

Comments

This property replaces Component.PartTypeECORegistered

PartType.Logic

This property returns the logic family of the part type.

Prototype

Logic As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the logic family of the part type 7400, assuming it exists in the open schematic. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "The Logic family of 7400 is " &
ActiveDocument.PartTypes("7400").Logic
End Sub
```

PartType.Name

This property returns the name of the part type. For example, this property returns the string "7402" for part type 7402.

Prototype

Name As String

Arguments

None

Comments

This property is the default property for the Component object.

Sample

The following sample code retrieves all part types in the open schematic and places that list in a custom dialog list box. When a part type is selected in the list box, the sample selects all parts of that part type in PADS Layout. This sample uses the UserDialog Editor in the Sax Basic Engine in PADS Layout. See" Running Code Samples" for more information on running this sample.

See also: Sax Basic Editor On Line Help (C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\sbe5_000.hlp)

```
Dim ListPkgs$(10000)
Sub Main
index = 0
For Each nextPkg In ActiveDocument.PartTypes
ListPkgs$(index) = nextPkg.Name
index = index + 1
Next nextPkg
' This piece of code is automatically generated by the Basic Dialog Editor
in PADS Layout.
Begin Dialog UserDialog 180,238,"Part Types",.CallbackFunc '
%GRID:10,7,1,1
ListBox 10,7,160,203,ListPkgs(),.ListBox1
OKButton 10,210,160,21
End Dialog
Dim dlg As UserDialog
Dialog dlg
End Sub
' The following function is automatically called by the system when
something has happened
' in the dialog; it is used to easily process user actions.
Function CallbackFunc%(DlgItem$, Action%, SuppValue%)
Select Case Action%
Case 2 ' Value changing or button pressed
If DlgItem$ = "ListBox1" Then
ActiveDocument.SelectObjects(ppcbObjectTypeAll, , False)
'get part by name
Set pkg = ActiveDocument.PartTypes(ListPkgs(SuppValue%))
'select part
```

```
pkg.Selected = True
'activate sheet where first gate of the part is located
pkg.Components(1).Gates(1).Sheet.Activate
End If
End Select
End Function
```

PartType.ObjectType

This property returns the type of the object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property returns ppcbObjectTypePartType.

All PADS Layout database objects in the PADS Layout Automation server implement this property to compensate for the lack of Visual Basic equivalent for the Visual C++® QueryInterface function.

This property is generally used:

To identify the kind of PADS Layout database objects in a heterogeneous Objects collection.

When implementing a generic routine that depends on the type of the PADS Layout database object passed as argument. For example:

```
Sub DoSomething(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeComponent
' Do something specific to component objects
Case ppcbObjectTypeNet
' Do something specific to net objects
Case ppcbObjectTypePin
' Do something specific to pin objects
Case ppcbObjectTypePartType
' Do something specific to part type objects
Case Else
' Do something about other PADS Layout objects
End Select
End Sub
```

PartType.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

None

PartType.Selected

This property sets or returns whether the components of the part type are selected.

Prototype

Selected As Boolean

Arguments

None

Comments

The part type is considered selected when one or more components of this part type are selected. You can also select a PADS Layout database object using the Document.SelectObjects or Objects.Select methods.

Sample

The following sample code selects part type 7400 only, assuming it exists in the open schematic, and activates the sheet on which it resides. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.SelectObjects(,,False)
ActiveDocument.PartTypes("7400").Selected = True
End Sub
```

See Also

Document.SelectionChange Event

Pin.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an automation object. All automation server applications have an application object and all automation objects have an application property. This property is usually used in automation client applications that handle large volumes of objects from different sources, such as different automation server applications, to quickly identify the application to which the object belongs.

Pin.Attributes

This property returns the collection of all attributes of the pin.

Prototype

Attributes As Attributes

Attributes(*name* As String) As Attribute

Argument

name Name of an existing pin attribute.

Comments

When an existing attribute name is passed to this property, it returns that pin Attribute object. If the attribute name does not exist, this property returns the collection of all pin attributes in an Attributes collection object.

Sample

The following sample code retrieves the number of attributes assigned to pin U1.1, assuming it exists in the open design, using the Attributes.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set attrs = ActiveDocument.Pins("U1.1").Attributes
MsgBox "There are " & attrs.Count & " attribute(s) in pin U1.1."
End Sub
```

Pin.Component

This property returns the component owning the pin.

Prototype

Component As Component

Arguments

None

Comments

None

Sample

The following sample code retrieves the component to which pin U1.1 belongs, assuming U1.1 exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "Pin U1.1 belongs to component " &
ActiveDocument.Pins("U1.1").Component.Name
End Sub
```

Pin.DrillSize

This property returns the drill size of the pin.

Prototype

DrillSize([*unit* As PPcbUnit]) As Double

Argument

unit	[Optional] Unit in which the drill size value is returned.

Comments

This property returns 0.0 for SMD pins.

Sample

The following sample code retrieves the drill size of pin U1.1, assuming U1.1 exists in the open design. See “Running Code Samples” or more information on running this sample.

```
Sub Main
MsgBox "Pin U1.1 drill size is " & ActiveDocument.Pins("U1.1").DrillSize
End Sub
```

See Also

Pin.IsSMD

Pin.ElectricalType

This property returns the gate type of the pin.

Prototype

ElectricalType As PPcbPinElectricalType

Arguments

None

Comments

None

Sample

The following sample code retrieves the electrical type of pin U1, assuming U1 exists in the open design. See “Running Code Samples” for more information on running this sample.

```
Function ElectricalTypeName(theType As Long) As String
Select Case theType
Case ppcbElectricalTypeUnknown
ElectricalTypeName = "unknown"
Case ppcbElectricalTypeSource
ElectricalTypeName = "source"
Case ppcbElectricalTypeBidirectional
ElectricalTypeName = "bidirectional"
Case ppcbElectricalTypeOpenCollector
ElectricalTypeName = "open collector"
Case ppcbElectricalTypeOrTieableSource
ElectricalTypeName = "tiable source"
Case ppcbElectricalTypeTristate
ElectricalTypeName = "tristate"
Case ppcbElectricalTypeLoad
ElectricalTypeName = "load"
Case ppcbElectricalTypeTerminator
ElectricalTypeName = "terminator"
Case ppcbElectricalTypePower
ElectricalTypeName = "power"
Case ppcbElectricalTypeGround
ElectricalTypeName = "ground"
Case Else
ElectricalTypeName = "unknown"
End Select
End Function
Sub Main
MsgBox "Pin U1.1 electrical type is " &
ElectricalTypeName(ActiveDocument.Pins("U1.1").ElectricalType)
End Sub
```

Pin.FunctionName

This property returns the function name of the pin.

Prototype

FunctionName As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the function name of pin U1, assuming U1 exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "Pin U1.1 function name is " &
ActiveDocument.Pins("U1.1").FunctionName
End Sub
```

Pin.Glued

This property sets or returns whether the pin is glued.

Prototype

Glued As Boolean

Arguments

None

Comments

None

Sample

The following sample code glues all pins in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each nextPin In ActiveDocument.Pins
nextPin.Glued = True
Next nextPin
End Sub
```

Pin.Highlighted

This property returns whether the pin is highlighted.

Prototype

Highlighted as Boolean

Argument

None

Pin.IsSMD

This property returns whether the pin is SMD.

Prototype

IsSMD As Boolean

Arguments

None

Comments

None

Sample

The following sample code retrieves the number of SMD pins in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
nbSMDPins = 0
For Each nextPin In ActiveDocument.Pins
If nextPin.IsSMD = True Then nbSMDPins = nbSMDPins +1
Next nextPin
MsgBox "There are " & nbSMDPins & " SMD pins (out of " &
ActiveDocument.Pins.Count & ") in " & ActiveDocument.Name
End Sub
```

See Also

Pin.DrillSize

Pin.Name

This property returns the name of the pin.

Prototype

Name As String

Arguments

None

Comments

For example, this property returns the string “U1.1” for pin U1.1.

This property is the default property for the Pin object.

Sample

The following sample code lists all pins in the open design by name and then places that list in a custom dialog box. When a pin is selected in the list box, the sample selects that pin.

This sample uses the UserDialog Editor in the Sax Basic Engine in PADS Layout. See “Running Code Samples” for more information on running this sample.

See also: Sax Basic Editor On Line Help (C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\sbe5_000.hlp)

```
Dim ListPins$(10000)
Sub Main
index = 0
For Each nextPin In ActiveDocument.Pins
ListPins$(index) = nextPin.Name
index = index + 1
Next nextPin
' This piece of code is automatically generated by the Basic Dialog Editor
in PADS Layout.
Begin Dialog UserDialog 180,238,"Pins",.CallbackFunc ' %GRID:10,7,1,1
ListBox 10,7,160,203,ListPins(),.ListBox1
OKButton 10,210,160,21
End Dialog
Dim dlg As UserDialog
Dialog dlg
End Sub
' The following function is automatically called by the system when
something has happened
' in the dialog; it is used to easily process user actions.
Function CallbackFunc%(DlgItem$, Action%, SuppValue%)
Select Case Action%
Case 2 ' Value changing or button pressed
If DlgItem$ = "ListBox1" Then
ActiveDocument.SelectObjects(ppcbObjectTypeAll, , False)
ActiveDocument.SelectObjects(ppcbObjectTypePin, ListPins(SuppValue%))
```

```
End If
End Select
End Function
```

Pin.Net

This property returns the net connected to the pin.

Prototype

Net As Net

Arguments

None

Comments

This property returns nothing if the pin is not connected.

Sample

The following sample code identifies the net connected to pin U1.1, assuming U1.1 exists in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
MsgBox "Pin U1.1 is connected to net " &
ActiveDocument.Pins("U1.1").Net.Name
End Sub
```

Pin.Number

This property returns the number of the pin.

Prototype

Number As Long

Arguments

None

Comments

None

Sample

The following sample code retrieves the pin number of pin U1.1, assuming U1.1 exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
MsgBox "Pin U1.1 is pin number " & ActiveDocument.Pins("U1.1").Number & "
in component " & ActiveDocument.Pins("U1.1").Component.Name
End Sub
```

Pin.ObjectType

This property returns the type of the object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property returns ppcbObjectTypePin.

All database objects in the automation server implement this property to compensate for the lack of a Visual Basic equivalent for the Visual C++ QueryInterface function.

This property is generally used:

- To identify the kind of database objects in a heterogeneous Objects collection.
- When implementing a generic routine that depends on the type of the database object passed as argument. For example:

```
Sub DoSomething(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeComponent
' Do something specific to component objects
Case ppcbObjectTypeNet
' Do something specific to net objects
Case ppcbObjectTypePin
' Do something specific to pin objects
Case ppcbObjectTypeVia
' Do something specific to via objects
Case ppcbObjectTypeConnection
' Do something specific to connection objects
Case ppcbObjectTypeRouteSegment
' Do something specific to route segment objects
Case ppcbObjectTypeJumper
' Do something specific to jumper objects
Case Else
MsgBox "Not a PADS Layout database object"
End Select
End Sub
```

Pin.PadStackLayers

This property returns the collection of all padstack layers for this pin.

Prototype

PadStackLayers as Objects

PadStackLayers(*layerName* as String) as PadStackLayer

Argument

layerName	Name of padstack layer

Return Values

When a layer name is passed to this property, it returns that PadStackLayer Object. If the name is not specified, this property returns the collection of all padstack layers in an Objects collection object.

Sample

```
For Each comp In Application.ActiveDocument.Components
  For Each pin In comp.Pins
    For Each layer in pin.PadStackLayers

      MsgBox layer.Number & ", " & layer.Name

      pad = layer.Pad
      MsgBox pad.Name & ", " & pad.Shape & ", " & pad.Diameter & ", "
             & pad.InnerDiameter & ", " & pad.Width & ", "
             & pad.Length & ", " & pad.offset & ", "
             & pad.Orientation & ", " & pad.CornerType & ", "
             & pad.CornerRadius

      thermalpad = layer.ThermalPad
      If thermalpad Is Nothing Then
        MsgBox "Thermal pad not defined on this layer"
      Else
        MsgBox thermalpad.Name & ", " & thermalpad.Shape & ", "
               & thermalpad.InnerSize & ", "
               & thermalpad.OuterSize & ", " & thermalpad.Spokes & ", "
               & thermalpad.SpokeAngle  & ", " & thermalpad.SpokeWidth
      End If

      antipad = layer.AntiPad
      If antipad Is Nothing Then
        MsgBox "Anti pad not defined on this layer"
      Else
        MsgBox antipad.Name & ", " & antipad.Shape & ", "
               & antipad.Size
      End If

    Next layer
  Next pin
```

```
Next comp
```

Pin.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

None

Pin.PlaneThermal

This property returns whether the pin has a plane thermal.

Prototype

PlaneThermal As Boolean

Arguments

None

Comments

None

Sample

The following sample code retrieves the number of plane thermal pins in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
nbPTPins = 0
For Each nextPin In ActiveDocument.Pins
If nextPin.PlaneThermal = True Then nbPTPins = nbPTPins +1
Next nextPin
MsgBox "There are " & nbPTPins & " plane thermal pins (out of " &
ActiveDocument.Pins.Count & ") in " & ActiveDocument.Name
End Sub
```

Pin.Plated

This property returns whether the pin is plated.

Prototype

Plated As Boolean

Arguments

None

Comments

None

Sample

The following sample code retrieves the number of non-SMD, nonplated pins in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
nbPlatedPins = 0
For Each nextPin In ActiveDocument.Pins
If nextPin.Plated = False And nextPin.IsSMD = False Then nbPlatedPins =
nbPlatedPins +1
Next nextPin
MsgBox "There are " & nbPlatedPins & " non-SMD non-plated pins (out of " &
ActiveDocument.Pins.Count & ") in " & ActiveDocument.Name
End Sub
```

Pin.PositionX

This property returns the x-coordinate of the pin.

Prototype

PositionX([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the x-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the location of pin U1.1, assuming U1.1 exists in the open design, in current design units. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set pinU1_1 = ActiveDocument.Pins("U1.1")
MsgBox "U1.1 position is = (" & pinU1_1.PositionX & ", " &
pinU1_1.PositionY & ")"
End Sub
```

See Also

Pin.PositionY

Pin.PositionY

This property returns the y-coordinate of the pin.

Prototype

PositionY([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the y-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the location of the pin U1.1, assuming U1.1 exists in the open design, in current design units. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set pinU1_1 = ActiveDocument.Pins("U1.1")
MsgBox "U1.1 position is = (" & pinU1_1.PositionX & ", " &
pinU1_1.PositionY & ")"
End Sub
```

See Also

Pin.PositionX

Pin.Selected

This property sets or returns whether the pin is selected.

Prototype

Selected As Boolean

Arguments

None

Comments

You can also select a PADS Layout database object using the Document.SelectObjects and Objects.Select methods.

Sample

The following sample code selects pin U1.1 only, assuming U1.1 exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.SelectObjects(,,False)
ActiveDocument.Pins("U1.1").Selected = True
End Sub
```

See Also

Document.SelectionChange Event

Pin.SlotLength

This property returns the slot length of the pin.

Prototype

SlotLength (*Unit* as PPcbUnit) as Double

Argument

unit [Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.

Comments

If a pin is not slotted, it returns zero. The property could be used to check if a pin has a slot or round drill.

Sample

The following sample code displays the slot length of the selected pin. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypePin,,True)
For Each pin In selected
MsgBox "Slot length:" & pin.SlotLength
Exit For
Next pin
End Sub
```

Pin.SlotOffset

This property returns the slot offset of the pin.

Prototype

SlotOffset (*Unit* as PPcbUnit) as Double

Argument

unit [Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.

Comments

If pin is not slotted, it returns zero.

Sample

The following sample code displays the slot length of the selected pin. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypePin,,True)
For Each pin In selected
MsgBox "Slot offset:" & pin.SlotOffset
Exit For
Next pin
End Sub
```

Pin.SlotOrientation

This property returns the slot orientation of the pin, in degrees.

Prototype

SlotOffset as Double

Arguments

None

Comments

If pin is not slotted, it returns zero.

Sample

The following sample code displays the slot orientation of the selected pin. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypePin,,True)
For Each pin In selected
MsgBox "Slot orientation:" & pin.SlotOrientation
Exit For
Next pin
End Sub
```

Pin.TestPoint

This property sets or returns whether the pin is a test point.

Prototype

TestPoint As PPcbTestPointType

Arguments

None

Comments

None

Sample

The following sample code removes all test points from the open design and adds one top layer test point per net, arbitrarily choosing a pin on the net for the test point. See "Running Code Samples" for more information on running this sample.

```
Sub Main
' Remove all test points
For Each nextPin In ActiveDocument.Pins
nextPin.TestPoint = ppcbTestPointNone
Next nextPin
' Add one top layer test point per net
For Each nextNet In ActiveDocument.Nets
Set arbitPin = nextNet.Pins.Item(0)
arbitPin.TestPoint = ppcbTestPointTopLayer
Next nextNet
End Sub
```

Polyline.Application

This property returns the Application object.

Prototype

Application as Application

Arguments

None

Comments

This is a Microsoft-required property.

Polyline.CenterX

This property returns a center x-coordinate for an arc in the polyline.

Prototype

CenterX (*Corner* as Long, *Unit* as PPcbUnit, *Origin* as PPcbOriginType) as Double

Arguments

corner	Starting corner for the arc.
unit	[Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.
Origin	[Optional]Type of reference point from which the result is counted. The default value is ppcbOriginTypeDesign.

Comments

None

Sample

The following sample code displays center coordinates of the selected polyline. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,True)
For Each drw In selected
Dim geom As Object
For Each geom In drw.Geometry
If geom.ObjectType = ppcbObjectTypePolyline Then
p = geom.Points
n = UBound(p, 1)
For i = 1 To n
If p(i, 3) <> 0 Then
m = "Arc " & i & " : (" & p(i, 1) & ", " & p(i, 2) & ") "
m = m & "Center: ("& geom.CenterX(i) & ", " & geom.CenterY(i) & ")"
MsgBox m
End If
Next i
Exit For
End If
Next geom
Exit For
Next drw
End Sub
```

Polyline.CenterY

This property returns a center y-coordinate for an arc in the polyline.

Prototype

CenterY (*Corner* as Long, *Unit* as PPcbUnit, *Origin* as PPcbOriginType) as Double

Arguments

corner	Starting corner for the arc.
unit	[Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.
origin	[Optional]Type of reference point from which the result is counted. The default value is ppcbOriginTypeDesign.

Comments

None

Sample

The following sample code displays center coordinates of the selected polyline. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,True)
For Each drw In selected
Dim geom As Object
For Each geom In drw.Geometry
If geom.ObjectType = ppcbObjectTypePolyline Then
p = geom.Points
n = UBound(p, 1)
For i = 1 To n
If p(i, 3) <> 0 Then
m = "Arc " & i & " : (" & p(i, 1) & ", " & p(i, 2) & ") "
m = m & "Center: ("& geom.CenterX(i) & ", " & geom.CenterY(i) & ")"
MsgBox m
End If
Next i
Exit For
End If
Next geom
Exit For
Next drw
End Sub
```

Polyline.Geometry

This property returns a collection of objects, currently, polylines or circles, representing this object's child geometry objects.

Prototype

Geometry as Collection

Arguments

None

Comments

None

Sample

The following sample code shows the number of child objects. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
For Each geom In drw.Geometry
MsgBox "Child object count: " & geom.Geometry.Count
Next geom
Next drw
End Sub
```

Polyline.Layer

This property returns the layer number of the object

Prototype

Layer as Long

Arguments

None

Comments

None

Sample

The following sample code shows the layer number of the selected polyline. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
MsgBox "Layer number: " & geom.Layer
Next geom
Next drw
End Sub
```

Polyline.LineWidth

This property returns the width of the polyline.

Prototype

LineWidth (*Unit* as PPcbUnit) as Double

Argument

unit	[Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.

Comments

None

Sample

The following sample code shows the line width of the selected polyline. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
For Each geom In drw.Geometry
MsgBox "Line width: " & geom.LineWidth
Next geom
Next drw
End Sub
```

Polyline.ObjectType

This property returns the type of object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Sample

The following sample code tests the ObjectType property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each drw In ActiveDocument.Drawings
For Each geom In drw.Geometry
t = geom.ObjectType
If t <> pcbObjectTypePolyline And t <> ppcbObjectTypeCircle Then
MsgBox "Test failed"
End If
Next geom
Next drw
End Sub
```

Polyline.OutlineType

This property returns the outline type of the polyline.

Prototype

OutlineType as PPcbOutlineType

Arguments

None

Comments

None

Sample

The following sample code shows the outline type of the selected polyline. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
Select Case geom.OutlineType
Case ppcbOutLineTypeCenter
s = "Center line"
Case ppcbOutLineTypeOuter
s = "Outer line"
Case ppcbOutLineTypeInner
s = "Inner line"
End Select
MsgBox "Outline type: " & s
Next geom
Next drw
End Sub
```

Polyline.Parent

This property returns the parent of the object.

Prototype

Parent as Document

Arguments

None

Comments

This is a Microsoft-required property.

Polyline.Points

This property returns coordinates of the polyline points.

Prototype

Points (*Unit* as PPcbUnit, *Origin* as PPcbOriginType) as Variant

Arguments

unit	[Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.
origin	[Optional]Type of reference point from which the result is counted. The default value is ppcbOriginTypeDesign.

Return Values

A two-dimensional array of coordinates representing ends of the polyline's segments. Points (n, 1) are the X coordinate and points (n, 2) are the Y coordinate. Points (n, 3) contain the arc angle, if the point n and point n + 1 are connected with an arc, or zero. The angle is positive if the arc has counterclockwise direction, and negative when the arc has clockwise direction. The angle is measured in degrees.

Comments

For shape types Hollow, Filled, and Void, the last point duplicates the first one. For example, points array dimensions of a filled rectangle object (5, 3). The angle for the last point is always 0.

Sample

The following sample code displays the corner coordinates of the selected polyline. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,True)
For Each drw In selected
  For Each geom In drw.Geometry
  If geom.ObjectType = ppcbObjectTypePolyline Then
  p = geom.Points
n = UBound(p, 1)

  For i = 1 To n - 1
   If p(i, 3) = 0 Then
   m = "Segment " & i
   m = m & " From (" & p(i, 1) & ", " & p(i, 2) & ")"
   m = m & " To (" & p(i + 1, 1) & ", " & p(i + 1, 2) & ")"
   Else
   m = "Arc " & i
   m = m & " From (" & p(i, 1) & ", " & p(i, 2) & ")"
   m = m & " To (" & p(i + 1, 1) & ", " & p(i + 1, 2) & ")"
   m = m & " Angle: " & p(i, 3)
```

```
      End If
      MsgBox m
      Next i
     Exit For
  End If
   Next geom
   Exit For
  Next drw
  End Sub
```

Polyline.Radius

This property returns a radius for an arc in the polyline.

Prototype

Radius (*Corner* as Long, *Unit* as PPcbUnit) as Double

Arguments

corner	Starting corner for the arc.
unit	[Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.

Comments

None

Sample

The following sample code displays the arc radii of the selected polyline. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,True)
For Each drw In selected
Dim geom As Object
For Each geom In drw.Geometry
If geom.ObjectType = ppcbObjectTypePolyline Then
p = geom.Points
n = UBound(p, 1)
For i = 1 To n
If p(i, 3) <> 0 Then
m = "Arc " & i & " : (" & p(i, 1) & ", " & p(i, 2) & ") "
m = m & "Radius: " & geom.Radius(i)
MsgBox m
End If
Next i
Exit For
End If
Next geom
Exit For
Next drw
End Sub
```

Polyline.ShapeType

This property returns the shape type of the polyline.

Prototype

ShapeType as PPcbShapeType

Arguments

None

Comments

None

Sample

The following sample code shows the shape type of the selected polyline. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeDrawing,,TRUE)
For Each drw In selected
Dim geom as Object
For Each geom In drw.Geometry
Select Case geom.ShapeType
Case ppcbShapeTypeOpen
s = "Open"
Case ppcbShapeTypeHollow
s = "Hollow"
Case ppcbShapeTypeFilled
s = "Filled"
Case ppcbShapeTypeVoid
s = "Void"
End Select
MsgBox "Polyline type: " & s & " shape"
Next geom
Next drw
End Sub
```

RouteSegment.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an Automation object. All Automation server applications have an Application object and all Automation objects have an Application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

RouteSegment.Layer

This property returns the mounting layer of the trace segment.

Prototype

Layer As Long

Arguments

None

Comments

None

Sample

The following sample code retrieves the layer on which the first routed trace segment found in the open design exists, assuming at least one trace segment exists. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set firstRteSeg = ActiveDocument.RouteSegments(1)
MsgBox "Route segment " & firstRteSeg.Name & " is on layer " &
firstRteSeg.Layer
End Sub
```

RouteSegment.Length

This property returns the length of the trace segment.

Prototype

Length([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the length value is returned.

Comments

None

Sample

The following sample code retrieves the length of the first trace segment found in the open design, assuming at least one trace segment exists. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set firstRteSeg = ActiveDocument.RouteSegments(1)
MsgBox "Route segment " & firstRteSeg.Name & " length is " &
firstRteSeg.Length
End Sub
```

RouteSegment.Name

This property returns the name of the trace segment.

Prototype

Name As String

Arguments

None

Comments

This property is the default property for the RouteSegment object.

Sample

The following sample code lists all trace segments in the open design by name and then places that list in a custom dialog box. When a trace segment is selected in the list box, the sample selects that trace segment.

This sample uses the UserDialog Editor in the Sax Basic Engine. See "Running Code Samples" for more information on running this sample.

See also: Sax Basic Editor On Line Help (C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\sbe5_000.hlp)

```
Dim ListRteSegs$(10000)
Sub Main
index = 0
For Each nextRteSeg In ActiveDocument.RouteSegments
ListRteSegs$(index) = nextRteSeg.Name
index = index + 1
Next nextRteSeg
' This piece of code is automatically generated by the Basic Dialog Editor
in PADS Layout.
Begin Dialog UserDialog 180,238,"Route Segments",.CallbackFunc '
%GRID:10,7,1,1
ListBox 10,7,160,203,ListRteSegs(),.ListBox1
OKButton 10,210,160,21
End Dialog
Dim dlg As UserDialog
Dialog dlg
End Sub
' The following function is automatically called by the system when
something has happened
' in the dialog; it is used to easily process user actions.
Function CallbackFunc%(DlgItem$, Action%, SuppValue%)
Select Case Action%
Case 2 ' Value changing or button pressed
If DlgItem$ = "ListBox1" Then
ActiveDocument.SelectObjects(ppcbObjectTypeAll, , False)
```

```
ActiveDocument.SelectObjects(ppcbObjectTypeRouteSegment,
ListRteSegs(SuppValue%))
End If
End Select
End Function
```

RouteSegment.Net

This property returns the net connected to the trace segment.

Prototype

Net As Net

Arguments

None

Comments

None

Sample

The following sample code retrieves the net of the first trace segment found in the open design, assuming at least one trace segment exists. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set firstRteSeg = ActiveDocument.RouteSegments(1)
MsgBox "Route segment " & firstRteSeg.Name & " is connected on net " &
firstRteSeg.Net.Name
End Sub
```

RouteSegment.ObjectType

This property returns the type of the object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property returns ppcbObjectTypeRouteSegment.

All database objects in the Automation server implement this property to compensate for the lack of a Visual Basic equivalent for the Visual C++ QueryInterface function.

This property is generally used:

- To identify the kind of PADS Layout database objects in a heterogeneous Objects collection.
- When implementing a generic routine that behavior depends on the type of database object passed as argument. For example:

```
Sub DoSomething(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeComponent
' Do something specific to component objects
Case ppcbObjectTypeNet
' Do something specific to net objects
Case ppcbObjectTypePin
' Do something specific to pin objects
Case ppcbObjectTypeVia
' Do something specific to via objects
Case ppcbObjectTypeConnection
' Do something specific to connection objects
Case ppcbObjectTypeRouteSegment
' Do something specific to route segment objects
Case ppcbObjectTypeJumper
' Do something specific to jumper objects
Case Else
MsgBox "Not a PADS Layout database object"
End Select
End Sub
```

RouteSegment.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

None

RouteSegment.Points

This property returns the array of points defining the trace segment.

Prototype

Points([*unit* As PPcbUnit]) AsVariant

Argument

unit	[Optional] Unit in which the point coordinate values are returned.

Comments

None

RouteSegment.SegmentType

This property returns the type of the trace segment.

Prototype

SegmentType As PPcbSegmentType

Arguments

None

Comments

None

Sample

The following sample code retrieves the segment type of the first trace segment found in the open design, assuming at least one trace segment exists. See "Running Code Samples" for more information on running this sample.

```
Function SegmentTypeName(theType As Long) As String
Select Case theType
Case ppcbSegmentUnknown
SegmentTypeName = "unknown"
Case ppcbSegmentLine
SegmentTypeName = "line"
Case ppcbSegmentArc
SegmentTypeName = "arc"
Case Else
SegmentTypeName = "unknown"
End Select
End Function
Sub Main
Set firstRteSeg = ActiveDocument.RouteSegments(1)
MsgBox "Route segment " & firstRteSeg.Name & " is of type " &
SegmentTypeName(firstRteSeg.SegmentType)
End Sub
```

RouteSegment.Selected

This property sets or returns whether the trace segment is selected.

Prototype

Selected As Boolean

Arguments

None

Comments

None

Sample

The following sample code selects the first trace segment found in the open design, assuming at least one trace segment exists. See “Running Code Samples” for more information on running this sample.

```
Sub Main
ActiveDocument.SelectObjects(,,False)
ActiveDocument.RouteSegments(1).Selected = True
End Sub
```

See Also

Document.SelectionChange Event

RouteSegment.Width

This property returns the width of the trace segment or the width of draw lines used to fill copper items such a split planes or copper pours.

Prototype

Width([*unit* As PPcbUnit]) As Double

Arguments

Argument	**Description**
unit	[Optional] Unit in which the width value is returned.

Comments

None

Sample

The following sample code retrieves the width of the first trace segment found in the open design, assuming at least one trace segment exists. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set firstRteSeg = ActiveDocument.RouteSegments(1)
MsgBox "Route segment " & firstRteSeg.Name & " width is " &
firstRteSeg.Width
End Sub
```

SBP.Application

This property returns the Application object.

Restriction: This information applies to only the BGA toolkit.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an Automation object. All Automation server applications have an application object and all Automation objects have an application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

This is a Microsoft-required property.

SBP.CBPs

This property returns the collection of CBPs linked to the SBP object.

Restriction: This information applies to only the BGA toolkit.

Prototype

CBPs As Objects

Arguments

None

Comments

None

Sample

The following sample code retrieves the number of CBPs connected to each SBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox "Number of CBPs linked to " & aSBP.Name & ": " & aSBP.CBPs.Count
Next
End If
Next
End Sub
```

See Also

SBP.Wirebonds

SBP.Component

This property returns the Component object of the SBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Component As Component

Arguments

None

Comments

None

Sample

The following sample code represents a subroutine that depends on the SBP object passed as an argument. The code retrieves the name of the component and the name of the decal to which the SBP belongs. See “Running Code Samples” for more information on running this sample.

```
Sub QuerySBP(aSBP As SBP)
MsgBox "Substrate Bond Pad" & aSBP.Name &
"belongs to component" & aSBP.Component.Name &
"(" & aSBP.Component.Decal &")"
End Sub
```

SBP.Function

This property returns the function of the SBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Function As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the function of each SBP in each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox aSBP.Name & " Function: " & aSBP.Function
Next
End If
Next
End Sub
```

SBP.Layer

This property returns the LIQ layer of the SBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Layer As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the layer of each SBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox aSBP.Name & " Layer: " & aSBP.Layer
Next
End If
Next
End Sub
```

SBP.Length

This property returns the length of the SBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Length([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the length value is returned.

Comments

None

Sample

The following sample code retrieves the length of each SBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox aSBP.Name & " Length: " & Format (aSBP.Length, "#.###")
Next
End If
Next
End Sub
```

See Also

SBP.Width

SBP.Name

This default property returns the name of the SBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Name As String

Arguments

None

Comments

This is a Microsoft-required property.

Sample

The following sample code retrieves the name of each SBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox aSBP.Name
Next
End If
Next
End Sub
```

SBP.ObjectType

This property returns the type of the object.

Restriction: This information applies to only the BGA toolkit.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property always returns ppcbObjectTypeSBP.

This property is generally used to identify the kind of database objects in a heterogeneous Objects collection or when implementing a generic routine that depends on the type of the database object passed as argument.

All database objects in the Automation server implement this property to compensate for the lack of an equivalent for the Visual C++ QueryInterface function.

Sample

See "Running Code Samples" for more information on running this sample.

```
Sub DoSomethingToDieObject(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeCBP
' Do something specific to CBP objects
Case ppcbObjectTypeSBP
' Do something specific to SBP objects
Case ppcbObjectTypeWirebond
' Do something specific to Wirebond objects
Case Else
MsgBox "Not a Die object"
End Select
End Sub
```

SBP.Orientation

This property returns SBP orientation, in degrees.

Restriction: This information applies to only the BGA toolkit.

Prototype

Orientation As Double

Arguments

None

Comments

None

Sample

The following sample code retrieves the orientation of each SBP in each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox aSBP.Name & " Orientation: " & Format (aSBP.Orientation, "#.###")
Next
End If
Next
End Sub
```

SBP.Parent

The property returns the parent of the object.

Restriction: This information applies to only the BGA toolkit.

Prototype

Parent As Document

Arguments

None

Comments

This is a Microsoft-required property.

SBP.Position X

This property returns the x-coordinate of the SBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

PositionX([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the x-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the position of each SBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox aSBP.Name & ": (" & aSBP.PositionX & ", " & aSBP.PositionY & ")"
Next
End If
Next
End Sub
```

See Also

SBP.PositionY

SBP.PositionY

This property returns the y-coordinate of the SBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

PositionY([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the y-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the position of each SBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox aSBP.Name & ": (" & aSBP.PositionX & ", " & aSBP.PositionY & ")"
Next
End If
Next
End Sub
```

See Also

SBP.Position X

SBP.Shape

This property returns the shape of the SBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Shape As PPcbBondPadShape

Arguments

None

Comments

None

Sample

The following sample code retrieves the shape of each SBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
Select Case aSBP.Shape
Case ppcbBondPadShapeUnknown
MsgBox aSBP.Name & " Shape: Unknown"
Case ppcbBondPadShapeRectangle
MsgBox aSBP.Name & " Shape: Rect"
Case ppcbBondPadShapeOval
MsgBox aSBP.Name & " Shape: Oval"
Case Else
MsgBox aSBP.Name & " Shape: Unknown"
End Select
Next
End If
Next
End Sub
```

SBP.Tier

This property returns the LIQ tier in which the SBP is placed.

Restriction: This information applies to only the BGA toolkit.

Prototype

Tier As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the tier for each SBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox aSBP.Name & " Tier: " & aSBP.Tier
Next
End If
Next
End Sub
```

SBP.Width

This property returns the width of the SBP.

Restriction: This information applies to only the BGA toolkit.

Prototype

Width([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the width value is returned.

Comments

None

Sample

The following sample code retrieves the width of each SBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox aSBP.Name & " Width: " & Format (aSBP.Width (ppcbUnitInch),
"#.###")
Next
End If
Next
End Sub
```

See Also

SBP.Length

SBP.Wirebonds

This property returns the collection of bond wires attached to the SBP object.

Restriction: This information applies to only the BGA toolkit.

Prototype

Wirebonds As Objects

Arguments

None

Comments

None

Sample

The following sample code retrieves the number of bond wires connected to each SBP in each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aSBP in comp.SBPs
MsgBox "Number of WBs connected to " & aSBP.Name & ":" &
aSBP.Wirebonds.Count
Next
End If
Next
End Sub
```

See Also

SBP.CBPs

Text.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This is a Microsoft-required property.

Text.Drawing

This property returns a drawing with which the text is associated.

Prototype

Drawing as Drawing

Arguments

None

Comments

None

Sample

The following sample function displays the name of a drawing with which the text is associated. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText, , TRUE)
For Each txt In selected
Dim drw as Drawing
Set drw = txt.Drawing
If Not drw Is Nothing Then
s = "The " & txt.Name & " text"
MsgBox s & " is associated with " &drw.Name & " drawing"
End If
Next txt
End Sub
```

Text.Height/Label.Height

This property returns or sets text or label height.

Prototype

Height (*Unit* as PPcbUnit) as Double

Argument

unit [Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.

Comments

None

Samples

The following example shows the height of a selected text object, and then sets the height to 100 mils. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
MsgBox "Selected text height: " & text.Height
text.Height(ppcbUnitMils) = 100
Next text
End Sub
```

The following example shows the height of selected label object, and then sets height to 100 mils. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,, TRUE)
For Each label In selected
MsgBox "Selected label height: " & label.Height
label.Height(ppcbUnitMils) = 100
Next label
End Sub
```

Text.HorzJustification/Label.HorzJustification

This property returns or sets the horizontal justification type of the text or label object.

Prototype

HorzJustification as PPcbHorizontalJustification

Arguments

None

Comments

None

Samples

The following example shows the horizontal justification setting of the selected text object, and then sets the justification to center. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
Select Case text.HorzJustification
Case ppcbJustifyLeft
MsgBox "Horizontal justification: Left"
Case pcbJustifyHCenter
MsgBox "Horizontal justification: Center"
Case pcbJustifyRight
MsgBox "Horizontal justification: Right"
End Select
text.HorzJustification = ppcbJustifyHCenter
Next text
End Sub
```

The following example shows the horizontal justification setting of the selected label object, and then sets the justification to center. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,, TRUE)
For Each label In selected
Select Case label.HorzJustification
Case ppcbJustifyLeft
MsgBox "Horizontal justification: Left"
Case ppcbJustifyHCenter
MsgBox "Horizontal justification: Center"
Case ppcbJustifyRight
MsgBox "Horizontal justification: Right"
End Select
label.HorzJustification = ppcbJustifyHCenter
Next label
End Sub
```

Text.Layer/Label.Layer

This property returns or sets the layer number of the text or label.

Prototype

Layer as Long

Arguments

None

Comments

None

Samples

The following example shows the layer number of the selected text object, and then sets the layer to 1. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
MsgBox "Selected text layer number: " & text.Layer
text.Layer = 1
Next text
End Sub
```

The following example shows the layer number of the selected label object, and then sets the layer to 1. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,, TRUE)
For Each label In selected
MsgBox "Selected label layer number: " & label.Layer
label.Layer = 1
Next label
End Sub
```

Text.LineWidth/Label.LineWidth

This property returns or sets the line width of the text or label.

Prototype

LineWidth (*Unit* as PPcbUnit) as Double

Argument

unit [Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.

Comments

None

Samples

The following example shows the line width of the selected text objects, and then sets the width to 10 mils. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
MsgBox "Selected text line width: " & text.LineWidth
text.LineWidth(ppcbUnitMils) = 10
Next text
End Sub
```

The following example shows the line width of the selected label objects, and then sets the width to 10 mils. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,, TRUE)
For Each label In selected
MsgBox "Selected label line width: " & label.LineWidth
label.LineWidth(ppcbUnitMils) = 10
Next label
End Sub
```

Text.Mirror/Label.Mirror

This property returns or sets the mirrored state of the text or label.

Prototype

Mirror(*Origin* as PPcbOriginType) as Boolean

Argument

origin Shows whether the mirror status is relative to the object's parent or to the design (optional). The default value is ppcbOriginTypeDesign.

Comments

None

Samples

The following example shows whether the selected text object is mirrored, and then mirrors the text. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
MsgBox "Is selected text mirrored? " & text.Mirror
text.Mirror = True
Next text
End Sub
```

The following example shows whether the selected label object is mirrored, and then mirrors the label. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,, TRUE)
For Each label In selected
MsgBox "Is selected label mirrored? " & label.Mirror
label.Mirror = True
Next label
End Sub
```

Text.Name

This default property returns the name of the Text object.

Prototype

Name as String

Arguments

None

Comments

None

Sample

The following example displays a message box showing the name of the first free text in the active document. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set text = ActiveDocument.Texts(1)
MsgBox "The first free text's name is " & text.Name
End Sub
```

Text.ObjectType

This property returns the type of object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

Sample

The following sample shows the number of selected texts. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set sel = ActiveDocument.GetObjects(,,TRUE)
n = 0
For Each obj In sel
If obj.ObjectType = ppcbObjectTypeText Then n = n + 1
Next obj
MsgBox n & " text(s) selected"
End Sub
```

Text.Orientation/Label.Orientation

This property returns or sets the rotation angle of the text or label.

Prototype

Orientation(*Origin* as PPcbOriginType) as Double

Argument

origin	Shows whether the orientation value is relative to the object's parent or to the design (optional). The default value is ppcbOriginTypeDesign.

Comments

None

Samples

The following example shows the orientation of the selected text object, and then sets the orientation to 90 degrees. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
MsgBox "Selected text orientation: " & text.Orientation
text.Orientation = 90
Next text
End Sub
```

The following example shows the orientation of the selected label object, and then sets the orientation to 90 degrees. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,, TRUE)
For Each label In selected
MsgBox "Selected label orientation: " & label.Orientation
label.Orientation = 90
Next label
End Sub
```

Text.Parent

This property returns the parent of this object.

Prototype

Parent As Document

Arguments

None

Comments

This is a Microsoft-required property.

Sample

Not required.

Text.PositionX/Label.PositionX

This property returns or sets the x-coordinate of the origin of the Text or Label object.

Prototype

PositionX (*Unit* as PPcbUnit, Origin as PPcbOriginType) as Double

Arguments

unit	[Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.
origin	[Optional]Type of reference point from which the result is counted. The default value is pcpbOriginTypeDesign.

Comments

None

Samples

The following example shows the position of a selected text object, and then sets the position to (200, 200) mil. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
MsgBox "Selected text position: (" & text.PositionX & ", " &
text.PositionX & ")"
text.PositionX(ppcbUnitMils) = 200
text.PositionY(ppcbUnitMils) = 200
Next text
End Sub
```

The following example shows the position of a selected label object, and then sets the position to (200, 200) mil. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,, TRUE)
For Each label In selected
MsgBox "Selected label position: (" & label.PositionX & ", " &
label.PositionX & ")"
label.PositionX(ppcbUnitMils) = 200
label.PositionY(ppcbUnitMils) = 200
Next label
End Sub
```

Text.PositionY/Label.PositionY

This property returns or sets the y-coordinate of the origin of the text or label object.

Prototype

PositionY(*Unit* as PPcbUnit, Origin as PPcbOriginType) as Double

Arguments

unit	[Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.
origin	[Optional]Type of reference point from which the result is counted. The default value is ppcbOriginTypeDesign.

Comments

None

Samples

The following example shows the position of the selected text object, and then sets position to (200, 200) mil. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
MsgBox "Selected text position: (" & text.PositionX & ", " &
text.PositionX & ")"
text.PositionX(ppcbUnitMils) = 200
text.PositionY(ppcbUnitMils) = 200
Next text
End Sub
```

The following example shows the position of the selected label object, and then sets position to (200, 200) mil. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,, TRUE)
For Each label In selected
MsgBox "Selected label position: (" & label.PositionX & ", " &
label.PositionX & ")"
label.PositionX(ppcbUnitMils) = 200
label.PositionY(ppcbUnitMils) = 200
Next label
End Sub
```

Text.Selected

This property sets or returns whether the text object is selected.

Prototype

Selected as Boolean

Arguments

None

Comments

None

Sample

The following sample displays a message indicating whether the first text is selected, then selects the text. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each txt In ActiveDocument.Texts
Msgbox "Is Text " & txt.Text & " selected ? " & txt.Selected
txt.Selected = True
Exit For
Next txt
End Sub
```

Text.Text

This property returns or sets contents of the text object.

Prototype

Text as String

Arguments

None

Comments

None

Sample

This sample shows the text string in the selected text object and then sets the string to “Hello!” See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
MsgBox "Selected text: " & text.Text
text.Text = "Hello !"
Next text
End Sub
```

Text.VertJustification/Label.VertJustification

This property returns or sets the vertical justification type of the text or label object.

Prototype

VertJustification as PPcbVerticalJustification

Arguments

None

Comments

None

Samples

The following example shows the vertical justification setting of the selected text object, then sets the justification to center. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
Select Case text.VertJustification
Case ppcbJustifyBottom
MsgBox "Vertical justification: Bottom"
Case ppcbJustifyVCenter
MsgBox "Vertical justification: Center"
Case ppcbJustifyTop
MsgBox "Vertical justification: Top"
End Select
text.VertJustification = ppcbJustifyVCenter
Next Text
End Sub
```

The following example shows the vertical justification setting of the selected label object, then sets the justification to center. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,, TRUE)
For Each label In selected
Select Case label.VertJustification
Case ppcbJustifyBottom
MsgBox "Vertical justification: Bottom"
Case ppcbJustifyVCenter
MsgBox "Vertical justification: Center"
Case ppcbJustifyTop
MsgBox "Vertical justification: Top"
End Select
label.VertJustification = ppcbJustifyVCenter
Next Label
End Sub
```

ThermalPad.Application

This property returns the application object.

Prototype

Application as Application

Argument

None

ThermalPad.CornerRadius

This property returns the Thermal Pad corner radius.

Prototype

```
CornerRadius as Double
      CornerRadius (unit as PPcbUnit)  as Double
```

Argument

unit - [Optional]   Unit in which the value is returned.

ThermalPad.CornerType

This property returns the Thermal Pad corner type.

Prototype

```
CornerType as PPcbPadCornerType
```

Argument

None

ThermalPad.InnerLength

This property returns the Thermal Pad inner length.

Prototype

```
InnerLength as Double
      InnerLength (unit as PPcbUnit)  as Double
```

Argument

unit - [Optional]	Unit in which the value is returned.

ThermalPad.InnerSize

This property returns the thermal pad's inner size. For shape ppcbThermalPadShapeRound it returns inner diameter.

Prototype

```
InnerSize (unit as PPcbUnit) as Double
```

Argument

unit - [Optional]	Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

ThermalPad.Name

This property returns the name of this thermal pad.

Prototype

```
Name as String
```

Argument

None

ThermalPad.ObjectType

This property returns the type of the object - ppcbObjectTypeThermalPad.

Prototype

```
ObjectType as PPcbObjectType
```

Argument

None

ThermalPad.Offset

This property returns the Thermal Pad offset.

Prototype

```
Offset as Double
      Offset  (unit as PPcbUnit)  as Double
```

Argument

unit - [Optional]   Unit in which the value is returned.

ThermalPad.Orientation

This property returns the Thermal Pad orientation.

Prototype

```
Orientation as Double
```

Argument

None

ThermalPad.OuterSize

This property returns the thermal pad's outer size. For shape ppcbThermalPadShapeRound it returns outer diameter.

Prototype

```
OuterSize (unit as PPcbUnit) as Double
```

Argument

unit - [Optional]	Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

ThermalPad.PadStackLayer

This property returns the PadStackLayer Object to which this thermal pad belongs to.

Prototype

```
PadStackLayer as PadStackLayer
```

Argument

None

ThermalPad.Parent

This property returns the parent of the object.

Prototype

```
Parent as Document
```

Argument

None

ThermalPad.Shape

This property returns the thermal pad shape.

Prototype

```
Shape as PPcbThermalPadShape
```

Argument

None

ThermalPad.SpokeAngle

This property returns the thermal pad's spoke angle.

Prototype

```
SpokeAngle as Double
```

Argument

None

ThermalPad.Spokes

This property returns the thermal pad's number of spokes.

Prototype

```
Spokes as Integer
```

Argument

None

ThermalPad.SpokeWidth

This property returns the thermal pad's spoke width.

Prototype

```
SpokeWitdth (unit as PPcbUnit) as Double
```

Argument

unit - [Optional]	Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

Via.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an Automation object. All Automation server applications have an application object and all Automation objects have an application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

Via.Attributes

This property returns the collection of all attributes of the via.

Prototype

Attributes As Attributes

Attributes(*name* As String) As Attribute

Argument

name	Name of an existing via attribute.

Comments

When an existing attribute name is passed to this property, it returns that via Attribute object. If the attribute name does not exist, this property returns the collection of all via attributes in an Attributes collection object.

Sample

The following sample code retrieves the number of attributes assigned to the first via found in the open design, assuming at least one via exists, using the Attributes.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set firstVia = ActiveDocument.Vias(1)
MsgBox "There are " & firstVia.Attributes.Count & " attribute(s) in via "
& firstVia.Name
End Sub
```

Via.DrillSize

This property returns the drill size of the via.

Prototype

DrillSize([*unit* As PPcbUnit]) As Double

Argument

unit	[Optional] Unit in which the drill size value is returned.

Comments

None

Sample

The following sample code retrieves the drill size of the first via found in the open design, assuming at least one via exists. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set firstVia = ActiveDocument.Vias(1)
MsgBox firstVia.Name & " has a drill of " & firstVia.DrillSize
End Sub
```

Via.EndLayer

This property returns the end layer of the via.

Prototype

EndLayer As Long

Arguments

None

Comments

None

Sample

The following sample code retrieves the first/end layer of the first via found in the open design, assuming at least one via exists. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set firstVia = ActiveDocument.Vias(1)
MsgBox firstVia.Name & " starts on layer " & firstVia.StartLayer & " and
ends on layer " & firstVia.EndLayer
End Sub
```

See Also

Via.StartLayer

Via.Glued

This property sets or returns whether the via is glued.

Prototype

Glued As Boolean

Arguments

None

Comments

None

Sample

The following sample code glues all vias in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each nextVia In ActiveDocument.Vias
nextVia.Glued = True
Next nextVia
End Sub
```

Via.Highlighted

This property returns whether the via is highlighted.

Prototype

Highlighted as Boolean

Argument

None

Via.Name

This property returns the name of the via.

Prototype

Name As String

Arguments

None

Comments

This property is the default property for the Via object.

Sample

The following sample code lists all vias in the open design by name and then places that list in a custom dialog box. When a via is selected in the list box, the sample selects that via in PADS Layout.

This sample uses the User Dialog Editor in the Sax Basic Engine in PADS Layout. See "Running Code Samples" for more information on running this sample.

See also: Sax Basic Editor On Line Help (C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs\sbe5_000.hlp)

```
Dim ListVias$(10000)
Sub Main
index = 0
For Each nextVia In ActiveDocument.Vias
ListVias$(index) = nextVia.Name
index = index + 1
Next nextVia
' This piece of code is automatically generated by the Basic Dialog Editor
in PADS Layout.
Begin Dialog UserDialog 180,238,"Vias",.CallbackFunc ' %GRID:10,7,1,1
ListBox 10,7,160,203,ListVias(),.ListBox1
OKButton 10,210,160,21
End Dialog
Dim dlg As UserDialog
Dialog dlg
End Sub
' The following function is automatically called by the system when
something has happened
' in the dialog; it is used to easily process user actions.
Function CallbackFunc%(DlgItem$, Action%, SuppValue%)
Select Case Action%
Case 2 ' Value changing or button pressed
If DlgItem$ = "ListBox1" Then
ActiveDocument.SelectObjects(ppcbObjectTypeAll, , False)
ActiveDocument.SelectObjects(ppcbObjectTypeVia, ListVias(SuppValue%))
End If
```

```
End Select
End Function
```

Via.Net

This property returns the net attached to the via.

Prototype

Net As Net

Arguments

None

Comments

None

Sample

The following sample code retrieves the net connected to the first via found in the open design, assuming at least one via exists. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set firstVia = ActiveDocument.Vias(1)
MsgBox firstVia.Name & " is connected to net " & firstVia.Net.Name
End Sub
```

Via.ObjectType

This property returns the type of the object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property returns ppcbObjectTypeVia.

All PADS Layout database objects in the PADS Layout Automation server implement this property to compensate for the lack of a Visual Basic equivalent for the Visual C++ QueryInterface function.

This property is generally used:

- To identify the kind of PADS Layout database objects in a heterogeneous Objects collection.
- When implementing a generic routine that depends on the type of the PADS Layout database object passed as argument. For example:

```
Sub DoSomething(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeComponent
' Do something specific to component objects
Case ppcbObjectTypeNet
' Do something specific to net objects
Case ppcbObjectTypePin
' Do something specific to pin objects
Case ppcbObjectTypeVia
' Do something specific to via objects
Case ppcbObjectTypeConnection
' Do something specific to connection objects
Case ppcbObjectTypeRouteSegment
' Do something specific to route segment objects
Case ppcbObjectTypeJumper
' Do something specific to jumper objects
Case Else
MsgBox "Not a PADS Layout database object"
End Select
End Sub
```

Via.PadStackLayers

This property returns the collection of all padstack layers for this via.

Prototype

PadStackLayers as Objects

PadStackLayers(*layerName* as String) as PadStackLayer

Argument

layerName	Name of padstack layer

Return Values

When a layer name is passed to this property, it returns that PadStackLayer Object. If the name is not specified, this property returns the collection of all padstack layers in an Objects collection object.

Sample

```
For Each via In Application.ActiveDocument.Vias
  For Each layer in via.PadStackLayers

    MsgBox layer.Number & ", " & layer.Name

    pad = layer.Pad
    MsgBox pad.Name & ", " & pad.Shape & ", " & pad.Diameter & ", "
           & pad.InnerDiameter & ", " & pad.Width & ", "
           & pad.Length & ", " & pad.offset & ", "
           & pad.Orientation & ", " & pad.CornerType & ", "
           & pad.CornerRadius

    thermalpad = layer.ThermalPad
    If thermalpad Is Nothing Then
      MsgBox "Thermal pad not defined on this layer"
    Else
      MsgBox thermalpad.Name & ", " & thermalpad.Shape & ", "
             & thermalpad.InnerSize & ", "
             & thermalpad.OuterSize & ", " & thermalpad.Spokes & ", "
             & thermalpad.SpokeAngle  & ", " & thermalpad.SpokeWidth
    End If

    antipad = layer.AntiPad
    If antipad Is Nothing Then
      MsgBox "Anti pad not defined on this layer"
    Else
      MsgBox antipad.Name & ", " & antipad.Shape & ", "
             & antipad.Size
     End If

  Next layer
Next via
```

Via.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

None

Via.PlaneThermal

This property returns whether the via has a plane thermal.

Prototype

PlaneThermal As Boolean

Arguments

None

Comments

None

Sample

The following sample code retrieves the number of plane thermal vias in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
nbPTVias = 0
For Each nextVia In ActiveDocument.Vias
If nextVia.PlaneThermal = True Then nbPTVias = nbPTVias +1
Next nextVia
MsgBox "There are " & nbPTVias & " plane thermal vias (out of " &
ActiveDocument.Vias.Count & ") in " & ActiveDocument.Name
End Sub
```

Via.Plated

This property returns whether the via is plated.

Prototype

Plated As Boolean

Arguments

None

Comments

None

Sample

The following sample code retrieves the number of nonplated vias in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
nbPlatedVias = 0
For Each nextVia In ActiveDocument.Vias
If nextVia.Plated = False Then nbPlatedVias = nbPlatedVias +1
Next nextVia
MsgBox "There are " & nbPlatedVias & " non-plated pins (out of " &
ActiveDocument.Vias.Count & ") in " & ActiveDocument.Name
End Sub
```

Via.PositionX

This property returns the x-coordinate of the via.

Prototype

PositionX([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the x-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the location of the first via found in the open design, assuming at least one via exists, in current design units. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set firstVia = ActiveDocument.Vias(1)
MsgBox firstVia.Name & " position is (" & firstVia.PositionX & ", " &
firstVia.PositionY & ")"
End Sub
```

See Also

Via.PositionY

Via.PositionY

This property returns the y-coordinate of the via.

Prototype

PositionY([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the y-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the location of the first via found in the open design, assuming at least one via exists, in current design units. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set firstVia = ActiveDocument.Vias(1)
MsgBox firstVia.Name & " position is (" & firstVia.PositionX & ", " &
firstVia.PositionY & ")"
End Sub
```

See Also

Via.PositionX

Via.Selected

This property sets or returns whether the via is selected.

Prototype

Selected As Boolean

Arguments

None

Comments

None

Sample

The following sample code selects the first via found in the open design, assuming at least one via exists. See “Running Code Samples” for more information on running this sample.

```
Sub Main
ActiveDocument.SelectObjects(,,False)
ActiveDocument.Vias(1).Selected = True
End Sub
```

See Also

Document.SelectionChange Event

Via.StartLayer

This property returns the start layer of the via.

Prototype

StartLayer As Long

Arguments

None

Comments

None

Sample

The following sample code retrieves the first/end layer of the first via found in the open design, assuming at least one via exists. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set firstVia = ActiveDocument.Vias(1)
MsgBox firstVia.Name & " starts on layer " & firstVia.StartLayer & " and
ends on layer " & firstVia.EndLayer
End Sub
```

See Also

Via.EndLayer

Via.Stitching

This property sets or returns the via's stitching status.

Prototype

Stitching As Boolean

Arguments

None

Comments

None

Sample

The following sample code selects all stitching vias in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each nextVia In ActiveDocument.Vias
If nextVia.Stitching Then
nextVia.Selected = True
End If
Next nextVia
End Sub
```

Via.TestPoint

This property sets or returns whether the via is a test point.

Prototype

TestPoint As PPcbTestPointType

Arguments

None

Comments

None

Sample

The following sample code removes all via test points from the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each nextVia In ActiveDocument.Vias
nextVia.TestPoint = ppcbTestPointNone
Next nextVia
End Sub
```

Via.Type

This property returns the type of the via.

Prototype

Type As String

Arguments

None

Comments

None

Sample

The following sample code retrieves the type of the first via found in the open design, assuming at least one via exists. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set firstVia = ActiveDocument.Vias(1)
MsgBox firstVia.Name & " type is " & firstVia.Type
End Sub
```

View.Application

This property returns the Application object.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an Automation object. All Automation server applications have an application object and all Automation objects have an application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications), to quickly identify the application to which the object belongs.

View.BottomRightX

This property returns the x-coordinate of the lower right corner of the view.

Prototype

BottomRightX([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the lower right x-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the coordinates of the current view in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
x0 = Format$(ActiveDocument.ActiveView.TopLeftX, "Fixed")
y0 = Format$(ActiveDocument.ActiveView.TopLeftY, "Fixed")
x1 = Format$(ActiveDocument.ActiveView.BottomRightX, "Fixed")
y1 = Format$(ActiveDocument.ActiveView.BottomRightY, "Fixed")
MsgBox "View is (" & x0 & ", " & y0 & ") - (" & x1 & ", " & y1 & ")
End Sub
```

See Also

View.BottomRightY,View.TopLeftX,View.TopLeftY

View.BottomRightY

This property returns the y-coordinate of the lower right corner of the view.

Prototype

BottomRightY([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the lower right y-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the coordinates of the current view in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
x0 = Format$(ActiveDocument.ActiveView.TopLeftX, "Fixed")
y0 = Format$(ActiveDocument.ActiveView.TopLeftY, "Fixed")
x1 = Format$(ActiveDocument.ActiveView.BottomRightX, "Fixed")
y1 = Format$(ActiveDocument.ActiveView.BottomRightY, "Fixed")
MsgBox "View is (" & x0 & ", " & y0 & ") - (" & x1 & ", " & y1 & ")
End Sub
```

See Also

View.BottomRightX, View.TopLeftX,View.TopLeftY

View.CenterX

This property returns the x-coordinate of center of the view.

Prototype

CenterX (*Unit* as PPcbUnit) as Double

Argument

unit [Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.

Comments

None

Sample

The following sample code displays the current view's center and zoom level. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set v = ActiveDocument.ActiveView
Dim msg as String
msg = "View parameters:" & Chr(13) & Chr(10)
msg = msg & "Center: (" & v.CenterX & ", " & v.CenterY & ")" & Chr(13) &
Chr(10)
MsgBox msg & "Zoom: " & v.Zoom
End Sub
```

View.CenterY

This property returns the y-coordinate of center of the view.

Prototype

CenterY(*Unit* as PPcbUnit) as Double

Argument

unit [Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.

Comments

None

Sample

The following sample code displays the current view's center and zoom level. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set v = ActiveDocument.ActiveView
Msg = "View parameters:" & Chr(13) & Chr(10)
Msg = Msg & "Center: (" & v.CenterX & ", " & v.CenterY & ")" & Chr(13) &
Chr(10)
MsgBox msg & "Zoom: " & v.Zoom
End Sub
```

View.Name

This property returns the name of the view. For example, in PADS Layout, this function returns the string “Current View.”

Prototype

Name As String

Arguments

None

Comments

This property is the default property for the view object.

Sample

The following sample code retrieves the name of the current view. See “Running Code Samples” for more information on running this sample.

```
Sub Main
MsgBox ActiveDocument.ActiveView.Name
End Sub
```

View.ObjectType

This property returns the type of this object.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

None

View.Parent

This property returns the parent of the object.

Prototype

Parent As Document

Arguments

None

Comments

None

View.PointerX

This property returns the pointer's X position.

Prototype

PointerX (*unit* as PPcbUnit) as Double

Argument

unit	[Optional] Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

Sample

```
' load preview.pcb
Application.ModelessCommand("s")
DlgModelessCmd.Command = "s C8"
DlgModelessCmd.OnOk()

doc = Application.ActiveDocument
view = doc.ActiveView
MsgBox view.PointerX & ", " & view.PointerY
```

View.PointerY

This property returns the pointer's Y position.

Prototype

PointerY (unit as PPcbUnit) as Double

Argument

unit	[Optional] Unit in which the value is returned. This optional argument is ppcbUnitCurrent by default.

Sample

```
' load preview.pcb
Application.ModelessCommand("s")
DlgModelessCmd.Command = "s C8"
DlgModelessCmd.OnOk()

doc = Application.ActiveDocument
view = doc.ActiveView
MsgBox view.PointerX & ", " & view.PointerY
```

View.TopLeftX

This property returns the x-coordinate of the upper left corner of the view.

Prototype

TopLeftX([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the upper left x-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the coordinates of the current view in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
x0 = Format$(ActiveDocument.ActiveView.TopLeftX, "Fixed")
y0 = Format$(ActiveDocument.ActiveView.TopLeftY, "Fixed")
x1 = Format$(ActiveDocument.ActiveView.BottomRightX, "Fixed")
y1 = Format$(ActiveDocument.ActiveView.BottomRightY, "Fixed")
MsgBox "View is (" & x0 & ", " & y0 & ") - (" & x1 & ", " & y1 & ")
End Sub
```

See Also

View.BottomRightX,View.BottomRightY,View.TopLeftY

View.TopLeftY

This property returns the y-coordinate of the upper left corner of the view.

Prototype

TopLeftY([*unit* As PPcbUnit]) As Double

Argument

unit [Optional] Unit in which the upper left y-coordinate is returned.

Comments

None

Sample

The following sample code retrieves the coordinates of the current view in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
x0 = Format$(ActiveDocument.ActiveView.TopLeftX, "Fixed")
y0 = Format$(ActiveDocument.ActiveView.TopLeftY, "Fixed")
x1 = Format$(ActiveDocument.ActiveView.BottomRightX, "Fixed")
y1 = Format$(ActiveDocument.ActiveView.BottomRightY, "Fixed")
MsgBox "View is (" & x0 & ", " & y0 & ") - (" & x1 & ", " & y1 & ")
End Sub
```

See Also

View.BottomRightX,View.BottomRightY,View.TopLeftX,

View.Zoom

This property returns the current zoom factor.

Prototype

Zoom as Double

Arguments

None

Comments

The zoom factor is 1 when the view is zoomed out as far as possible. When zooming in, the zoom factor grows in proportion to the size of object.

Sample

The following sample code displays the current view's center and zoom factor. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set v = ActiveDocument.ActiveView
msg = "View parameters:" & Chr(13) & Chr(10)
msg = msg & "Center: (" & v.CenterX & ", " & v.CenterY & ")" & Chr(13) &
Chr(10)
MsgBox msg & "Zoom: " & v.Zoom
End Sub
```

Wirebond.Angle

This property returns the angle between the wire bond and the edge of the die, in degrees.

Restriction: This information applies to only the BGA toolkit.

Prototype

Angle As Double

Arguments

None

Comments

None

Sample

The following sample code retrieves the angle between the bond wire and the edge of the die for each wire bond in the die, in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.Name & ": Angle: " & Format (aWB.Angle, "#.###")
Next
End If
Next
End Sub
```

Wirebond.Application

This property returns the Application object.

Restriction: This information applies to only the BGA toolkit.

Prototype

Application As Application

Arguments

None

Comments

This property identifies the object as an Automation object. All Automation server applications have an Application object and all Automation objects have an Application property. This property is usually used in Automation client applications that handle large volumes of objects from different sources, such as different Automation server applications, to quickly identify the application to which the object belongs.

This is a Microsoft-required property.

Wirebond.Component

This property returns the component object of the wire bond.

Restriction: This information applies to only the BGA toolkit.

Prototype

Component As Component

Arguments

None

Comments

None

Sample

The following sample code represents a subroutine that depends on the Wirebond object passed as an argument. The code retrieves the name of the component and the name of the decal to which the wire bond belongs. See "Running Code Samples" for more information on running this sample.

```
Sub QueryWB(aWB As Wirebond)
MsgBox "Bond Wire" & aWB.Name &
"belongs to component" & aWP.Component.Name &
"(" & aWP.Component.Decal &")"
End Sub
```

Wirebond.EndOffsetX

This property returns the x offset of the endpoint of the bond wire from the center of the end pad.

Restriction: This information applies to only the BGA toolkit.

Prototype

EndOffsetX([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the X offset value is returned.

Comments

None

Sample

The following sample code retrieves the end offset of each bond wire for each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.Name & ": (" & aWB.EndOffsetX & ", " & aWB.EndOffsetY & ")"
Next
End If
Next
End Sub
```

See Also

Wirebond.EndOffsetY,Wirebond.EndPad,Wirebond.StartOffsetX,Wirebond.StartOffsetY

Wirebond.EndOffsetY

This property returns the y offset of the endpoint of the bond wire from the center of the end pad.

Restriction: This information applies to only the BGA toolkit.

Prototype

EndOffsetY([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the Y offset value is returned.

Comments

None

Sample

The following sample code retrieves the end offset of each bond wire for each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.Name & ": (" & aWB.EndOffsetX & ", " & aWB.EndOffsetY & ")"
Next
End If
Next
End Sub
```

See Also

Wirebond.EndOffsetX,Wirebond.EndPad,Wirebond.StartOffsetX,Wirebond.StartOffsetY

Wirebond.EndPad

This property returns the end pad to which the bond wire is attached.

Restriction: This information applies to only the BGA toolkit.

Prototype

EndPad As Object

Arguments

None

Comments

This property can return either the CBP or SBP object.

Sample

The following sample code retrieves the names of the end pads of each bond wire for each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.StartPad.Name & " - " & aWB.EndPad.Name
Next
End If
Next
End Sub
```

See Also

Wirebond.StartPad, Wirebond.StartOffsetX, Wirebond.StartOffsetY

Wirebond.EndX

This property returns the x-coordinate for the end of the bond wire.

Restriction: This information applies to only the BGA toolkit.

Prototype

EndX([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the x-coordinate value is returned.

Comments

None

Sample

The following sample code retrieves the end position of each bond wire for each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.Name & " End: (" & aWB.EndX & ", " & aWB.EndY & ")"
Next
End If
Next
End Sub
```

See Also

Wirebond.EndY,Wirebond.EndPad, Wirebond.StartX, Wirebond.StartY

Wirebond.EndY

This property returns the y-coordinate for the end of the bond wire.

Restriction: This information applies to only the BGA toolkit.

Prototype

EndY([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the y-coordinate value is returned.

Comments

None

Sample

The following sample code retrieves the end position of each bond wire for each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.Name & " End: (" & aWB.EndX & ", " & aWB.EndY & ")"
Next
End If
Next
End Sub
```

See Also

Wirebond.EndX,Wirebond.EndPad, Wirebond.StartX, Wirebond.StartY

Wirebond.Name

This default property returns the name of the bond wire.

Restriction: This information applies to only the BGA toolkit.

Prototype

Name As String

Arguments

None

Comments

This is a Microsoft-required property.

Sample

The following sample code retrieves the name of each SBP and CBP to which the bond wire is attached. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.Name
Next
End If
Next
End Sub
```

Wirebond.ObjectType

This property returns the object type of the bond wire.

Restriction: This information applies to only the BGA toolkit.

Prototype

ObjectType As PPcbObjectType

Arguments

None

Comments

This property returns PPcbObjectTypeWirebond.

This property is generally used to identify the kind of database objects in a heterogeneous Objects collection or when implementing a generic routine that depends on the type of the database object passed as argument.

All database objects in the Automation server implement this property to compensate for the lack of an equivalent for the Visual C++ QueryInterface function.

Sample

See "Running Code Samples" for more information on running this sample.

```
Sub DoSomethingToDieObject(dbObject As Object)
Select Case dbObject.ObjectType
Case ppcbObjectTypeCBP
' Do something specific to CBP objects
Case ppcbObjectTypeSBP
' Do something specific to SBP objects
Case ppcbObjectTypeWirebond
' Do something specific to Wirebond objects
Case Else
MsgBox "Not a Die object"
End Select
End Sub
```

Wirebond.Parent

This property returns the parent object of the bond wire.

Restriction: This information applies to only the BGA toolkit.

Prototype

Parent As Document

Arguments

None

Comments

This is a Microsoft-required property.

Wirebond.StartOffsetX

This property returns the x offset of the start point of the bond wire from the center of the start pad.

Restriction: This information applies to only the BGA toolkit.

Prototype

StartOffsetX([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the x offset value is returned.

Comments

None

Sample

The following sample code retrieves the start offset of each bond wire for each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.Name & ": (" & aWB.StartOffsetX & ", " & aWB.StartOffsetY &
")"
Next
End If
Next
End Sub
```

See Also

Wirebond.StartOffsetY,Wirebond.StartPad,Wirebond.EndOffsetX,Wirebond.EndOffsetY

Wirebond.StartOffsetY

This property returns the y offset of the start point of the bond wire from the center of the start pad.

Restriction: This information applies to only the BGA toolkit.

Prototype

StartOffsetY([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit [Optional] Unit in which the y offset value is returned.

Comments

None

Sample

The following sample code retrieves the start offset of each bond wire for each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.Name & ": (" & aWB.StartOffsetX & ", " & aWB.StartOffsetY &
")"
Next
End If
Next
End Sub
```

See Also

Wirebond.StartOffsetX,Wirebond.StartPad,Wirebond.EndOffsetX,Wirebond.EndOffsetY

Wirebond.StartPad

This property returns the start pad to which the bond wire is attached.

Restriction: This information applies to only the BGA toolkit.

Prototype

StartPad As Object

Arguments

None

Comments

This property can return either the CBP or the SBP object.

Sample

The following sample code retrieves the names of the end pads of each bond wire for each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.StartPad.Name & " - " & aWB.EndPad.Name
Next
End If
Next
End Sub
```

See Also

Wirebond.EndPad,Wirebond.EndOffsetX,Wirebond.EndOffsetY

Wirebond.StartX

This property returns the x-coordinate for the start of the bond wire.

Restriction: This information applies to only the BGA toolkit.

Prototype

StartX([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the x-coordinate value is returned.

Comments

None

Sample

The following sample code retrieves the start position of each bond wire for each die in the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.Name & " Start: (" & aWB.StartX & ", " & aWB.StartY & ")"
Next
End If
Next
End Sub
```

See Also

Wirebond.StartY, Wirebond.StartPad,Wirebond.EndX, Wirebond.EndY

Wirebond.StartY

This property returns the y-coordinate for the start of the bond wire.

Restriction: This information applies to only the BGA toolkit.

Prototype

StartY([*unit* As PPcbUnit = ppcbUnitCurrent]) As Double

Argument

unit	[Optional] Unit in which the y-coordinate value is returned.

Comments

None

Sample

The following sample code retrieves the start position of each bond wire for each die in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each comp In ActiveDocument.Components
If comp.IsDiePart Then
For Each aWB in comp.Wirebonds
MsgBox aWB.Name & " Start: (" & aWB.StartX & ", " & aWB.StartY & ")"
Next
End If
Next
End Sub
```

See Also

Wirebond.StartX, Wirebond.StartPad, Wirebond.EndX, Wirebond.EndY

Optional Argument

An optional argument is an argument to a property or a method that can be omitted because a default value is used when the optional argument is not specified. The default value is the value that is statistically used most, or one that is most representative for the argument.

For example if the method M([arg1], [arg2]) has both arg1 and arg2 arguments set as *optional*, you can call the method M in four different ways:

M() The default values of both arg1 and arg2 are passed.

M(<value1>) Arg1 <value1> and arg2 default value are passed.

M(, <value2>) Arg1 default value and arg2 <value2> are passed.

M(<value1>, <value2>)
Both arg1 <value1> and arg2 <value2> are passed.

Variant

A Variant is a data type which can contain or represent any kind of data, such as a Boolean value, an Integer value, a Long value, a Double value, a String value, or an Array.

The Variant data type is used in Automation in two different situations:

- When the argument of a property or method is data that can be represented as different values and value types. For example, a specific object in a collection of objects is referenced by its index in the collection (a Long value) or its name (a String value).
- When the argument or the return value of a property or a method is a complex type that is not explicitly defined. For example, an array of points is represented as a Variant.

Exception

An Automation exception is a special notification from the server that signals the Automation client of an error. For example, if a Basic script is trying to delete an attribute that doesn't exist, the program generates an exception. When the script receives the exception, the following occurs:

- If an exception handler is implemented using the On Error Basic statement, then the flow of execution of the client code is rerouted to the exception handler.
- If an exception handler is not implemented, then the client's default handler is called. In all Basic interpreters, the default handler is a break point at the line that generated the exception.

Methods

Application.CreateLibrary

This method creates a new library.

Prototype

Creates new Library

CreateLibrary(*Name* as String, overwrite as Boolean) as Library

Argument

name	Name of the library to create. Should not include wildcards.
overwrite	Should be overwritten if it exists.

Return Values

Created library object or null when failed.

Comments

None

Application.ExportLibraryItems

This method generates a PADS format ASCII file from the library items.

Prototype

ExportLibraryItems(*Filename* as String, *Items* as Collection)

Arguments

filename	Name of the file to which to export the library item. Do not specify a file name extension.
items	Optional collection of items to export. If omitted, all items from the collection are exported.

Return Values

None

Comments

Up to four files can be generated.

Sample

The following sample code exports library items, beginning with the letter 'R', from a library to a specified file. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set coll = GetLibraryItems(, "R*")
ExportLibraryItems("C:\sample", coll)
End Sub
```

Application.GetConfigParamInt

This method retrieves the integer value from the specified parameter and section in the powerpcb.ini file.

Prototype

GetConfigParamInt(

sectionName as String,

paramName as String,

defaultValue as Integer) as Integer

Argument

sectionName	Name of the section containing the parameter name.
paramName	Name of the parameter whose associated integer value is to be retrieved.
defaultValue	If parameter name cannot be found, the default value is returned

Return Values

The parameter value or default value.

Sample

```
MsgBox Application.GetConfigParamString(
      "directories", " FileDir", " C:\PADS Projects\")

MsgBox Application.GetConfigParamInt(
      "general", " Display_Start_up_File_Dialog", 0)
```

Application.GetConfigParamString

This method retrieves the string from the specified parameter and section in the powerpcb.ini file.

Prototype

GetConfigParamString(

sectionName as String,

paramName as String,

defaultValue as String) as String

Argument

sectionName	Name of the section containing the parameter name.
paramName	Name of the parameter whose associated string is to be retrieved.
defaultValue	If parameter name cannot be found, the default value is returned

Return Values

The parameter value or default value.

Sample

```
MsgBox Application.GetConfigParamString(
      "directories", " FileDir", " C:\PADS Projects\")

MsgBox Application.GetConfigParamInt(
      "general", " Display_Start_up_File_Dialog", 0)
```

Application.GetLibraryItems

This method returns a collection of library items in all available libraries, a collection of all items of a given type, or a specific item.

Prototype

GetLibraryItems (*Type* as PPcbLibraryItemType, *Name* as String) as LibraryItem

Arguments

type	Type of items (optional) to retrieve. Default value is ppcbLibraryItemTypeAll.
name	Name of the item to retrieve (optional). May include wildcards, ranges, and lists.

Comments

None

Sample

The following sample code displays the number of available library items. See “Running Code Samples” for more information on running this sample.

```
Sub Main
MsgBox "Number of library items: " & GetLibraryItems().Count
End Sub
```

Application.LockServer

This method locks the Automation server.

Prototype

LockServer()

Arguments

None

Return Values

None

Comments

To speed OLE call processing, use this function when your client makes many OLE calls to the server.

Warning: Never forget to unlock a locked server. Do not keep the server locked for more than a few minutes.

The server locking mechanism speeds up processing by a factor of 2X to 8X the access time of the OLE server method or property calls. The server locking mechanism is dangerous because it disables many internal background tasks in the server, such as memory cleanup and visual updates. These server tasks must be performed regularly, but are disabled so that OLE incoming calls can be processed quickly.

Do not keep the server locked for more than a few minutes.

Sample

The following sample code shows how to use this method. See "Running Code Samples" for more information on running this sample.

```
Sub Main
LockServer
' Do something lengthy, which makes many calls to the PADS Layout
Automation server
UnlockServer
End Sub
```

See Also

Application.UnlockServer

Application.Measure

This method creates and returns a measure value object.

Prototype

Measure(*Value* As Variant, [*DefaultUnit* As String = ""]) As Measure

Arguments

value	String or number representing a measure value including optional prefix and optional physical unit.
defaultunit	Optional string that contains default prefix and/or physical unit.

Comments

This property parses string value such as “100pF” and creates a special object from which you can extract additional information such as real value, unit name, quantity name, and so on.

If the *Value* parameter contains unit information then *DefaultUnit* parameter is ignored.

If the *Value* parameter does not contain unit information and *DefaultUnit* is empty then a new Measure of Size/Dimension is created, and Value parameter will be interpreted by default as a number in current PADS Layout design units (mils).

If the parser cannot recognize measure in the Value parameter it creates a dummy Measure object with the Measure.Value property equal to 0.0 and Measure.Text property equal to the Value parameter.

Notice that there is a difference between Measure (“100”, “pF”) and Measure (“100pF”). Though both versions represent the same physical measure 100pF, the first one stores exact text representation without units.

Sample

The following sample adds a measure. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set M1 = Measure("500pF")
MsgBox M1.Text 'displays 500pF
End Sub
```

See Also

Attribute.Measure

Application.OpenDocument Method

This method opens a design file.

Prototype

OpenDocument(*filename* As String) As Document

Argument

filename	Name of the file to open.

Return Values

If the function succeeds, the return value is the newly opened Document.

If the function fails, the return value is current document.

Comments

If *filename* does not contain the full path to the file, the program uses the path specified by the Application.DefaultFilePath property to locate the file.

If *filename* is an empty string, a new blank design file is created.

If the file specified by *filename* cannot be found or cannot be opened, the return value is current document.

This method does not check if the currently opened file was saved or not. It is the client's responsibility to use the Document.Saved property to check if the open design needs to be saved.

This method generates an exception if a security failure occurs during processing.

Sample

The following sample code opens PWRDEMOA.PCB, assuming that it exists in the folder specified in the Application.DefaultFilePath property. The sample then displays the name of the newly opened file. See "Running Code Samples" for more information on running this sample.

```
Sub Main
OpenDocument(DefaultFilePath & "\PWRDEMOA.PCB")
MsgBox ActiveDocument.FullName & " has just been opened."
End Sub
```

See Also

Application.ActiveDocument, Application.DefaultFilePath, Document.Name, Document.Saved, Application.OpenDocument Event

Application.OpenDocumentNoLock Method

This method opens a design file without file locking.

Prototype

OpenDocumentNoLock(*filename* As String) As Document

Argument

filename	Name of the file to open.

Return Values

If the function succeeds, the return value is the newly opened Document.

If the function fails, the return value is current document.

Application.OpenTempDocument Method

This method works in the same way as OpenDocumentNoLock but additionally doesn't add the filename to the MRU (Most Recently Used) list. This method is mainly intended for macro tests.

Prototype

OpenTempDocument(*filename* As String) As Document

Argument

filename	Name of the file to open.

Return Values

If the function succeeds, the return value is the newly opened Document.

If the function fails, the return value is current document.

Application.Quit Method

Warning: Do not use this method.

This method shuts down the program.

Prototype

Quit()

Arguments

None

Return Values

None

Comments

Microsoft requires that all Automation Application objects implement this method; however, calling this method violates some important client/server rules:

- A server cannot shut down until all clients disconnect from it. Since it is, by definition, not possible for a client to call the Quit method (or any other server method) after it disconnects from the server, it is not possible for a client to shut down a server.
- A client cannot know if other clients are connected to the server. It, therefore, should not shut down a server.
- A server has its own shutdown management process: when the last client disconnects from the server, the server automatically shuts down only if its Graphical User Interface (GUI) is not active (not visible). Otherwise, the server remains active.

To force a shutdown, an Automation client needs to make the program invisible using the Application.Visible property, and then disconnect from it. If no other clients are connected to PADS Layout at that time, the program automatically shuts down. If an Automation client is a Visual Basic script running in the Sax Basic Engine, it can never successfully shut down the Automation server.

See Also

Application.Quit Event

Application.RunMacro

This method runs a macro.

Prototype

RunMacro(*filename* As String, *macroname* As String)

Arguments

filename	Name of the macro file to use.
macroname	Name of the macro to run.

Return Values

None

Comments

If *filename* does not contain the full path to the file, the program uses the path specified by the Application.DefaultFilePath property to locate the file.

If *filename* is Empty, or is an empty string, the current default macro file is used.

If *macroname* is Empty, or not a valid macro name, nothing is performed.

This method generates an exception if a security failure occurs during processing.

Sample

The following sample code runs the macro MACRO1, recorded in the macro file MACROS.MCR, assuming that the macro exists in the folder specified by the Application.DefaultFilePath property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
RunMacro(DefaultFilePath & "\MACROS.MCR", "MACRO1")
End Sub
```

Application.UnlockServer

This method unlocks the Automation server.

Prototype

UnlockServer()

Arguments

None

Return Values

None

Sample

The following sample code shows how to use this method. See "Running Code Samples" for more information on running this sample.

```
Sub Main
LockServer
' Do something lengthy, which makes many calls to the PADS Layout
Automation server
UnlockServer
End Sub
```

See Also

Application.LockServer

AssemblyOptions.Add

This method adds a new Assembly option.

Prototype

Add(*name* As String) As Document

Argument

name Name of the new assembly option to add.

Return Values

The new Assembly Option packaged as a Document object.

Comments

This property generates an exception if the name argument is already an existing Assembly Option or if it is not a valid Assembly Option name.

See Also

AssemblyOptions.Delete

AssemblyOptions.Delete

This method deletes an Assembly Option.

Prototype

Delete(*index* As Long)

Delete(*name* As String)

Arguments

index	Index (in the collection) of the Assembly Option to delete.
name	Name of the Assembly Option to delete.

Return Values

None

Comments

This property generates an exception if:

- The name argument is not an existing Assembly Option.
- The name argument is not a valid Assembly Option to delete.
- The index argument is greater than the number of existing Assembly Options.

Sample

The following sample code deletes all Assembly Options from the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set assopts = ActiveDocument.AssemblyOptions
For index=assopts.Count To 1 Step -1
assopts.Delete(index)
Next index
End Sub
```

See Also

AssemblyOptions.Add

AssemblyOptions.Merge

This method merges two object collections.

Prototype

Merge(*objects* As Objects)

Argument

objects	The collection with objects to merge with the current object collection.

Return Values

None

Comments

This method adds all objects in the *objects* collection object to the current object collection.

AssemblyOptions.Remove

This method removes an object from the collection.

Prototype

Remove(*index* As Long)

Remove(*name* As String)

Arguments

index	*Index* of the object to remove.
name	*Name* of the object to remove.

Return Values

None

Comments

This property generates an exception if the index argument is not valid.

See Also

AssemblyOptions.Add

AssemblyOptions.Reset

This method resets the object collection.

Prototype

Reset()

Arguments

None

Return Values

None

Comments

None

See Also

AssemblyOptions.Remove

AssemblyOptions.Select

This method selects or deselects all objects in the collection.

Prototype

Select([*bSelect* As Boolean = True])

Argument

bSelect	[Optional] True to select. False to deselect.

Return Values

None

Comments

None

AssemblyOptions.Sort

This method sorts objects in the collection by object name.

Prototype

Sort()

Arguments

None

Return Values

None

Comments

None

Attributes.Add

This method adds a new attribute.

Prototype

Add(*name* As String, [*value* AsVariant]) As Attribute

Arguments

name	Name of the new attribute.
value	Optional value of the new attribute. The attribute may be of type Boolean, Byte, Single, Integer, PortInt, Long, Double, String, and Measure object.

Return Values

The new attribute packaged as an Attribute object.

Comments

This property generates an exception if the name argument is already an existing attribute or if it is not a valid attribute name.

See Also

Attributes.Delete, Attribute.Measure, Application.Measure

Attributes.Delete

This method deletes an attribute.

Prototype

Delete(*index* As Long)

Delete(*name* As String)

Arguments

index	Index (in the collection) of the attribute to delete.
name	Name of the attribute to delete.

Return Values

None

Comments

This property generates an exception in the following cases:

- If the *name* argument is not an existing attribute
- If the *name* argument is not a valid attribute to be deleted
- If the *index* argument is higher than the number of existing attributes

Sample

The following sample code deletes the Cost attribute of all components in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each nextComp In ActiveDocument.Components
' Avoid exceptions generated when that attribute does not exist
  If Not nextComp.Attributes("Cost") Is Nothing
    nextComp.Attributes.Delete("Cost")
  End If
Next nextComp
End Sub
```

See Also

Attributes.Add

Attributes.Merge

This method merges two object collections.

Prototype

Merge(*objects* As Objects)

Argument

objects	The collection with objects to merge with the current object collection.

Return Values

None

Comments

This method adds all objects in the objects collection object to the current object collection.

Attributes.Remove

This method removes an object from the collection.

Prototype

Remove(*index* As Long)

Remove(*name* As String)

Arguments

index	Index of the object to remove.
name	Name of the object to remove.

Return Values

None

Comments

This property generates an exception if the index argument is not valid.

See Also

Attributes.Add

Attributes.Reset

This method resets the object collection.

Prototype

Reset()

Arguments

None

Return Values

None

Comments

None

See Also

Attributes.Remove

Attribute.Measure

This property returns the measure value object.

Prototype

Measure As Measure

Arguments

None

Comments

This property parses an attribute string value such as “100pF” and creates a special object from which you can extract additional information such as real value, unit name, quantity name, and so on.

If the internal parser cannot recognize a measure value object in the attribute it creates a dummy Measure object with the Measure.Value property equal to 0.0 and Measure.Text property equal to Attribute.Value property.

Sample

The following sample shows a quantity name of the capacitor's Value attribute (Capacitance) of part C1 assuming that it exists in the open schematic. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Set C1 = ActiveDocument.Components("C1")
MsgBox C1.Attributes("Value").Measure.Quantity 'shows "Capacitance"
End Sub
```

See Also

Application.Measure

Attributes.Select

This method selects or deselects all objects in the collection.

Prototype

Select([*bSelect* As Boolean = True])

Argument

bSelect [Optional] True to select. False to deselect.

Return Values

None

Comments

None

Attributes.Sort

This method sorts objects in the collection by object name.

Prototype

Sort()

Arguments

None

Return Values

None

Comments

None

Component.AddLabel

This method adds a new label to the component.

Prototype

AddLabel (*Type* as PPcbLabelType, *Attribute* as Object, *Layer* as Long, *PositionX* as Double, *PositionY* as Double, *Height* as Double, *LineWidth* as Double, *Units* as PPcbUnit, *Orientation* as Double, *Mirror* as Boolean) as Label

Arguments

type	Type of new label (optional). Default value is ppcbLabelTypeRefDesignator.
attribute	Attribute object to which the label is linked (optional).
layer	Number of layer on which to place the new label (optional). The default is the component layer.
positionX	x-coordinate of the new object relative to the component's origin (optional). Default is 0.
positionY	y-coordinate of the new object relative to the component's origin (optional). Default is 0.
height	Height of the label (optional). If omitted, the value specified in the Tools/Options/Drafting tab/Reference Designators frame. The Height edit box is used.
linewidth	Line width of the label. If omitted, the value specified in the Tools/Options/Drafting tab/Reference Designators frame. The Width edit box is used.
units	[Optional] Unit in which PositionX, PositionY, Height, and LineWidth are returned. Default is ppcbUnitCurrent.
orientation	Orientation of the label (optional). The default is 0.
mirror	True if the label is mirrored (optional). The default is False.

Return Values

Returns the newly added label object.

Comments

None

Sample

The following sample code adds a reference designator to a U1 component at a position (200 mils, 200 mils) relative to the component's origin. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Dim c as Component
Set c = Document.Components("U1")
c.AddLabel(ppcbLabelTypeRefDesignator, , , 200, 200, , , ppcbUnitMils)
End Sub
```

Component.Move

This method moves the component to a new location.

Prototype

Move(*x* As Double, *y* As Double, [*unit* As PPcbUnit])

Arguments

x	x-coordinate of the new component position.
y	y-coordinate of the new component position.
unit	[Optional] Unit in which the x and y coordinates are expressed. This optional argument is ppcbUnitCurrent by default.

Return Values

None

Comments

The proper completion of this method depends on the items listed below. To force a component move, you must first disable the following:

- Glued status, which is set using the Component.Glued property.
- DRC mode, which is set using the Document.Preference property.
- Nudge mode, which is set using the Document.Preference property.
- ModifyUnionMember mode, which is set using the Document.Preference property.

In addition, a component cannot be moved if it belongs to a physical design reuse component or if it is attached to protected traces.

This property generates an exception if any of the above constraints are violated or if a security failure occurs during processing.

Sample 1

The following sample code moves all components to the origin of the open design, after ungluing them and disabling all automatic DRC and Nudge checking options. See “Running Code Samples” for more information on running this sample.

```
Sub Main
ActiveDocument.Preference("DRC") = ppcbDRCOff
ActiveDocument.Preference("Nudge") = ppcbNudgeOff
For Each nextComp In ActiveDocument.Components
nextComp.Glued = False
nextComp.Move(0,0)
Next nextComp
End Sub
```

Sample 2

The following sample code places all components, in the open design, in a circle centered on the design origin, with a 5000 mils radius. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.Preference("DRC") = ppcbDRCOff
ActiveDocument.Preference("Nudge") = ppcbNudgeOff
angleInc = 2*3.14159/ActiveDocument.Components.Count
counter = 0
For Each nextComp In ActiveDocument.Components
nextComp.Glued = False
nextComp.Move(5000.0*Cos(counter*angleInc), 5000.0*Sin(counter*angleInc),
ppcbUnitMils)
counter = counter+1
Next nextComp
End Sub
```

See Also

Document.Preference, Component.PositionX, Component.PositionY, Document.PositionsChange

Component.MoveCenter

This method moves the component so that the specified point becomes the component's center.

Prototype

MoveCenter (*x* as Double, *y* as Double, *unit* as PPcbUnit, Origin as PPcbOriginType)

Arguments

x	x-coordinate of new center.
y	y-coordinate of new center.
unit	[Optional] Unit in which the x and y coordinates are expressed. This optional argument is ppcbUnitCurrent by default.
origin	[Optional]Type of reference point from which the result is counted. The default value is ppcbOriginTypeDesign.

Return Values

None

Comments

This method moves the component by x and y coordinates simultaneously, as opposed to assigning values to CenterX and CenterY properties.

Sample

The following sample code moves component U1 to the point with coordinates (3200 mils, 1500 mils). See “Running Code Samples” for more information on running this sample.

```
Sub Main
Dim c as Component
Set c = Document.Components("U1")
c.MoveCenter(3200, 1500, pcbUnitMils)
End Sub
```

Document.Activate

This method activates the window associated with a document.

Prototype

Activate()

Arguments

None

Return Values

None

Comments

This is a Microsoft requirement. However, because PADS Layout is an SDI (Single Document Interface) server application, this function has no effect.

Document.AddText

This method adds new text to the document.

Prototype

AddText (*Text* as String, *Layer* as Long, *PositionX* as Double, *PositionY* as Double, *Height* as Double, *LineWidth* as Double, *Units* as PPcbUnit, *Orientation* as Double, *Mirror* as Boolean) as Text

Arguments

text	Contents of the new text object (required).
layer	Number of layer on which to place the new text (optional). The default is 1.
positionX	x-coordinate of the new object (optional). The default is 0.
positionY	y-coordinate of the new object (optional). The default is 0.
height	Height of the text (optional). If omitted, the value specified in the Tools/Options/Drafting tab/Text frame/Height option is used.
linewidth	Line width of the text. If omitted, the value specified in the Tools/Options/Drafting tab/Text frame/Width option is used.
units	[Optional] Unit in which the unit of measurement is returned. This optional argument is ppcbUnitCurrent by default.
orientation	Orientation of the text (optional). The default is 0.
mirror	True if the text is mirrored (optional). The default is False.

Return Values

Returns the newly added text object.

Comments

None

Sample

This sample adds the text “Hello!” to the design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
ActiveDocument.AddText("Hello !", 1, 200, 200, , , pcbUnitMils)
End Sub
```

Document.CheckASCII

This method compares the given PADS Layout format netlist with current PADS Layout format netlist.

Prototype

CheckASCII(*name* As String, [*ignorenet* As String]) As Long

Arguments

name:	Name of an existing netlist file to compare against.
ignorenet:	[Optional] Ignore the specified net.

Return Values

If the function succeeds, the return value is nonzero. If the function fails, the return value is zero (0).

Comments

The function fails if the specified file *name* does not exist or its format is incorrect.

This method generates an exception if a PADS Layout security failure occurs during processing.

Sample

The following sample code compares the specified netlist (padsnet.asc) with the current netlist in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.CheckASCII(DefaultFilePath & "\padsnet.asc")
End Sub
```

Document.ExportASCII

This method generates a PADS Layout-format ASCII file from the current design.

Prototype

ExportASCII(*name* As String, [*sections* As PPcbASCIISections = ppcbASCIISectionAll], [*ver* As PPcbASCIIVersion], [*expandAttrs* As PPcbAttrFlags = ppcbAttrNone]) As Long

Arguments

name	Name of the file to which to export the netlist.
sections	[Optional] Sections of ASCII file to output (all by default).
ver	[Optional] Version in which to export (for backward compatibility).
expandAttrs	[Optional] Attribute expansion options (none by default).

Return Values

If the function succeeds, the return value is nonzero.

If the function fails, the return value is zero (0).

Comments

This method is an advanced version of Document.ExportNetList, extended by the *sections* argument.

If the specified file *name* does not exist the function creates a new file. If the file *name* does exist, this method overwrites the existing file.

This property generates an exception if the *ver* argument is not a valid PADS Layout ASCII version number.

Sample

The following sample code creates an ASCII file with the specified name padsnet.asc, containing only part and connection sections. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.ExportASCII(DefaultFilePath & "\padsnet.asc",
ppcbASCIISectionParts Or ppcbASCIISectionConnections)
End Sub
```

See Also

Document.ImportNetList, Document.ExportNetList

Document.ExportECOFile

This method generates an .eco file with the specified file name.

Prototype

ExportECOFile(*name* As String) As Long

Argument

name	Name of the new file to which to copy.

Return Values

If the function succeeds, the return value is nonzero.

If the function fails, the return value is zero (0).

Comments

The function fails if the .eco file does not currently exist. To create an ECO file, the function has to exit .eco mode and clear the undo buffer .

Sample

The following sample code creates an ECO file with the specified name pcb2eco.eco. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.ExportECOFile(DefaultFilePath & "\pcb2eco.eco")
End Sub
```

See Also

Document.ImportECOFile

Document.ExportNetList

This method generates a PADS Layout format netlist from the current design.

Prototype

ExportNetList(*name* As String, [*ver* As PPcbASCIIVersion]) As Long

Arguments

name	Name of the file to which to export the netlist.
ver	[Optional] Version in which to export (for backward compatibility).

Return Values

If the function succeeds, the return value is nonzero.

If the function fails, the return value is zero (0).

Comments

If the specified file *name* does not exist the function creates new file. If the file *name* does exist, this method overwrites the existing file.

This property generates an exception if the ver argument is not a valid PADS Layout ASCII version number.

Sample

The following sample code creates a netlist file with the specified name padsnet.asc. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.ExportNetList(DefaultFilePath & "\padsnet.asc")
End Sub
```

See Also

Document.ImportNetList

Document.ExportRules

This method generates an ASCII file with rules from the current design.

Prototype

ExportRules(*name* As String, [*ver* As PPcbASCIIVersion]) As Long

Arguments

name	Name of the file to which to export rules.
ver	[Optional] Version in which to export (for backward compatibility).

Return Values

If the function succeeds, the return value is nonzero.

If the function fails, the return value is zero (0).

Comments

If the specified file name does not exist, the function creates a new file. If the file name does exist, this method overwrites the existing file.

This property generates an exception if the ver argument is not a valid PADS Layout ASCII version number.

Sample

The following sample code creates a rules file with the specified name padsnet.rul. See “Running Code Samples” for more information on running this sample.

```
Sub Main
ActiveDocument.ExportRules(DefaultFilePath & "\padsnet.rul")
End Sub
```

Document.GetColor

This method returns the color of the specified document element.

Prototype

GetColor(*colorType* as PPcbDocumentColor) as Integer

Argument

colorType	Specifies the document element

Return Values

The index of the color in the palette.

Sample

```
doc = Application.ActiveDocument

msg = ""
msg = msg & doc.GetColor(ppcbDocumentColorBackground) & ", "
msg = msg & doc.GetColor(ppcbDocumentColorSelection) & ", "
msg = msg & doc.GetColor(ppcbDocumentColorHighlight) & ", "
msg = msg & doc.GetColor(ppcbDocumentColorBoardOutline) & ", "
msg = msg & doc.GetColor(ppcbDocumentColorConnection)

MsgBox msg

curr_bkg_color = doc.GetColor(ppcbDocumentColorBackground)
doc.SetColor(ppcbDocumentColorBackground, 2)

MsgBox "Press any key"

doc.SetColor(ppcbDocumentColorBackground, curr_bkg_color)
```

Document.GetObjects

This method returns a collection of PADS Layout database objects.

Prototype

GetObjects([*type* As PPcbObjectType = ppcbObjectTypeAll], [*value* As String], [*selected* As Boolean = False]) As Objects

Arguments

type	[Optional] Type of PADS Layout database object to get.
value	[Optional] Value or name of the object(s) to get.
selected	[Optional] True to get selected objects only. False to get all objects.

All arguments to this method are optional, which means that it can be called with no argument at all, or with any combination of arguments. See the samples below for more information.

Name argument supports wildcarding ("U*"), lists of items delimited by comma ("U1, U2, R1"), ranges specified by two object names and the dash character ("U1 - U10, U12, R1 - R20"). Dash must be surrounded by spaces since the dash is a legal symbol in an object name. Only one wildcard per name is allowed and you cannot specify wildcards in a range. You can pass name such as "U*, R*, C1 – C100" but you cannot pass name such as "U*1*" or "C1* - C10*".

Return Values

The returned object is an Objects collection object. If no objects satisfy the request, the returned collection is empty.

Comments

To get all objects of the same type, use the corresponding object Document property instead of this method. For example, to get all connections in the open design, use Document.Connections instead of Document.GetObjects (ppcbObjectTypeConnection).

This method does not support all of the object types defined in PPcbObjectType. The method will generate an exception if the type argument is not one of the following values:

ppcbObjectTypeAssociatedNet

ppcbObjectTypeComponent

ppcbObjectTypeNet

ppcbObjectTypePin

ppcbObjectTypeVia

ppcbObjectTypeConnection

ppcbObjectTypeRouteSegment

ppcbObjectTypeJumper

ppcbObjectTypePartType

ppcbObjectTypeCBP

ppcbObjectTypeSBP

ppcbObjectTypeWirebond

ppcbObjectTypeNetClass

ppcbObjectTypeDrawing

ppcbObjectTypeText

ppcbObjectTypeLabel

ppcbObjectTypeAll

Sample

The following sample code shows different ways to use this method, displaying the number of objects retrieved for each way, using the Objects.Count property. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Dim objs As Object
' Ex1: Get all objects of all types
Set objs = ActiveDocument.GetObjects
MsgBox "Ex1: " & objs.Count & " objects."
' Ex2: Get all selected objects of all types
Set objs = ActiveDocument.GetObjects(,,True)
MsgBox "Ex2: " & objs.Count & " selected objects."
' Ex3: Get all net objects
Set objs = ActiveDocument.GetObjects(ppcbObjectTypeNet)
MsgBox "Ex3: " & objs.Count & " net objects."
' Ex4: Get all net objects of name "VCC" (there is at least 1 of course)
Set objs = ActiveDocument.GetObjects(ppcbObjectTypeNet, "VCC")
MsgBox "Ex4: " & objs.Count & " VCC net objects."
' Ex5: Get all part objects which names begin with U
Set objs = ActiveDocument.GetObjects(ppcbObjectTypeComponent, "U*")
MsgBox "Ex3: " & objs.Count & " U* part objects."
End Sub
```

See Also

Document.SelectObjects, Document.Components, Document.Jumpers, Document.Connections, Document.Nets, Document.Pins, Document.RouteSegments, Document.Vias, Document.PartTypes

Document.GetVisibility

This method returns the visibility of the specified design object type.

Prototype

GetVisibility(*objectType* as PPcbDesignObject) as Boolean

Argument

objectType	Specifies design object type.

Return Values

TRUE if visible, FALSE if hidden.

Sample

```
doc = Application.ActiveDocument

If doc.GetVisibility(ppcbDesignObjectPad) = true Then
      MsgBox "Pads are visible"
Else
      MsgBox "Pads are invisible"
End If

doc.SetVisibility(ppcbDesignObjectTrace, false)

MsgBox "Press any key"

doc.SetVisibility(ppcbDesignObjectTrace, true)
```

Document.ImportECOFile

This method imports an .eco file.

Prototype

ImportECOFile(*name* As String) As Long

Argument

name Name of an existing .eco file to import.

Return Values

If the function succeeds, the return value is nonzero.

If the function fails, the return value is zero (0).

Comments

The function fails if the specified file *name* does not exist or its format is incorrect.

This method generates an exception if a PADS Layout security failure occurs during processing.

Sample

The following sample code imports the specified .eco file, pcb2eco.eco, into the current design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.ImportECOFile(DefaultFilePath & "\pcb2eco.eco")
End Sub
```

See Also

Document.ExportECOFile

Document.ImportNetList

This method imports a PADS Layout format netlist file.

Prototype

ImportNetList(*name* As String) As Long

Argument

name Name of an existing netlist file to import.

Return Values

If the function succeeds, the return value is nonzero.

If the function fails, the return value is zero (0).

Comments

The function fails if the specified file *name* does not exist or its format is incorrect.

This method generates an exception if a PADS Layout security failure occurs during processing.

Sample

The following sample code imports the specified netlist file, padsnet.asc, into the current design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
ActiveDocument.ImportNetList(DefaultFilePath & "\padsnet.asc")
End Sub
```

See Also

Document.ExportNetList

Document.IntegrityTest

This method runs the integrity test.

Prototype

IntegrityTest() as Boolean

Argument

None

Return Values

TRUE if the test passed without errors, FALSE in other cases.

Document.Save Method

This method saves the document.

Prototype

Save

Arguments

None

Return Values

None

Comments

This method generates an exception if a PADS Layout security failure occurs during processing.

Sample

The following sample code saves, if needed, the open design. See “Running Code Samples” for more information on running this sample.

```
Sub Main
If ActiveDocument.Saved = False Then ActiveDocument.Save
End Sub
```

See Also

Document.SaveAs, Document.Saved, Document.Save Event

Document.SaveAs

This method saves the document with a new name.

Prototype

Save(*name* As String)

Argument

name	New name for the file.

Return Values

None

Comments

This method generates an exception if a PADS Layout security failure occurs during processing.

Sample

The following sample code creates a custom backup of the open design. If the PCB design file name is XXX.PCB, the backup file is saved in the same folder with the name XXX (Backup on 12-28-2001 4h40).PCB. See “Running Code Samples” for more information on running this sample.

```
Sub Main
curDesignName = ActiveDocument.FullName
theDate = Month(Date) & "-" & Day(Date) & "-" & Year(Date) & " at " &
Hour(Now) & "h" & Minute(Now)
bakDesignName = Left$(curDesignName, Len(curDesignName)-4) & " (Backup on
" & theDate & ").pcb"
ActiveDocument.SaveAs(bakDesignName)
OpenDocument(curDesignName)
End Sub
```

See Also

Document.Save Method, Document.Saved, Document.Save Event

Document.SaveAsNoLock

This method saves the document with a new name and releases the file lock.

Prototype

SaveAsNoLock(*filename* as String)

Argument

filename	New name for the file

Return Values

This method generates an exception if a PADS Layout security failure occurs during processing.

Document.SaveNoLock

This method saves the document and releases the file lock.

Prototype

SaveNoLock

Argument

None

Return Values

This method generates an exception if a PADS Layout security failure occurs during processing.

Document.SaveAsTemp

This method works in the same way as SaveAsNoLock but additionally it doesn't add the filename to the MRU (Most Recently Used) list. This method is mainly intended for macro tests.

Prototype

SaveAsTemp(*filename* as String)

Argument

filename	New name for the file

Return Values

This method generates an exception if a PADS Layout security failure occurs during processing.

Document.SaveTemp

This method works in the same way as SaveNoLock but additionally it doesn't add the filename to the MRU (Most Recently Used) list. This method is mainly intended for macro tests.

Prototype

SaveTemp

Argument

None

Return Values

This method generates an exception if a PADS Layout security failure occurs during processing.

Document.SelectObjects

This method selects or deselects PADS Layout database objects.

Prototype

SelectObjects([*type* As PPcbObjectType = ppcbObjectTypeAll], [*value* As String], [*select* As Boolean = True])

Arguments

type	[Optional] Type of database object to select/deselect.
value	[Optional] Value or name of the objects to select/deselect.
select	[Optional] True to select. False to deselect.

All arguments for this method are optional, which means that it can be called with no argument at all, or with any combination of arguments. See the samples below for more information.

Name argument supports wildcarding (“U*”), lists of items delimited by comma (“U1, U2, R1”), ranges specified by two object names and the dash character (“U1 - U10, U12, R1 - R20”). Dash must be surrounded by spaces since the dash is a legal symbol in an object name. Only one wildcard per name is allowed and you cannot specify wildcards in a range. You can pass *name* such as “U*, R*, C1 – C100” but you cannot pass *name* such as “U*1*” or “C1* - C10*”.

Return Values

None

Comments

This property generates an exception if the type argument is not a valid PADS Layout database object type.

Sample

The following sample code shows different ways to use this method, displaying the number of objects selected for each way, using the Document.GetObjectsmethod and the Objects.Count property. See “Running Code Samples” for more information on running this sample.

```
Sub Main
Dim objs As Object
' Ex1: Select all objects of all types
ActiveDocument.SelectObjects
Set objs = ActiveDocument.GetObjects(,,True)
MsgBox "Ex1: " & objs.Count & " selected objects (all)."
' Ex2: Unselelect all objects of all types
ActiveDocument.SelectObjects(,,False)
Set objs = ActiveDocument.GetObjects(,,True)
MsgBox "Ex2: " & objs.Count & " selected objects (none)."
' Ex3: Select all net objects
```

```
ActiveDocument.SelectObjects(ppcbObjectTypeNet)
Set objs = ActiveDocument.GetObjects(,,True)
MsgBox "Ex3: " & objs.Count & " selected objects (all nets)."
' Ex4: Unselect net VCC
ActiveDocument.SelectObjects(ppcbObjectTypeNet, "VCC", False)
Set objs = ActiveDocument.GetObjects(,,True)
MsgBox "Ex4: " & objs.Count & " selected objects (all nets except VCC)."
' Ex5: Select only part objects which names begin with U
ActiveDocument.SelectObjects(,,False)
 ActiveDocument.SelectObjects(ppcbObjectTypeComponent, "U*")
Set objs = ActiveDocument.GetObjects(,,True)
MsgBox "Ex5: " & objs.Count & " selected U* part objects."
End Sub
```

See Also

Document.GetObjects, Document.SelectionChange Event

Document.SetColor

This method sets the color for the specified document element.

Prototype

SetColor(*colorType* as PPcbDocumentColor, *colorIndex* as Integer)

Argument

colorType	Specifies document element
colorIndex	Index of the color in the palette. Must be between 0 and 31

Sample

```
doc = Application.ActiveDocument

msg = ""
msg = msg & doc.GetColor(ppcbDocumentColorBackground) & ", "
msg = msg & doc.GetColor(ppcbDocumentColorSelection) & ", "
msg = msg & doc.GetColor(ppcbDocumentColorHighlight) & ", "
msg = msg & doc.GetColor(ppcbDocumentColorBoardOutline) & ", "
msg = msg & doc.GetColor(ppcbDocumentColorConnection)

MsgBox msg

curr_bkg_color = doc.GetColor(ppcbDocumentColorBackground)
doc.SetColor(ppcbDocumentColorBackground, 2)

MsgBox "Press any key"

doc.SetColor(ppcbDocumentColorBackground, curr_bkg_color)
```

Document.SetVisibility

This method sets visibility for the specified design object type.

Prototype

SetVisibility(*objectType* as PPcbDesignObject, *objectVisibility* as Boolean)

Argument

objectType	Specifies design object type.
objectVisibility	TRUE if visible, FALSE if hidden.

Sample

```
doc = Application.ActiveDocument

If doc.GetVisibility(ppcbDesignObjectPad) = true Then
      MsgBox "Pads are visible"
Else
      MsgBox "Pads are invisible"
End If

doc.SetVisibility(ppcbDesignObjectTrace, false)

MsgBox "Press any key"

doc.SetVisibility(ppcbDesignObjectTrace, true)
```

Label.Delete

This method deletes the Label object.

Prototype

Delete as Boolean

Arguments

None

Return Value

Returns True if the object was successfully deleted. Returns False if the object cannot be deleted.

Comments

None

Sample

The following sample deletes selected labels. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeLabel,,TRUE)
For Each label In selected
If label.Delete() = False Then
MsgBox "Cannot delete the label!"
Else
MsgBox "The label was deleted successfully!"
End If
Next label
End Sub
```

Layer.GetColor

This method returns color of the specified layer element.

Prototype

GetColor(*colorType* as PPcbLayerColor) as Integer

Argument

colorType	Specifies layer element.

Return Values

Index of the color in the palette.

Layer.GetDielectricConstant

This method returns the layer dielectric constant.

Prototype

GetDielectricConstant(*dielectricLayer* as PPcbDielectricLayer) as Double

Argument

dielectricLayer	Specifies dielectric layer relevant to this electrical layer.

Return Values

The dielectric constant.

Layer.GetDielectricThickness

This method gets the layer dielectric thickness.

Prototype

GetDielectricThickness (

dielectricLayer as PPcbDielectricLayer,

unit as PPcbUnit) as Double

Argument

dielectricLayer	Specifies dielectric layer relevant to this electrical layer.
unit	[Optional] Unit in which value is returned. This optional argument is ppcbUnitCurrent by default.

Return Values

The dielectric thickness.

Layer.GetDielectricType

This method returns the layer dielectric type.

Prototype

GetDielectricType(*dielectricLayer* as PPcbDielectricLayer) as PPcbDielectricType

Argument

dielectricLayer	Specifies dielectric layer relevant to this electrical layer.

Return Values

The dielectric type.

Layer.SetColor

This method sets color for the specified layer element.

Prototype

SetColor(*colorType* as PPcbLayerColor, *colorIndex* as Integer)

Argument

colorType	Specifies layer element.
colorIndex	Index of the color in the palette. Must be between 0 and 31.

Layer.SetDielectricConstant

This method sets the layer dielectric constant.

Prototype

SetDielectricConstant(

dielectricLayer as PPcbDielectricLayer,

dielectricConstant as Double)

Argument

dielectricLayer	Specifies dielectric layer relevant to this electrical layer.
dielectricConstant	Specifies dielectric constant.

Layer.SetDielectricThickness

This method sets the layer dielectric thickness.

Prototype

SetDielectricThickness (

dielectricLayer as PPcbDielectricLayer,

dielectricThickness as Double,

unit as PPcbUnit)

Argument

dielectricLayer	Specifies dielectric layer relevant to this electrical layer.
dielectricThickness	Specifies dielectric thickness.
unit	[Optional] Unit in which value is passed to this method. This optional argument is ppcbUnitCurrent by default.

Layer.SetDielectricType

This method sets the layer dielectric type.

Prototype

SetDielectricType (

dielectricLayer as PPcbDielectricLayer,

dielectricType as PPcbDielectricType)

Argument

dielectricLayer	Specifies dielectric layer relevant to this electrical layer.
dielectricType	Specifies dielectric type.

Library.GetLibraryItems

This method returns either the object collection of all items in this library or a specified item.

Prototype

GetLibraryItems (*Type* as PPcbLibraryItemType, *Name* as String) As Collection

Arguments

name	Name of LibraryItem objects to retrieve (optional). May contain wildcards, lists, and ranges. If omitted, matches any name.
type	Parameter specifying the type of the objects to retrieve (optional). It is ppcbLibraryItemTypeAll by default.

Comments

None

Sample

This sample displays the number of items in a library. See “Running Code Samples” for more information on running this sample.

```
Sub Main
For Each lib In Libraries
count = lib.GetLibraryItems(ppcbLibraryItemTypeDecal, "DIP*").Count
MsgBox "Library " & lib.Name & " has " & count & " DIP decals"
Exit For
Next lib
End Sub
```

Library.ImportLibraryItems

This method reads library items from PowerPCB-format or PADS Layout-format ASCII files.

Prototype

ImportLibraryItems (*Filename* as String) as Collection

Arguments

name	Name of the file from which to import library items.
return value	Returns the collection of imported items.

Return Values

Returns the collection of imported items.

Comments

None

Sample

This sample imports library items from a file, and displays the number of imported items. See "Running Code Samples" for more information on running this sample.

```
Sub Main
For Each lib In Libraries
Set coll = lib.ImportLibraryItems("C:\sample")
MsgBox coll.Count & " items are successfully imported"
Exit For
Next lib
End Sub
```

Library.ImportLibraryItems2

This method reads library items from PowerPCB-format or PADS Layout-format ASCII files.

Prototype

ImportLibraryItems2 (*Filename* as String, ImportOption as PcbImportLibMode) as Collection

Arguments

Filename	Name of the file from which to import library items.
ImportOption	specifies the preference for overwriting existing items.

Return Values

Returns the collection of just imported items.

Comments

If ImportOption is pcbImportLibModePrompt, then the Application object fires the OverwriteLibraryItemPrompt event. The client application must process this event to specify whether the existing library item should be overwritten.

Objects.Add

This method adds an object to the collection.

Prototype

Add(*obj* As object)

Argument

obj Object to add to the collection. It must be a database object, such as Component, Net, Jumper, Pin, Via, Connection, or RouteSegment.

Return Values

None

Comments

This property generates an exception if the object argument is not a database object.

See Also

Objects.Remove

Objects.Merge

This method merges two object collections.

Prototype

Merge(*objects* As Objects)

Argument

objects	The collection with objects to merge with the current object collection.

Return Values

None

Comments

This method adds all objects in the objects collection object to the current object collection.

Objects.Remove

This method removes an object from the collection.

Prototype

Remove(*index* As Long)

Remove(*name* As String)

Arguments

index	Index of the object to remove.
name	Name of the object to remove.

Return Values

None

Comments

This property generates an exception if the *index* argument is not valid.

See Also

Objects.Add

Objects.Reset

This method resets the object collection.

Prototype

Reset()

Arguments

None

Return Values

None

Comments

None

See Also

Objects.Remove

Objects.Select

This method selects or deselects all objects in the collection.

Prototype

Select([*bSelect* As Boolean = True])

Argument

bSelect	[Optional] True to select. False to deselect.

Return Values

None

Comments

None

Objects.Sort

This method sorts objects in the collection by object name.

Prototype

Sort()

Arguments

None

Return Values

None

Comments

None

Text.Delete

This method deletes the Text object.

Prototype

Delete as Boolean

Arguments

None

Comments

None

Sample

The following sample deletes the selected text. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set selected = ActiveDocument.GetObjects(ppcbObjectTypeText,, TRUE)
For Each text In selected
If text.Delete() = False Then
MsgBox "Cannot delete the text!"
Else
MsgBox "The text was deleted successfully!"
End If
Next text
End Sub
```

View.Pan

This method pans the view to a specified location.

Prototype

Pan(*x* As Double, *y* As Double, [*unit* As PPcbUnit])

Arguments

x	x-coordinate of the point to which to pan.
y	y-coordinate of the point to which to pan.
unit	[Optional] Unit in which the X and Y values are given.

Return Values

None

Comments

None

Sample

The following sample code centers the view to the location of component U1, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
x = ActiveDocument.Components("U1").PositionX
y = ActiveDocument.Components("U1").PositionY
ActiveDocument.ActiveView.Pan(x,y)
End Sub
```

See Also

View.Change

View.Refresh

This method refreshes the view.

Prototype

Refresh

Arguments

None

Return Values

None

Comments

None

Sample

The following sample code redraws the view. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.ActiveView.Refresh
End Sub
```

View.SetExtents

This method sets the view extents.

Prototype

SetExtents(*tlx* As Double, *tly* As Double, *brx* As Double, *bry* As Double, [*unit* As PPcbUnit])

Arguments

tlx	x-coordinate of the top left corner of the new view.
tly	y-coordinate of the top left corner of the new view.
brx	x-coordinate of the bottom right corner of the new view.
bry	y-coordinate of the bottom right corner of the new view.
unit	[Optional] Unit in which the tlx, tly, brx, and bry values are given.

Return Values

None

Comments

None

Sample

The following sample code sets the view to the extents of all pins connected to net VCC, assuming it exists in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
xMin = 100000000.0
yMin = 100000000.0
xMax = -100000000.0
yMax = -100000000.0

For Each nextPin In ActiveDocument.Nets("VCC").Pins
If nextPin.PositionX < xMin Then xMin = nextPin.PositionX
If nextPin.PositionX > xMax Then xMax = nextPin.PositionX
If nextPin.PositionY < yMin Then yMin = nextPin.PositionY
If nextPin.PositionY > yMax Then yMax = nextPin.PositionY
Next nextPin
ActiveDocument.ActiveView.SetExtents(xMin, yMin, xMax, yMax)
End Sub
```

See Also

View.Change,View.SetExtentsToAll,View.SetExtentsToBoard

View.SetExtentsToAll

This method sets the view extents to all objects in the design.

Prototype

SetExtentsToAll()

Arguments

None

Return Values

None

Comments

None

Sample

The following sample code sets the view extents to all objects in the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.ActiveView.SetExtentsToAll
End Sub
```

See Also

View.SetExtents,View.SetExtentsToBoard,View.Change

View.SetExtentsToBoard

This method sets the view extents to the design board extents.

Prototype

SetExtentsToBoard()

Arguments

None

Return Values

None

Comments

None

Sample

The following sample code sets the view extents to the board of the open design. See "Running Code Samples" for more information on running this sample.

```
Sub Main
ActiveDocument.ActiveView.SetExtentsToBoard
End Sub
```

See Also

View.Change,View.SetExtents, View.SetExtentsToAll

View.SetExtentsToSelection

This method sets the current zoom level and view center to show all currently selected objects.

Prototype

SetExtentsToSelection

Arguments

None

Comments

None

Sample

The following sample code sets view parameters so that C1 and C2 components are shown. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set v = ActiveDocument.ActiveView
ActiveDocument.SelectObjects ppcbObjectTypeAll, "*", False
ActiveDocument.Components("C1").Selected = True
ActiveDocument.Components("C2").Selected = True
v.SetExtentsToSelection
End Sub
```

View.SetScale

This method sets the current zoom level and view center.

Prototype

SetScale (*Zoom* as Double, *CenterX* as Double, *CenterY* as Double, *Units* as PPcbUnit)

Arguments

zoom	A new zoom level. Required.
centerX	x-coordinate of new view center. Optional, if omitted, current Center x-coordinate is used.
centerY	y-coordinate of new view center. Optional, if omitted, current Center y-coordinate is used.
units	[Optional] Unit in which the result is to be represented. This optional argument is ppcbUnitCurrent by default.

Return Values

None

Comments

None

Sample

The following sample code zooms in the view. See "Running Code Samples" for more information on running this sample.

```
Sub Main
Set v = ActiveDocument.ActiveView
v.SetScale( v.Zoom * 2 )
End Sub
```

Events

Application.OpenDocument Event

This event occurs after the program opens a new document.

Prototype

Application_OpenDocument()

Arguments

None

Return Values

None

Comments

None

See Also

Application.OpenDocument Method

Application.ProgressChange

This event occurs after the status bar value is changed.

Prototype

Application_ProgressChange

Arguments

None

Return Values

None

Comments

None

Application.Quit Event

This event occurs before the program exits.

Prototype

Application_Quit()

Arguments

None

Return Values

None

Comments

This event occurs before the program exits.

Document.SecurityLimit Event

This event occurs when the database security limits are reached.

Prototype

Document_SecurityLimit ()

Arguments

None

Return Values

None

Comments

None

Document.PositionsChange

This event occurs after the position of a component is changed.

Prototype

Document_PositionsChange()

Arguments

None

Comments

None

See Also

Component.Move, Component.Orientation

Document.Save Event

This event occurs after the document is saved.

Prototype

Document_Save()

Arguments

None

Return Values

None

Comments

None

See Also

Document.Save Method, Document.SaveAs

Document.SelectionChange Event

This event occurs when the current selection changes.

Prototype

Document_SelectionChange()

Arguments

None

Comments

None

See Also

Document.SelectObjects, Objects.Select

View.Change

This event occurs when the current view changes.

Prototype

View_Change()

Arguments

None

Comments

None

See Also

View.Pan,View.SetExtents,View.SetExtentsToBoard,View.SetExtentsToAll

Samples

Running Code Samples

Many topics in this help file provide code samples to illustrate how to use an Automation property, method, or event. You can run code samples either in PADS Layout or outside of PADS Layout.

The code sample is always in the following form:

```
Sub Main
 ' Do something
End Sub
```

For more information on running code samples, see "Troubleshooting the Code Samples" and "Enhancing Sample Code" .

Disclaimer: The code samples in PADS Layout Automation Server Help are *freeware*. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.

Running Code Samples in PADS Layout

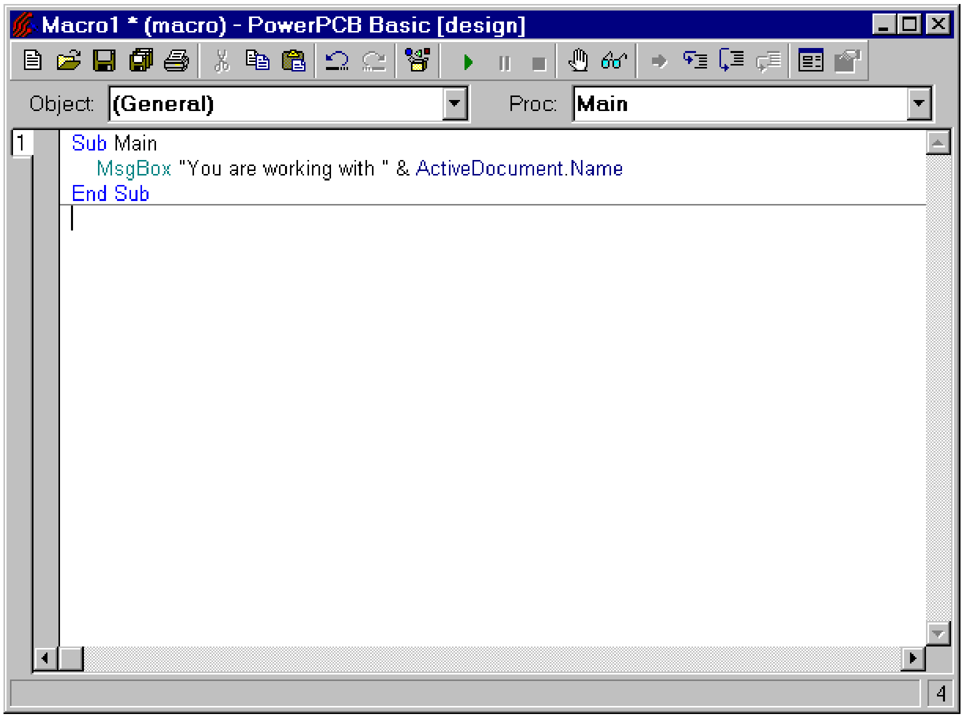
To use the code sample:

1. Select the sample, including the Sub Main and End Sub statements.
2. Copy the sample to the Clipboard using Edit/Copy.

3. Choose Tools/Basic Scripting/Script Editor. The Sax Basic Engine dialog box appears.
4. Paste the code sample into the Sax Basic editor using Edit/Paste.
5. Click the Start button on the toolbar in the Sax Basic Editor dialog box to run the sample.

Figure 46-6 shows the code sample for the Application.ActiveDocument property pasted into the Basic Engine dialog box.

Figure 46-6. Code Sample in Basic Engine Dialog Box



Running Code Samples Outside of PADS Layout

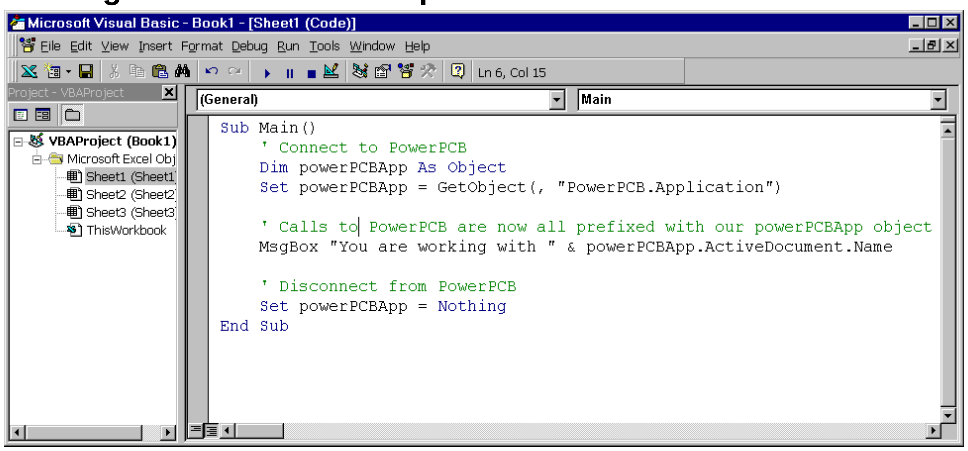
To use the code sample with other Basic scriptable applications (also called host applications) such as Visual Basic, Microsoft Excel, Microsoft Word:

1. Import the Automation server references into the host application. For example, to use Excel as the host application:
2. Choose Tools/Macro/Visual Basic Editor.
3. Choose Tools/References. The References dialog box appears.
4. In the list box, select PADS Layout <latest_version> Type Library.
5. Paste the code sample in the editor of the host application. To do this in Excel:
6. Choose View/Code.

7. Choose Edit/Paste.
8. Add code to the beginning of the code sample to connect the host application to PADS Layout using the Basic GetObject function.
9. Add the object returned by the Basic GetObject function to the beginning of each Automation method and property in the sample code.
10. Add code to the end of the sample to disconnect the application from the program.

Figure 46-7 shows the code sample for the Application.ActiveDocument property pasted and modified in Microsoft Excel's Visual Basic Editor. See the host application help for more information.

Figure 46-7. Code Sample in Excel Visual Basic Editor



Enhancing Sample Code

The code samples in the Automation Server Help are reduced to the minimum number of lines necessary to quickly illustrate how to use Automation properties and methods. You can use these code samples as a base for developing your own PADS Layout features. The following list provides ideas for possible enhancements:

- Add strict type checking, using the Option Explicit declaration at the beginning of the code and declare all variables using the Dim Basic keyword. This ensures that the program can interpret your variables properly and can generate compiling errors when a problem exists.
- Add error checking using the On Error Basic keyword. Error checking improves how code reacts to a run-time error.
- Display information in custom dialog boxes using the UserDialog Basic keyword, instead of displaying in simple dialog boxes created using the MsgBox keyword.

Disclaimer: The code samples in the Automation Server On-line Help are freeware. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and

Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.

Troubleshooting the Code Samples

If a code sample does not run correctly, check the following:

- Make sure a design is open.

Almost all code samples require a design file to be open when you run the sample.

- Check for any assumptions the samples make about the design files.

For example, some code samples assume that component U1 exists, others assume that the Assembly Option MyAssOpt exists, and others assume that at least one Assembly Option exists.

These assumptions are clearly stated in the text preceding each sample. If these assumptions are not true, the sample code will not run properly. You can adapt the sample to your design file.

- If you modified the sample code to run outside of the program, make sure that you properly applied all required changes to the code, as described in "Running Code Samples" .

Common mistakes include forgetting to import references and forgetting to prefix Automation constants.

Disclaimer: The code samples in the Automation Server Help are *freeware*. Mentor Graphics provides these samples as a courtesy to its users. Freeware is provided as is and Mentor Graphics makes no warranties with respect to freeware, either express or implied, including any implied warranties of merchantability or fitness for a particular purpose.

RGL Replacement

Replacing RGL Format Files

PADS Layout includes Automation Server support for all Report Generation Language (RGL) keywords. Examples for each RGL format file (.fmt) are included with the OLE samples as Basic scripts. You can customize these new report format files to fit your output requirements. The files, located in the C:\PADS Projects\Samples\Scripts\Layout\RGL_Samples folder, were created using the Sax Basic Engine. The file name extension used for these files is .bas. You can load these scripts into the Tools/Basic Scripts menu using the Basic Scripts dialog box.

The RGL replacement scripts described below (*.bas) include references to rgl.bas. This file is a library file that contains functions the .bas scripts need to operate. The .bas scripts *must* refer to rgl.bas (within the script) to run.

RGL keywords are replaced with Basic functions and subroutines.

See also: Automation for RGL Top Level Keywords, Automation for RGL SubLevel Keywords, Automation for RGL Field Keywords, New Automation Functions for RGL

Script Names and Report Types

To help you determine the type of output each program provides, Table 46-1 identifies the Sax Basic script name and type of report it generates.

Table 46-1. Sax Basic Script Names and Report Type

RGL Report	Sax Basic File (.bas)	Description
Netlist without pin info	Net List without pin info.bas	Reports signals by netname without pin information.
Netlist with pin info	Net List with pin info.bas	Reports signals by netname with pin information.
Parts List 1	Part List 1.bas	Reports parts by reference designator.
Parts List 2	Part List 2.bas	Reports reference designators by part type.
Test points report	Test points report.bas	Reports test point locations and netname.
Jumper List	Jumper List.bas	Reports jumper locations and netnames.
PowerPCB Format Netlist	PowerPCB V2.0 Format Netlist.bas, PowerPCB V3.0 Format Netlist.bas	Reports a PowerPCB format netlist.
DFT Extended Test Point	DFT Extended test point report.bas	Reports test points by nets, nets without test points, and number of test points per net.

Automation for RGL Top Level Keywords

Table 46-2 and Table 46-3 list the Automation methods and functions that replace RGL top level keywords. Also see "Replacing RGL Format Files," "Automation for RGL SubLevel Keywords," "Automation for RGL Field Keywords,," and "New Automation Functions for RGL.

Table 46-2. Top Level Keywords Replaced by Direct Automation Methods

RGL Keywords	Method	Description
JOBNM	ActiveDocument.Name	Returns the JOB NAME.

Table 46-2. Top Level Keywords Replaced by Direct Automation Methods

COMPCNT	ActiveDocument.Components.Count	Returns the total number of components.
SIGCNT	ActiveDocument.Nets.Count	Returns the total signal count.
BOARDSZ	ActiveDocument.BoardOutlineSurface	Returns the board size in current units.

Table 46-3. Top Level Keywords Replaced by Functions and Subroutines in RGL.BAS

RGL Keywords	Automation Function	Description
TIME	Function GetTime() As String	Returns the current date and time in the format “ddd mmm dd hh:mm:ss yyyy.”
LAYERCNT	Function LayerCnt(doc As Object)	Returns the total number of component and routing layers.
PKGCNT	Function PkgCnt(doc As Object) As Integer	Returns the total number of packages in use on the PCB.
SYMCNT	Function SymCnt(doc As Object) As Integer	Returns the total number of decals in use on the PCB.
EQUIV_IC	Function EquivalentIC(doc As Object) As Double	Uses the total number of pins on all components on the board divided by 14.
BD_DENSITY	Function BdDensity(doc As Object) As Double	Calculates the board density by using the surface area of the PCB divided by the Equivalent IC value.
PSIGCNT	Function PSigCnt(doc As Object) As Integer	Returns the total number of Power Nets on the PCB.
SSIGCNT	Function SSigCnt(doc As Object) As Integer	Returns the total number of signal nets on the PCB, excluding power nets.
TOPCOMPCNT	Function TopCompCnt(doc As Object) As Integer	Returns the total number of components on the top side of the PCB.
BOTCOMPCNT	Function BottomCompCnt(doc As Object) As Integer	Returns the total number of components on the bottom side of the PCB.

Table 46-3. Top Level Keywords Replaced by Functions and Subroutines in RGL.BAS

PADCNT	Function PadCnt(doc As Object) As Integer	Returns the total number of pads (drilled and undrilled) on the PCB.
DRPADCNT	Function DrPadCnt(doc As Object) As Integer	Returns the total number of drilled pads on the PCB (Drill size > 0).
NDPADCNT	Function NDPadCnt(doc As Object) As Integer	Returns the total number of surface mount pads (drill size = 0).

Automation for RGL SubLevel Keywords

Table 46-4 and Table 46-5 list the Automation methods and functions that replace RGL sublevel keywords. Also see the "Automation for RGL Field Keywords."

Lower level keywords are indented.

Also see "Replacing RGL Format Files," "Automation for RGL Top Level Keywords," "Automation for RGL Field Keywords," "New Automation Functions for RGL"

Table 46-4. Sublevel Keywords Replaced by Direct Automation Methods

RGL Keywords	Description
SIGNALS	Search within signals. *' Iterate through all net of document doc like this* *For Each nextNet In doc.Nets* *… ' Do Something with that Net nextNet* *Next nextNet*
SIGNAME	*Net.Name* Name of signals.
PINS	Search within pins. *' Iterate through all pins of net nextNet like this* *For Each nextPin In nextNet.Pins* *… ' Do Something with that Pin nextPin* *Next nextPin*
COMPNAME	*Pin.Component.Name* Component name
PINNUM	*Pin.Number* Pin number

Table 46-4. Sublevel Keywords Replaced by Direct Automation Methods

CONNECTIONS	Search connections. *' Iterate through all connections of document doc like this* *For Each nextConn In doc.Connections* *… ' Do Something with that Connection nextConn* *Next nextConn*
SIGNAME	*Connection.Net.Name* Name of signal(s)
COMP1	*Connection.Pins(1).Component.Name* Reference designator of connection end one
PIN1	*Connection.Pins(1).Number* Pin number of connection end one
COMP2	*Connection.Pins(2).Component.Name* Reference designator of connection end two
PIN2	*Connection.Pins(2).Number* Pin number of connection end two
ROUTSEGS	Search within routes. *' Iterate through all route segs of conn. nextConn like this* *For Each nextSeg In nextConn.RouteSegments* *… ' Do Something with that route segment nextSeg* *Next nextSeg*
END1	First end point of segment *Points = RouteSegment.Points '! First, assign to variant !* *Points(1)*
END2	Second end point of segment *Points = RouteSegment.Points '! First, assign to variant !* *Points(2)*
WIDTH	*RouteSegment.Width* Width of a trace segment
LAYER	*RouteSegment.Layer* Layer number of a trace segment
VIAS	Search within Vias *' Iterate through all vias of connection nextConn like this* *For Each nextVia In nextConn.Vias* *… ' Do Something with that via nextVia* *Next nextVia*
LOCX	*Via.PositionX* X Coordinate of VIA
LOCY	*Via.PositionY* Y Coordinate of via.

Table 46-4. Sublevel Keywords Replaced by Direct Automation Methods

VIANAME		*Via.Type* Via name
PACKAGE		(in RGL.BAS: see PKGCNT)
PKGNAME		Package name *' Use the Component.PartType property*
PKGDSCR		*Component.Attributes("DESCRIPTION").Value* Package description
COMPONENTS		Search within packages. *' Iterate through all components of document doc like this* *For Each nextComp In doc.Components* *… ' Do Something with that Component nextComp* *Next nextComp*
COMPNAME		*Component.Name* Component name
SYMNAME		*Component.Decal* Symbol name
PKGTYPE		*Component.PartType* Package name
ANGLE		*Component.Orientation* Component placement angle
LOCX	X	*Component.PositionX* Coordinate of placement
LOCY	Y	*Component.PositionY* Coordinate of placement
TESTPOINTS		(in RGL.BAS)
TPNAME		*Pin.Name ' for pin test points* Via.Type ' for via test points Test point name. The test point name for component pins is the standard pin name, such as U1.2. The test point name for vias is the via type name, such as STANDARDVIA.
SIGNAME		*Pin.Net ' for pin test points* Via.Net ' for via test points Signal (net) name. If the test point is on an unused component pin, the name *NONE* is returned.
LOCX	X	*Pin.PositionX ' for pin test points* Via.PositionX ' for via test points Coordinate for test point

Table 46-4. Sublevel Keywords Replaced by Direct Automation Methods

LOCY	Y	*Pin.PositionY ' for pin test points* Via.PositionY ' for via test points Coordinate for test point
TESTSIDE		*Pin.TestPoint ' for pin test points* Via.TestPoint ' for via test points Testing side for the test point: TOP or BOTTOM

Table 46-5. Sublevel Keywords Replaced by Functions and Subroutines in RGL.BAS

RGL Keywords	Description
SIGNALS	Search within signals. (Direct automation method)
SIGNAME	Name of signals. (Direct automation method)
PINS	Search within pins. (Direct automation method)
PINTYP	Function PinType(Pin As Object) As String Returns a string value that identifies the type on pin, if it has been assigned by the user: Source="S", Bidirectional="B", OpenCollector="C", OrTieableSource="O", Tristate="T", Load="L", Terminator="Z", Power="P", Ground="G", Undefined="U"
TPASSIGNED	Function TPAssigned(Net As Object) As Boolean Returns a true/false value if a net has a test point on it.
TESTPOINTCNT	Function TPCnt(Net As Object) As Integer Returns the total number of test points on all nets.
TESTPINCNT	Function TPPinCnt(Net As Object) As Integer Returns the total number of pins assigned as test points.
TESTVIACNT	Function TPViaCnt(Net As Object) As Integer Returns the total number of vias as test points.
PACKAGE	Sub GetPackageList(doc As Object, ByRef pkgList() As Package, Optional bSort As Boolean = False Returns a list of all the packages on the PCB. Optional parameter to specify whether to sort the list.
TESTPOINTS	Function TestPoints(doc As Object) As objects Returns a list of test points on component pins, vias, and unused pins.

Automation for RGL Field Keywords

Table 46-6 lists the RGL field keywords that are replaced by functions and subroutines in RGL.bas.

Also see “Replacing RGL Format Files,” “Automation for RGL Top Level Keywords,” “Automation for RGL SubLevel Keywords,” and “New Automation Functions for RGL.”

Table 46-6. Field Keywords Replaced by Functions and Subroutines in RGL.BAS

RGL Keywords	Automation Function	Description
MAXCOLS	Sub MaxCols(cols As Integer)	Calculates the maximum number of columns in a report.
BETWEEN	Sub Between(Optional betweenCol As Integer = 0)	Calculates the space required between different columns of data.
BETWEEN	Sub End_Between()	Calculates the space required between different columns of data.
COLUMNS ,LEADING	Sub Columns(ParamArray formatParam())	Outputs the columns of a report in a table format.
COLUMNS, LEADING	Sub End_Columns()	Outputs the last column of data in a report.
DELIMITER	Not needed	

New Automation Functions for RGL

Table 46-6 lists the new functions and subroutines in RGL.bas to support RGL replacement.

Table 46-7. New Functions and Subroutines in RGL.BAS to Support RGL Replacement

Automation Function	Description
Function Format2(value As Double) As String	Returns floating point numbers as a string to 5 decimal places.
Function NetPinsSortedByConnection(aNet As Object) As objects	Returns a list of pin pairs sorted alphabetically.
Function OpenReport(file As String) As Integer	Opens an output stream to a filename that you supply.

Table 46-7. New Functions and Subroutines in RGL.BAS to Support RGL Replacement

Automation Function	Description
Sub CloseReport()	Closes the currently open output stream.
Sub Out(ParamArray formatParam())	Outputs the formatted data to the open stream.

See Also

Replacing RGL Format Files, Automation for RGL Top Level Keywords, Automation for RGL SubLevel Keywords, Automation for RGL Field Keywords

Chapter 47
The Macro Language

The macro language of this program is similar to standard Visual Basic Script (VBScript) language. It supports most of the VBScript features including the following:

- Variables and arrays of variables
- The full set of standard arithmetic and Boolean Expressions
- Functions and subroutines
- Statements
- Operators
- Objects, properties, and methods
- Automation Support for internal and external automation objects
- Internal Macro Objects

Variables

The macro engine of this program supports variables, which can either be Null or contain values of the following types:

- Numeric
- Logical
- String
- Object

Value types are converted to each other according to these rules, see Table 47-1.

Table 47-1. & Operator Arguments

From...To...	Logical	Numeric	String
Null	False	0	Empty string
Logical	None	0 if False, 1 if True	0 if False, 1 if True
Numeric	False if 0, True if not 0	None	String representation of the numeric value

Table 47-1. & Operator Arguments (cont.)

From...To...	Logical	Numeric	String
String	Converted to Numeric first	If the beginning of the string can be interpreted as a number, then the value of the number is used. Otherwise it is 0.	None

Numeric

The Numeric value represents a floating-point number.

Note: The Numeric and String value types are interchangeable; they automatically convert into one another upon assignment.

Logical

The Logical value can be True or False.

String

The String value represents a character string.

Note: The Numeric and String value types are interchangeable; they automatically convert into one another upon assignment.

Related Topics

Str function

Double

Represents numeric value.

The double and string types are interchangeable; that is, they automatically convert into one another upon assignment.

Object

Objects represent complex entities that are handled through an interface consisting of methods and properties. Objects are different from Numeric and String value types.

Objects may be of two types:

- **Macro objects:** Internal objects that are handled using the macro engine vocabularies, and may or may not have the Automation interface.
- **Automation objects:** Internal or external objects that are handled using Automation.

Syntax

The syntax for both object types is the same:

```
Object.Method arg1, ..., argn
var = Object.Method( arg1, ..., argn )
```

Expressions

The macro engine of this program uses either of the following expressions:

- **Numeric:** Any expression that can be evaluated as a number. Elements of a numeric expression can include any combination of keywords, variables, constants, and operators that result in a number.
- **String:** Any expression that evaluates a sequence of adjacent characters. Elements of a string expression can include a string, a string literal, or a string variable.

Operators

The Macro engine of this program uses the following operators:

- & Operator
- * Operator
- + Operator
- / Operator
- - Operator
- ^ Operator
- = Operator
- And Operator
- Comparison Operators
- Mod Operator
- Not Operator
- Or Operator
- Xor Operator

& Operator

Forces string concatenation of two expressions.

Syntax

```
result = expression1 & expression2
```

Arguments

The & operator has these arguments:

result	Required Any numeric variable
expression1	Required Any expression When the expression is not a string, it is converted to a string.
expression2	Required Any expression When the expression is not a string, it is converted to a string.

Example

```
S = "abc" & "123"
```

* Operator

Multiplies two numbers.

Syntax

```
result = number1 * number2
```

Arguments

The * operator has these arguments::

result	Required Any numeric variable
number1	Required Any expression. If the expression value is not numeric, it is converted to a numeric value.
number2	Required Any expression. If the expression value is not numeric, it is converted to a numeric value.

Example

```
X = y * z
```

+ Operator

Sums two numbers.

Syntax

```
result = expression1 + expression2
```

Arguments

The + operator syntax has these arguments:

result	Required Any numeric variable
expression1	Required Any expression
expression2	Required Any expression

Note: When you use the + operator, you may not be able to determine whether addition or string concatenation will occur. To force string concatenation, use the & operator instead. This will eliminate ambiguity and provide self-documenting code.

Table 47-2 describes the behavior of the + operator for the three combinations of types.

Table 47-2. + Operator Behavior

Arguments	Action
Both expressions are numeric	Add
Both expressions are string	Concatenate
One expression is numeric and the other is a string	Add

Example

```
X = y + z
```

/ Operator

Divides one number by a second number and returns a floating-point result.

Syntax

```
result = number1 / number2
```

Arguments

The / operator has these arguments:

result	Required Any numeric variable
number1	Required Any expression. If the expression value is not numeric, it is converted to a numeric value.
number2	Required Any expression. If the expression value is not numeric, it is converted to a numeric value.

Example

```
x = y/z
```

- Operator

Finds the difference between two numbers or indicates the negative value of a numeric expression.

Syntax 1

```
result = number1 - number2
```

The - operator is the arithmetic subtraction operator, which finds the difference between two numbers.

Syntax 2

```
-number
```

The - operator is the unary negation operator which indicates the negative value of an expression.

Arguments

The - operator has these arguments:

result	Required Any numeric variable
number1	Required Any expression. If the expression value is not numeric, it is converted to a numeric value.
number2	Required Any expression. If the expression value is not numeric, it is converted to a numeric value.

Example 1

```
x = y - z
```

Example 2

```
-x
```

= Operator

Assigns a value to a variable or property.

Syntax

variable = value

Arguments

The = operator has these arguments:

variable	Can only be a variable or a writable property. Can be a simple scalar variable or an element of an array.
Value	Any numeric expression, string expression, literal, or constant

Example

```
a = 1
```

^ Operator

Raises a number to the power of an exponent.

Syntax

```
result = number ^ exponent
```

Arguments

The ^ operator has these arguments:

result	Required Any numeric variable
number	Required Any expression
exponent	Required Any numeric expression A number can be negative only if the exponent is an integer.

When more than one exponentiation is performed in a single expression, the ^ operator is evaluated as it is encountered from left to right.

Example

```
x = y ^ z
```

And Operator

Performs a logical conjunction of two expressions.

Syntax

result = expression1 **And** expression2

Arguments

The And operator has these arguments:

result Required
Any numeric variable

expression1 Required
Any expression. If the expression value is not logical, it is converted to a logical value.

expression2 Required
Any expression. If the expression value is not logical, it is converted to a logical value.

Table 47-3 illustrates how the result is determine.

Table 47-3. And Operator Results

If expression1 is	And expression2 is	The result is
True	True	True
True	False	False
False	True	False
False	False	False

Example

```
a = b And c
```

Comparison Operators

Compare expressions.

Syntax

```
result = expression1 comparisonoperator expression2
```

Arguments

Comparison operators have these arguments:

result	Required Any variable
expression1	Required Any expression
expression2	Required Any expression
comparisonoperator	Required Can be any comparison operator. For more information, see Table 47-4 below.

Table 47-4 lists the comparison operators and the conditions that determine whether result is True, False, or Null.

Table 47-4. Comparison Operators and Results

Comparison Operator	True if	False if
< (Less than)	*expression1 < expression2*	*expression1 >= expression2*
<= (Less than or equal to)	*expression1 <= expression2*	*expression1 > expression2*
> (Greater than)	*expression1 > expression2*	*expression1 <= expression2*
>= (Greater than or equal to)	*expression1 >= expression2*	*expression1 < expression2*
= (Equal to)	*expression1 = expression2*	*expression1 <> expression2*
<> (Not equal to)	*expression1 <> expression2*	*expression1 = expression2*

Example

```
b = 1 > 2
```

Mod Operator

Divides one number by a second number and returns only the remainder. The modulus (remainder) operator rounds floating-point numbers to integers.

Syntax

```
result = number1 Mod number2
```

Arguments

The Mod operator has these arguments:

result	Required Any numeric variable
number1	Required Any numeric expression
number2	Required Any numeric expression

Example

```
x = y Mod z
```

Not Operator

Performs a logical negation on an expression.

Syntax

```
result = Not expression
```

Arguments

The Not operator has these arguments:

result Required
Any numeric variable

expression Required
Any expression. If the expression value is not logical, it is converted to a logical value.

Table 47-5 illustrates how result is determined:

Table 47-5. Not Operator Results

If expression is	Then result is
True	False
False	True

Example

```
x = Not y
```

Or Operator

Performs a logical disjunction on two expressions.

Syntax

```
result = expression1 Or expression2
```

Arguments

The Or operator has these arguments:

result Required
Any numeric variable

expression1 Required
Any expression. If the expression value is not logical, it is converted to a logical value.

expression2 Required
Any expression. If the expression value is not logical, it is converted to a logical value.

Table 47-6 illustrates how result is determined:

Table 47-6. Or Operator Results

If expression1 is	And expression2 is	Then result is
True	True	True
True	False	True
False	True	True
False	False	False

Example

```
x = y Or z
```

Xor Operator

Performs a logical exclusion on two expressions.

Syntax

```
[result =] expression1 Xor expression2
```

Arguments

The Xor operator has these arguments:

result Optional
Any numeric variable

expression1 Required
Any expression. If the expression value is not logical, it is converted to a logical value.

expression2 Required
Any expression. If the expression value is not logical, it is converted to a logical value.

Table 47-7 illustrates how result is determined::

Table 47-7. Xor Operator Results

If expression1 is	And expression2 is	Then result is
True	True	False
True	False	True
False	True	True
False	False	False

Example

```
x = y Xor z
```

Statements

The macro engine of this program supports the following VBScript and other statements:

- Call
- Close
- Dim
- Do...Loop
- For-Next

- Function
- If...Then...Else statement
- Input #
- Modal
- Open
- Print #
- ReDim
- Set
- Sub
- While...Wend
- Width #

Call

Transfers control to a sub procedure or function procedure.

Syntax

```
[Call] name [argumentlist]
```

When you specify the Call keyword to call a procedure that requires arguments, you must enclose argumentlist in parentheses. See example below.

Arguments

The Call statement has these arguments:

Call	Optional keyword You are not required to use the Call keyword when calling a procedure. If you omit the Call keyword, you must also omit the parentheses around argumentlist. If you use either the Call syntax to call any intrinsic or user-defined function, the function's return value is discarded.
name	Required Name of the procedure to call
argumentlist	Optional Comma-delimited list of variables, arrays, or expressions to pass to the procedure.

Example

```
Call MyProc(0)
```

Related Topics

Statements

Close

Concludes input/output (I/O) to a file opened using the Open statement.

When you close files that were opened for Output or Append, the final buffer of output is written to the operating system buffer for that file; All buffer space associated with the closed file is released; and the association of a file with its file number ends.

Syntax

```
Close [filenumberlist]
```

Arguments

The Close statement has this argument:

filenumberlist	Optional Can be one or more file numbers using the following syntax, where *filenumber* is any valid file number: `[[#]filenumber] [, [#]filenumber] . . .`

If you omit filenumberlist, all active files opened by the Open statement are closed.

Example

```
close #1
```

Related Topics

Statements

Dim

Declares variables and allocates storage space.

Syntax

```
Dim varname[([subscripts])] [,varname[([subscripts])]] . . .
```

Arguments

The Dim statement has these arguments:

varname	Required The variable name follows standard variable-naming conventions
subscripts	Optional Dimensions of an array variable The subscripts argument uses the following syntax: [lower To] upper [, [lower To] upper] . . . When not explicitly stated, the lower bound is zero.

You can also use the Dim statement, with empty parentheses, to declare a dynamic array. After declaring a dynamic array, use the ReDim statement to define the number of dimensions and elements in the array.

Example

```
Dim x(10), y(20)
```

Related Topics

Statements

Do...Loop

Repeats a block of statements while a condition is True or until a condition becomes True.

Syntax 1

```
Do [{While | Until} condition]
[statements]
[Exit Do]
[statements]
Loop
```

Syntax 2

```
Do
[statements]
[Exit Do]
[statements]
Loop [{While | Until} condition]
```

Arguments

The Do Loop statement has these arguments:

condition	Optional One or more of the following two types of expressions: A numeric expression that evaluates to True or False A string expression that evaluates to True or False When *condition* is Null, *condition* is treated as False.
statements	One or more statements that are repeated while, or until, *condition* is True

You may place any number of Exit Do statements anywhere in the Do…Loop statement as an alternative way to exit a Do Loop statement. Exit Do is often used after evaluating some condition, in which case the Exit Do statement transfers control to the statement immediately following the Loop.

When used within nested Do Loop statements, Exit Do transfers control to the loop that is nested one level above the loop where Exit Do occurs.

Example

```
Do while i < 10
i = i + 1
loop
```

Related Topics

Statements

For-Next

Repeats a group of statements a specified number of times.

Syntax

```
For counter = start To end [Step step]
[statements]
[Exit For]
[statements]
Next [counter]
```

Arguments

The For-Next statement has these arguments:

counter	Required Numeric variable used as a loop counter. *Counter* cannot be a Boolean element or an array element.
start	Required Initial value of *counter*
end	Required Final value of *counter*
step	Optional The number by which *counter* is incremented each time control passes through the loop. If not specified, step defaults to one.
statements	Optional One or more statements between For and Next that are executed the *counter*-specified number of times

The *step* argument can be either positive or negative. The value of step determines loop processing as described in Table 47-8.

Table 47-8. For-Next Statement Loop Counter

Value	Loop executes if
Positive or zero	counter <= end
Negative	counter >= end

After all statements in the loop execute, *step* is added to *counter*. At this point, either the statements in the loop execute again (based on the same test that caused the loop to execute initially), or the loop is exited and execution continues with the statement following the Next statement.

You can place any number of Exit For statements anywhere in the loop as alternative ways to exit. Exit For is often used after evaluating a condition to transfer control to the statement immediately following Next.

You can nest For-Next statements by placing one For-Next statement within another. Give each statement a unique variable name as its *counter*. The following construct is correct:

```
For A = 1 To 10
  For B = 2 To 20
  For C = 3 To 30
  ...
  Next C
  Next B
Next A
```

Example

```
for i = 0 to 10 step 2
s = sti
next i
```

Related Topics

Statements

Function

Declares the name, arguments, and code that form the body of a Function procedure.

Like a sub procedure, a function procedure is a separate procedure that can take arguments, perform a series of statements, and change the values of its arguments. However, unlike a sub procedure, a function procedure can be used on the right side of an expression in the same way you use any intrinsic function, such as Sqr, Cos, or Chr, when you want to use the value returned by the function.

There are two categories of variables you can use in function procedures:

- **Explicitly declared variables within the procedure:** These variables are always local to the procedure and use the Dim statement or the equivalent. The values of local variables in a function are not preserved between calls to the procedure.
- **Not explicitly declared variables within the procedure:** These variables are also local, unless they are explicitly declared at some higher level outside the procedure.

Syntax

```
Function name [(arglist)]
[statements]
[name = expression]
[Exit Function]
[statements]
[name = expression]
End Function
```

The Exit Function statement causes an immediate exit from a function procedure. Program execution continues with the statement that follows the statement that called the function procedure. You can add any number of Exit Function statements in a function procedure.

Arguments

The Function statement has these arguments:

name	Required Name of the function procedure Follows standard variable-naming conventions. To return a value from a function, assign a value to the function name. You can assign values to function names anywhere in the procedure. If no value is assigned to name, the procedure returns an empty value.
arglist	Optional List of variables representing arguments that are passed to the function procedure when the procedure is called. Use commas to separate multiple variables.

statements Optional
Any group of statements to execute within the body of the function procedure.

expression Return value of the function procedure.

The arglist argument has the following syntax:

```
[ByVal | ByRef] varname[( )]
```

Table 47-9 describes the arglist syntax elements:

Table 47-9. Function Statement arglist Syntax

Part	Description
ByVal	Indicates that the argument is passed by value.
ByRef	Indicates that the argument is passed by reference.
Varname	Name of the variable representing the argument. Follows standard variable naming conventions.

You cannot define a function procedure inside any other procedure such as a sub procedure or another function procedure.

For specific information about calling a function procedure, see the Call statement.

Example

The following example shows how to assign a return value to a function named Example. In this case, False is assigned to the name to indicate that a certain condition is not met.

```
Function Example()
    ...
' Value not found. Return False.
    If ConditionNotMet Then
Example = False
        Exit Function
    End If
    ...
Example = True
End Function
```

Related Topics

Statements

If...Then...Else statement

May execute a group of statements, based on the value of an expression.

Syntax

```
If condition Then [statements] [Else [elsestatements]]
```

Block Syntax

```
If condition Then
[statements]
[ElseIf condition-n Then
[elseifstatements]] ...
[Else
[elsestatements]]
End If
```

Arguments

The If...Then...Else statement has these arguments::

condition	Required Any expression. If the expression is not Logical, it is converted to Logical.
statements	Optional in block form; required in single-line form that has no Else clause One or more statements separated by colons; executed when condition is True.
condition-n	Optional Any logical expression. If the expression is not Logical, it is converted into a Logical expression.
elseifstatements	Optional One or more *statements* executed when associated *condition-n* is True
elsestatements	Optional One or more *statements* executed when no previous condition or when *condition-n* is True

You can use the single-line syntax for short, simple tests. For more structure and flexibility use the block syntax. The block syntax can also be easier to read, maintain, and debug.

With the single-line syntax, you can execute multiple statements as the result of an If...Then decision. All statements must be on the same line and separated by colons, as in the following statement:

```
If A > 10 Then A = A + 1 : B = B + A : C = C + B
```

A block If statement must be the first statement on a line. The Else, ElseIf, and End If parts of the statement can be preceded by only a line number or a line label. The block If statement must end with an End If statement.

To determine whether or not a statement is a block If statement, examine what follows the Then keyword. If anything other than a comment appears after Then on the same line, the statement is treated as a single-line If statement.

The Else and ElseIf clauses are both optional. You can have as many ElseIf clauses as you want in a block If statement, but none can appear after an Else clause. Block If statements can be nested.

During the execution of a block If statement, *condition* is tested. When *condition* is True, the statements following Then are executed. When condition is False, any ElseIf condition is evaluated in turn. When a True condition is found, the statements immediately following the associated Then are executed. If none of the ElseIf conditions are True (or if there are no ElseIf clauses), the statements following Else are executed. After executing the statements following Then or Else, execution continues with the statement following End If.

Examples

```
If x < y then x = y
```

and

```
If x < y then
x = y
End if
```

Related Topics

Statements

Input

Reads data from an open text file and assigns the data to variables. Use this statement only with files opened in Input mode. When read, string or numeric data is assigned to variables without modification.

Syntax

```
Input #filenumber, varlist
```

Arguments

The Input # statement has these arguments:

filenumber	Required Can be any valid file number
varlist	Required Can be a comma-delimited list of variables that are assigned values read from the file. This cannot be an array or object variable. However, you may use variables that describe an element of an array.

Related Topics

Statements

Modal

Opens access to a variable.

While a modal dialog is open in a PADS macro, access to all variables outside the open dialog is blocked—only controls within the open dialog are accessible. To make a variable accessible in the context of any open modal dialog, declare it “modal” at the beginning of the macro file.

Syntax

```
modal variablename
```

Arguments

The modal keyword has these arguments:

variablename	Required The name of the variable you want to make accessible.

Example

modal docname

Open

Enables input and output to a file.

You must open a file before performing any I/O operation on it. Open allocates an I/O buffer for the file and determines the mode of access to use with the buffer. If the file is already opened by another process and the specified type of access is not allowed, the Open operation fails and an error occurs.

Syntax

```
Open pathname For mode [Access access] [lock] As [#]filenumber
[Len=reclength]
```

Arguments

The Open statement has these arguments:

pathname	Required String expression that specifies a filename. May also include folder and drive names.
mode	Required Keyword specifying the file access mode: Append, Binary, Input, Output, or Random. If mode is unspecified, the file is opened for Random access.
access	Optional Keyword specifying the operations permitted on the open file: Read, Write, or Read Write
lock	Optional Keyword specifying the operations restricted on the open file by other processes: Shared, Lock Read, Lock Write, and Lock Read Write

If the file specified by pathname does not exist, it is created when a file is opened for Append, Binary, Output, or Random access modes.

Example

```
Open "C:\data.txt" for read as #1
```

Related Topics

Statements

Print

Writes formatted data to a sequential file.

Syntax

```
Print # filenumber, [outputlist]
```

Arguments

The Print # statement has these arguments:

filenumber	Required Any valid file number
outputlist	Optional An expression or list of expressions to print. Nothing is written to the file when *outputlist* is empty. However, when *outputlist* is Null, Null is written to the file.

If you omit *outputlist* and include only a list separator after *filenumber*, a blank line prints to the file.

You can separate multiple expressions with either a space or a semicolon. A space has the same effect as a semicolon.

Example

```
print #1, a, b, c
```

outputlist has the following syntax:

```
[{Spc(n) | Tab[(n)]}] [expression] [charpos]
```

Table 47-10 describes the outputlist syntax elements:

Table 47-10. Print # Statement outputlist Syntax

Setting	Description
Spc(n)	Inserts space characters in the output, where *n* is the number of spaces to insert.
Tab(n)	Positions the insertion point at an absolute column number, where *n* is the column number. Use Tab with no argument to position the insertion point at the beginning of the next print zone. Because Print # writes an image of the data to the file, you must delimit the data so it prints correctly. If you use Tab with no arguments to move the print position to the next print zone, Print # also writes the spaces between print fields to the file.

Table 47-10. Print # Statement outputlist Syntax

expression	Numeric or string expressions to print. You can separate multiple expressions with either a space or a semicolon.
charpos	Specifies the insertion point for the next character. If you omit *charpos*, the next character prints on the next line. Use a semicolon to position the insertion point immediately after the last displayed character.

Data written with Print # is usually read from a file with Input #.

Example

```
Print #1, a, Spc(3), b
```

Related Topics

Statements

ReDim

Reallocates storage space for dynamic array variables.

Use ReDim to size a dynamic array that has already been declared using the Dim statement with empty parentheses (without dimension subscripts). You can use ReDim repeatedly to change the number of elements and dimensions in an array.

Note: If you re-declare a dimension for an array variable whose size was explicitly specified in a Dim statement, an error occurs.

Syntax

```
ReDim [Preserve] varname(subscripts) [, varname(subscripts)] . . .
```

Arguments

The ReDim statement has these arguments:

Preserve	Optional Keyword used to preserve the data in an existing array when you change the size of the last dimension
varname	Required Name of the variable; follows standard variable-naming conventions
subscripts	Required Dimensions of an array variable The subscripts argument uses the following syntax: `[lower To] upper [,[lower To] upper] . . .` When not explicitly stated, lower bound is zero.

Note: If you make an array smaller than it was, any data in the eliminated elements is lost.

When using *Preserve* you can resize only the last array dimension and you cannot change the number of dimensions. For example, if your array has only one dimension, you can resize that dimension because it is the last and only dimension.

However, if your array has two or more dimensions, you can change the size of only the last dimension while preserving the contents of the array. The following example shows how you can increase the size of the last dimension of a dynamic array without erasing existing data contained in the array:

```
ReDim X(10, 10, 10)
. . .
ReDim Preserve X(10, 10, 15)
```

When you use *Preserve* you can change the size of the array only by changing the upper bound. Changing the lower bound causes an error.

Example

```
ReDim x(150)
```

Related Topics

Statements

Set

Assigns an object reference to a variable or property.

Syntax

```
Set objectvar = {objectexpression | Nothing}
```

Arguments

The Set statement has these arguments:

objectvar	Required Name of the variable, or property; follows standard variable-naming conventions
objectexpression	Required Expression, which consists of the name of an object, another declared variable of the same object type, or a function or method that returns an object of the same object type
Nothing	Optional Discontinues association of *objectvar* with any specific object. Assigning Nothing to *objectvar* releases all the system and memory resources associated with the previously referenced object when no other variable refers to it.

To be valid, *objectvar* must be of the same object type as the object being assigned to it.

The Dim and ReDim statements declare only the variable name, which refers to an object. No actual object is referred to unless the Set statement assigns a specific object.

When you use Set to assign an object reference to a variable, a reference to the object is created - not a copy of the object. More than one object variable can refer to the same object. Because such variables are references to the object rather than copies of the object, any change in the object is reflected in all variables that refer to it.

Example

```
Set obj = application
```

Related Topics

Statements

Sub

Declares the name, arguments, and code that form the body of a sub procedure.

Like a function procedure, a sub procedure is a separate procedure that can take arguments, perform a series of statements, and change the value of its arguments. However, unlike a function procedure, which returns a value, a sub procedure cannot be used in an expression.

There are two categories of variables you can use in sub statements:

- **Explicitly declared variables within the procedure:** These variables are always local to the procedure and use the Dim statement or the equivalent. The value of local variables in a Sub procedure is not preserved between calls to the procedure.
- **Not explicitly declared variables within the procedure:** These variables are also local, unless they are explicitly declared at some higher level outside the procedure.

Syntax

```
Sub name [(arglist)]
[statements]
[Exit Sub]
[statements]
End Sub
```

Arguments

The Sub statement has these arguments:

name	Required The Sub procedure name Follows standard variable-naming conventions.
arglist	Optional List of variables representing arguments that are passed to the sub procedure when the sub procedure is called. Use commas to separate multiple variables.
statements	Any group of statements to execute within the body of the sub procedure. The Exit Sub statement causes an immediate exit from a sub procedure. Program execution continues with the statement following the statement that called the sub procedure. Any number of Exit Sub statements can appear anywhere in a sub procedure.

Example

The *arglist* argument has the following syntax:

```
[ByVal | ByRef] varname[( )]
```

Table 47-11 describes the arglist syntax elements::

Table 47-11. Sub Statement arglist Syntax

Part	Description
ByVal	Indicates that the argument is passed by value.
ByRef	Indicates that the argument is passed by reference.
varname	Name of the variable representing the argument. Follows standard variable-naming conventions.

Note: You cannot define a sub procedure inside any other procedure such as a function or another sub procedure.

For specific information about calling sub procedures, see the Call statement.

Related Topics

Statements

While...Wend

Executes a series of statements as long as a given condition is True.

Syntax

```
While condition
[statements]
Wend
```

Arguments

The While Wend statement has these arguments:

condition	Required Any expression. If the expression is not Logical, it is converted to a Logical type.
statements	Optional One or more statements executed while *condition* is True

If *condition* is True, all *statements* are executed until the Wend statement is encountered. Control then returns to the While statement and *condition* is again checked. If *condition* is still True, the process is repeated. If *condition* is not True, execution resumes with the statement following the Wend statement.

You can nest While…Wend statements to any level. Each Wend matches the most recent While statement.

Example

```
While i < 10
i = i + 1
Wend
```

Related Topics

Statements

Width

Assigns an output line width to a file opened using the Open statement.

Syntax

```
Width #filenumber, width
```

Arguments

The Width # statement has these arguments:

filenumber	Required Any valid file number
width	Required Numeric expression in the range 0 to 255, inclusive. Indicates how many characters appear on a line before a new line starts. If width equals 0, there is no limit to the length of a line. The default value for width is 0.

Example

```
Width #2, 100
```

Related Topics

Statements

Functions

The macro engine of this program currently supports the following embedded functions:

- Asc
- Atn
- Chr
- Command
- Cos
- CreateObject
- CurDir
- Dir
- DoEvents
- Environ

- Environ
- Eof
- Exp
- GetObject
- GetTmpFileName
- InStr
- InStrRev
- Left
- Len
- Mid
- MkDir
- MoveFile
- MsgBox
- Right
- Sin
- Spc
- Str
- Tab
- Val

Asc

This function returns an integer representing the character code that corresponds to the first letter in a string.

Syntax

```
Asc(string)
```

Arguments

The Atn function has this argument:

string	Required Any valid string expression If the string contains no characters, a run-time error occurs.

Example

```
i = Asc("abc")
```

Atn

This function returns a Double specifying the arctangent of a number.

The range of the result is - pi/2 to pi/2 radians. To convert degrees to radians, multiply degrees by pi/180. To convert radians to degrees, multiply radians by 180/pi.

The Atn function takes the ratio of two sides of a right triangle and returns the corresponding angle in radians. The ratio is the length of the side opposite the angle divided by the length of the side adjacent to the angle.

Atn is the inverse trigonometric function of Tan, which takes an angle as its argument and returns the ratio of two sides of a right triangle. Do not confuse Atn with cotangent, which is the inverse of a tangent (1/tangent).

Syntax

```
Atn(number)
```

Arguments

The Atn function has this argument:

number	Required Any expression. If the expression is not Logical, it is converted to a Logical type.

Example

```
f = Atn(2)
```

Chr

This function returns a string containing the character associated with the specified character code.

Syntax

```
Chr(charcode)
```

Arguments

The Chr function has this argument:

charcode	Required A long that identifies a character. Values from 0 to 31 are standard, nonprintable ASCII codes. For example, Chr(10) returns a linefeed character. The normal range for charcode is 0 to 255, inclusive.

Example

```
c = chr(64)
```

Command

This function returns the command line used to start the program, including the path to the executable file and any subsequent arguments.

Syntax

```
Command
```

When you launch the program from the command line, the command line is available to Macro scripts.

Example

Assuming that the program is launched by the following command:

```
BlazeRouter log:logfile.log preview.pcb
```

The Command function returns:

```
"C:\Program Files\Mentor Graphics\PADS\<latest
release>\Programs\BlazeRouter.exe log:logfile.log preview.pcb"
```

Cos

This function returns a Double specifying the cosine of an angle.

The Cos function takes an angle and returns the ratio of the length of the side adjacent to the angle divided by the length of the hypotenuse. The result lies in the range -1 to 1. To convert degrees to radians, multiply degrees by pi/180. To convert radians to degrees, multiply radians by 180/pi.

Syntax

```
Cos(number)
```

Arguments

The Cos function has this argument

number	Required A double or any valid numeric expression expressing an angle in radians

Example

```
x = Cos(1.57)
```

CreateObject

This function creates and returns a reference to an ActiveX object.

Syntax

```
CreateObject(class,[servername])
```

Arguments

The CreateObject function syntax has these arguments:

class	Required String The application name and the class of the object to create
servername	Optional String The name of the network server where the object will be created If the remote server does not exist or is unavailable, a run-time error occurs.

The *class* argument uses the syntax *appname.objecttype* and has these elements:

appname	Required String The name of the application providing the object
objecttype	Required String The type or class of object to create

To create an ActiveX object, assign the object returned by CreateObject to an object variable.

Use CreateObject when there is no current instance of the object. If an instance of the object is already running, a new instance is started, and an object of the specified type is created. To use the current instance, or to start the application and have it load a file, use the GetObject function.

If an object has registered itself as a single-instance object, only one instance of the object is created, no matter how many times CreateObject is executed.

Example

```
set obj = CreateObject("PowerPCB.Application")
```

CurDir

This function returns a variant string representing the current path.

Syntax

```
CurDir [(drive)]
```

Argument

The CurDir function has this argument:

drive	Optional Any string expression that specifies an existing drive. If no drive is specified or if *drive* is a zero-length string (" "), CurDir returns the path for the current drive.

Example

```
s = CurDir("d:")
```

Dir

This function returns a string representing the name of a file or folder that matches a specified pattern, file attribute, or drive volume label.

Syntax

```
Dir(pathname)
```

Arguments

The Dir function has this argument:

pathname	Optional String expression that specifies a file name. May also include folder and drive names. A zero-length string ("") is returned if pathname is not found.

Dir supports the use of multiple character (*) and single character (?) wildcards to specify multiple files.

Examples

To get the first file in the c: drive root:

```
Dir("C:\*.*")
```

To get the next file in the same path:

```
Dir
```

To start another search in C:\PADS Projects\Samples:

```
Dir("C:\PADS Projects\Samples\*.*")
```

DoEvents

This function passes control to the operating system. The operating system returns control after it finishes processing the events in its queue.

Syntax

```
DoEvents()
```

DoEvents may be useful if the macro is performing long calculations. Inserting DoEvents calls every second or more may prevent the accumulation of unprocessed events in the queue.

Environ

This function returns the string associated with an operating system environment variable.

Syntax

```
Environ [(envstring)]
```

Arguments

The Environ function has this argument:

Envstring	Optional String expression containing the name of an environment variable

If envstring can't be found in the environment-string table, a zero-length string ("") is returned.

Environ returns the text assigned to the specified envstring; that is, the text which follows the equal sign (=) in the environment-string table for that environment variable.

Example

```
s = Environ ("path")
```

Eof

This function returns an integer, which represents the Boolean value True when the end of a file opened for random or sequential input has been reached.

Use the Eof function to avoid the error generated by attempting to get input after the end of a file.

Table 47-12 lists the Eof function returned values:

Table 47-12. Eof Function Returned Values

Condition	**Eof function returns**
Before end of file is reached	False
After end of file is reached	True
File opened for random access and Get statement is able to read an entire record	False
File opened for random access and Get statement is not able to read an entire record	True
File opened for binary access and Get statement is able to read an entire record	False
File opened for binary access and Get statement is not able to read an entire record	True
File opened for output	True

Syntax

```
Eof [(filenumber)]
```

Arguments

The Eof function has this argument:

filenumber Optional
An integer containing any valid file number

Example

```
If Eof (1) then ....
```

Exp

This function returns a Double specifying e (the base of natural logarithms) raised to a power. The constant *e* is approximately 2.718282.

Note: The Exp function complements the Log function and may be referred to as the antilogarithm.

Syntax

```
Exp(number)
```

Arguments

The Exp function has this argument:

number	Required A double or any valid numeric expression. When the value of *number* exceeds 709.782712893, an error occurs.

Example

```
x = exp(y)
```

GetObject

This function returns a reference to an object provided by an ActiveX component.

Syntax

```
GetObject([pathname] [, class])
```

Arguments

The GetObject function has these arguments:

pathname	Optional Variant string The full path (folder, and drive) and name of the file containing the object to retrieve. If *pathname* is omitted, class is required.
class	Optional Variant string A string representing the class of the object

The *class* argument uses the syntax *appname.objecttype*, which has these elements:

appname	Required Variant string The name of the application providing the object
objecttype	Required Variant string Type or class of object to create

Use the GetObject function to access an ActiveX object from a file and assign the object to an object variable. Use the Set statement to assign the object returned by the GetObject function to the object variable.

Example

```
obj = GetObject(, "PowerPCB.Application")
```

GetTmpFileName

This function returns a string specifying a new file name guaranteed to be unique in the folder identified by the string argument.

Syntax

```
GetTmpFileName(string)
```

Arguments

The GetTmpFileName function has this argument:

string	Required A string expression

Example

```
s = GetTmpFileName("d:\tmp")
```

InStr

This function returns the position of the first occurrence of one string within another string.

Syntax

```
InStr([start, ]string1, string2)
```

Arguments

The InStr function has these arguments:

start Optional
Numeric expression that sets the starting position for each search
When *start* is omitted, the search begins at the first character position.

string1 Required
String expression to search

string2 Required
String expression to find

Return Values

Table 47-13. InStr Function Return Values

If	InStr returns
string1 is zero-length	0
string2 is zero-length	start
string2 is not found	0
string2 is found within string1	position at which a match is found
start > string	0

Example

```
i = InStr(1,"cbc","c")
```

InStrRev

This function returns the position of an occurrence of one string within another, from the end of the string.

Syntax

```
InStrRev(string1, string2, [start])
```

Arguments

The InStrRev function has these arguments:

string1 Required
String expression being searched

string2 Required
String expression to find

start Optional
Numeric expression that sets the starting position for each search
If omitted, the search begins at the last character position.

Return Values

Table 47-14. InStrRev Function Return Values

If	InStrRev returns
string1 is zero-length	0
string2 is zero-length	Start
string2 is not found	0
string2 is found within string1	Position at which match is found
start > Len(string2)	0

Example:

The following sample code returns "4."

```
InStrRev("abcdbc", "bc")
```

Left

This function returns a specified number of characters from the left side of a string.

Syntax

```
Left(string, length)
```

Arguments

The Left function has these arguments:

string	Required A string expression from which the characters from the left side are returned
length	A numeric expression indicating the number of characters to return If *length* is 0, Left returns a zero-length string (" "). If *length* is greater than or equal to the number of characters in the string, Left returns the entire string.

Example

The following sample code returns "ab."

```
Left("abcd", 2)
```

Len

This function returns the number of characters in a string (the length of the string).

Syntax

```
Len(string)
```

Arguments

The Len function has this argument:

string	Required Any valid string expression

Example

The following sample code returns 4.

```
Len("abcd")
```

Mid

This function returns a specified number of characters from a string.

Syntax

```
Mid (string, start, [length])
```

Arguments

The Mid function has these arguments:

string	Required A string expression from which the characters are returned
start	Required The character position in *string* at which the part to return begins. If start is greater than the number of characters in *string*, Mid returns a zero-length string (" ").
length	Optional The number of characters to return. If omitted, when there are less characters in the string (including the character at start), than *length*, *Mid* returns all of the characters from the start position to the end of the string.

Example

The following sample code returns "cd."

```
Mid("abcdbc", 3, 2)
```

MkDir

This function creates a new folder.

Syntax

```
MkDir (path)
```

Arguments

The MkDir function has this argument:

path Required
A string expression that identifies the folder to create. The path may include the drive. When no drive is specified, MkDir creates the new folder on the current drive.

Example

```
MkDir ("D:\newdir")
```

MoveFile

This function moves the file identified by path1 argument to the location specified in path2.

Syntax

```
MoveFile(path1, path2)
```

Arguments

The MoveFile function has these arguments:

path1 Required
A string expression

path2 Required
A string expression

Example

```
MoveFile("C:\data.bin","D:\")
```

MsgBox

This function displays a message in a dialog box, waits for the user to click a button, and returns an integer indicating which button the user clicked.

Syntax

```
MsgBox(prompt [, buttons] [, title])
```

Arguments

The MsgBox function has these arguments:

prompt Required
String expression displays as the message in the dialog box
The maximum length of prompt is 1024 characters, depending on the width of the characters used. If *prompt* consists of more than one line, you can separate the lines using a carriage return character (Chr(13)), a linefeed character (Chr(10)), or carriage return - linefeed character combination (Chr(13) & Chr(10)) between each line.

buttons Optional
Numeric expression which is the sum of values specifying the number and type of buttons to display, the icon style to use, and the identity of the default button.
The default value for *buttons* is 0.

title Optional
String expression that displays in the title bar of the dialog box.
The default value for title is the application name.

Settings

The *buttons* argument settings are:

Table 47-15. MsgBox buttons Settings

Constant	Value	Description
mbOKOnly	0	Display OK button only.
mbOKCancel	1	Display OK and Cancel* buttons.
mbAbortRetryIgnore	2	Display Abort, Retry, and Ignore buttons.
mbYesNoCancel	3	Display Yes, No, and Cancel* buttons.
mbYesNo	4	Display Yes and No buttons.

Table 47-15. MsgBox buttons Settings

mbRetryCancel	5	Display Retry and Cancel* buttons.
mbCritical	16	Display Critical Message icon.
mbQuestion	32	Display Warning Query icon.
mbExclamation	48	Display Warning Message icon.
mbInformation	64	Display Information Message icon.
mbDefaultButton1	0	First button is default.
mbDefaultButton2	256	Second button is default.
mbDefaultButton3	512	Third button is default.

- The first group of values (0-5) describes the number and type of buttons to display in the dialog box
- The second group of values (16, 32, 48, 64) describes the icon style
- The third group of values (0, 256, 512) shows which button is the default.

When adding numbers to create a final value for the buttons argument, use only one number from each group.

Return Values

Table 47-16. MsgBox Return Values

Constant	Value	Description
mbOK	1	OK
mbCancel	2	Cancel*
mbAbort	3	Abort
mbRetry	4	Retry
mbIgnore	5	Ignore
mbYes	6	Yes
mbNo	7	No

*If the dialog box displays a Cancel button, pressing Esc has the same effect as clicking Cancel.

Example

```
MsgBox("Hello",mbOK, "This is a message box")
```

Right

This function returns a specified number of characters from the right side of a string.

Syntax

```
Right(string, length)
```

Arguments

The Right function has these arguments:

string Required
A string expression from which the characters on the right side are returned

length A numeric expression indicating how many characters to return
If *length* is 0, Right returns a zero-length string (" "). If *length* is greater than or equal to the number of characters in the string, Right returns the entire string.

Example

The following sample code returns "dbc."

```
Right("abcdbc", 3)
```

Sin

This function returns a Double specifying the sine of an angle.

The Sin function takes an angle and returns the ratio of two sides of a right triangle. The ratio is the length of the side opposite the angle divided by the length of the hypotenuse.

The result lies in the range -1 to 1.

To convert degrees to radians, multiply degrees by pi/180. To convert radians to degrees, multiply radians by 180/pi.

Syntax

```
Sin(number)
```

Arguments

The Sin function has this argument:

number	Required Any expression that expresses an angle in radians. If the expression is not numeric, it is converted to a Numeric type.

Example

```
x = sin(y)
```

Spc

This function is used with the Print # statement or the Print method to position output.

Syntax

```
Spc(n)
```

Arguments

The Spc function has this argument:

n Required
The number of spaces to insert before displaying or printing the next expression in a list

If *n* is less than the output line width, the next print position immediately follows the number of spaces printed. If *n* is greater than the output line width, Spc calculates the next print position using the formula:

```
currentprintposition + (n Mod width)
```

For example, if the current print position is 24, the output line width is 80, and you specify Spc(90), the next print will start at position 34 (current print position + the remainder of 90/80). If the difference between the current print position and the output line width is less than n (or n Mod width), the Spc function skips to the beginning of the next line and generates spaces equal to n - (width - currentprintposition).

Example

```
Spc(3)
```

Str

This function returns a string representation of a number.

Syntax

```
Str(number)
```

Arguments

The Str function has the following argument:

number	Required Any expression. If the expression is not numeric, it is converted to a Numeric type.

When numbers are converted to strings, a leading space is always reserved for the sign of *number*. If *number* is positive, the returned string contains a leading space and the plus sign is implied.

Example

```
x = Str(324)
```

Tab

This function is used with the Print # statement or the Print method to position output.

Syntax

```
Tab[(n)]
```

Arguments

The Tab function has this argument:

n Optional
The column number to move to before displaying or printing the next expression in a list

If *n* is omitted, Tab moves the insertion point to the beginning of the next print zone. This allows you to use Tab instead of a comma whenever you use the comma as a decimal separator.

If the current print position on the current line is greater than *n*, Tab skips to the nth column on the next output line. If *n* is less than 1, Tab moves the print position to column 1. If *n* is greater than the output line width, Tab calculates the next print position using the formula:

```
n Mod width
```

For example, if *width* is 80 and you specify *Tab(90)*, the next print will start at column 10 (the remainder of 90/80). If *n* is less than the current print position, printing begins on the next line at the calculated print position. If the calculated print position is greater than the current print position, printing begins at the calculated print position on the same line.

The leftmost print position on an output line is always 1. When you use theb statement to print to files, the rightmost print position is the current width of the output file, which you can set using the Width # statement.

Example

```
Tab(2)
```

Val

This function returns the numbers contained in a string as a numeric value of the appropriate type.

Syntax

```
Val(string)
```

Argument

The Val function has this argument:

string	Required Any valid string expression

The Val function stops reading the string at the first character it can't recognize as part of a number. However, the function recognizes the radix prefixes &O (for octal) and &H (for hexadecimal). Blanks, tabs, and linefeed characters are stripped from the argument. The Val function recognizes only the period (.) as a valid decimal separator.

Example

```
i=Val("123")
```

Automation Support

The macro engine in the program supports Automation through its automation objects. After an object is created using CreateObject( ) or connected to using GetObject( ), the object's methods may be called using the usual syntax.

```
Object.Method arg1, ..., argn
var = Object.Method( arg1, ..., argn )
var = Object.Property
Object.Property = expression
```

Related Topics

CreateObject

GetObject

Dialog Box Controls

The macro language of this program uses the following dialog box controls:

- CheckBox
- CheckListBox

- ComboBox
- EditBox
- GridControl
- ListBox
- PushButton
- RadioBox
- SliderControl
- SpinButton
- TabControl
- TreeItem
- TreeView

CheckBox

This control represents a check box on a dialog box. You can reference a particular check box using the Control method of a dialog box. The CheckBox object uses the State and Property methods.

State

This method sets the state of the check box.

Syntax

```
checkbox.State(iState)
```

Arguments

State has this argument:

iState Required
A numeric expression, representing the check box state to set

iState may have one of the following values:

Table 47-17. CheckBox.State istate Values

Value	Description
0, or False	Unchecked
1, or True	Checked
2	Indeterminate

Value Property

This method returns or sets the state of the check box control.

Syntax

```
checkbox.Value[=iState]
```

Arguments

The Value Property has this argument:

iState Required
A numeric expression, representing the check box state to set

iState may have one of the following values::

Table 47-18. CheckBox.Value istate Values

Value	Description

Table 47-18. CheckBox.Value istate Values

0, or False	Unchecked
1, or True	Checked
2	Indeterminate

CheckListBox

This control represents a check list box on a dialog box.

State

This method sets the selection state of the check list box.

Syntax

```
CheckListBox.State(string)
```

Arguments

The State method has this argument:

string	Required Any valid string expression, containing the list of items to select

SetCheck

This method sets the check state of the check list box.

Syntax

```
CheckListBox.SetCheck(string)
```

Arguments

The SetCheck method has this argument:

string	Required A string expression, containing the list of the items to check. All the items that do not appear in the list are not checked.

ListCount Property

This method returns the number of items in the check list box.

Syntax

```
CheckListBox.ListCount
```

SelCount Property

This method returns the number of selected items in the check list box.

Syntax

```
CheckListBox.SelCount
```

Selected Property

This method returns or sets the selection status of an item in the check list box. This property is an array of Boolean values with the same number of items as the List property.

Syntax

```
CheckListBox.Selected(index)[=boolean]
```

Check Property

This method returns or sets the checked status of an item in a check list box. This property is an array of Boolean values with the same number of items as the list property.

Syntax

```
CheckListBox.Check(index)[=boolean]
```

Text Property

This method returns the text of the item currently selected in the check list box.

Syntax

```
CheckListBox.Text
```

ComboBox

This control represents a combo box on a dialog box.

Select

This method sets the selection state of the combo box.

Syntax

`ComboBox.Select(``string`**`)`**

Arguments

The Select method has this argument:

string Required
A string expression, containing a string to select in the combo box

Example

```
ActiveLayer.Select("Top")
```

Edit

This method sets the edit state of the combo box.

Syntax

`ComboBox.Edit(``string`**`)`**

The Edit method has this argument:

string Required
A string expression, containing a string to insert into the edit box of the combo box

Text Property

This method returns or sets the combo box text.

Syntax

`ComboBox.Text``[=string]`

Arguments

The Text method has this argument:

string Required
A string expression, containing the text to set into the combo box

List Property

This method returns or sets the items contained in the combo box list. This list is a string array in which each element is a list item.

Syntax

```
ComboBox.List(index)
```

SelStart Property

This method returns or sets the starting point of the selected text. If no text is selected, this method indicates the position of the insertion point.

Syntax

```
ComboBox.SelStart[=index]
```

SelLength Property

This method returns or sets the number of characters selected.

Syntax

```
ComboBox.SelLength[=number]
```

SelText Property

This method returns or sets the string containing the currently selected text. If no characters are selected, this method returns a zero length string ("").

Syntax

```
ComboBox.SelText[=string]
```

Arguments

The argument is a string expression, containing the text to set into the edit box.

EditBox

This control represents an edit box on a dialog box. You can use the Control method of a dialog box to reference a particular edit box.

State

This method sets the state of the edit box.

Syntax

```
EditBox.State(string)
```

Arguments

State has this argument:

string	Required A string expression of the text to display in the edit box

Text Property

This method returns or sets the edit box text.

Syntax

```
EditBox.Text[=string]
```

The string argument is a string expression, containing the text to set into the edit box.

SelStart Property

This method returns or sets the starting point of selected text. If no text is selected this method indicates the position of the insertion point.

Syntax

```
EditBox.SelStart[=index]
```

SelLength Property

This method returns or sets the number of characters selected.

Syntax

```
EditBox.SelLength[=number]
```

SelText Property

This method returns or sets the string containing the currently selected text. If no characters are selected this method returns a zero length string ("").

Syntax

```
EditBox.SelText[=string]
```

Arguments

The SelText method has this argument:

string	A string expression, containing the text to set into the edit box

GridControl

This control represents a grid control on a dialog box. You can use the control method to reference a particular grid control.

ListBox

This control represents a list box on a dialog box.

State

This method sets the selection state of the list box.

Syntax

```
ListBox.State(string)
```

Arguments

The State method has this argument:

string Required
A string expression, containing the item numbers to select. All the items that do not appear in the list are not selected.

List Property

This method returns the items contained in a dialog box list portion. The list is a string array in which each element is a list item.

Syntax

```
ListBox.List
```

ListCount Property

This method returns the number of items in the list box.

Syntax

```
ListBox.ListCount
```

SelCount Property

This method returns the number of selected items in a list box.

Syntax

```
ListBox.SelCount
```

Selected Property

This method returns or sets the selection status of an item in a list box. This property is an array of Boolean values with the same number of items as the list property.

Syntax

```
ListBox.Selected(index) [=boolean]
```

Text Property

This method returns the currently selected (focused) item's text in a list box.

Syntax

```
ListBox.Text
```

PushButton

This control represents a push button (also called a command button) on a dialog box.

Click

This method emulates pressing a push button.

Syntax

```
button.Click()
```

Example

```
dlg.control("OK").click()
```

RadioBox

This control represents an option button on a dialog box.

State

This method checks the state of the option button.

Syntax

```
RadioBox.State(iState)
```

Arguments

The State method has this argument:

iState Required
A numeric expression, representing the position of an option button to check. -1 means that no button is selected.

Value Property

This method returns or checks the state of the option button.

Syntax

```
RadioBox.Value[=iState]
```

Arguments

Value Property has this argument:

iState Required
A numeric expression, representing the position of an option button to check. -1 means that no button is selected.

SliderControl

This control represents a slider control on a dialog box.

State

This method sets the state of the slider.

Syntax

```
SliderControl.State(iState)
```

Arguments

State has this argument:

iState Required
A numeric expression

Value Property

This method returns or sets the current slider position.

Syntax

```
Slider.Value[=val]
```

Arguments

Value Property has this argument:

val Required
A numeric expression

SpinButton

This control represents a spin button on a dialog box.

State

This method sets the state of the spin button.

Syntax

```
SpinButton.State(iState)
```

Arguments

State has this argument:

iState	Required A numeric expression

TabControl

This control represents a tab on a dialog box.

State

This method sets the selection state of the tab.

Syntax

```
TabControl.State(iState)
```

Arguments

State has this argument:

iState Required
A numeric expression, which represents the position of a tab.

Example

```
dlg.Control("Tab").State(3)
```

Value Property

This method returns or sets the current tab position.

Syntax

```
TabControl.Value[=tab]
```

Arguments

Value Property has this argument:

tab Required
Either a numeric expression representing the tab position or a string expression representing the tab caption

TreeItem

This control represents a tree item on a dialog box.

Select

This method sets the selection state of the tree item.

Syntax

```
TreeItem.Select(flag)
```

Arguments

The Select method has this argument:

flag Required
A numeric expression

flag may have one of the following values:

Table 47-19. TreeItem.Select flag Values

Value	Description
0, or False	Unselect
1, or True	Select

Example

```
item.Select(true)
```

Expand

This method sets the expand state of the tree item.

Syntax

```
TreeItem.Expand(flag)
```

Arguments

Expand has this argument:

flag Required
A numeric expression

flag may have one of the following values:

Table 47-20. TreeItem.Expand flag Values

Value	Description

Table 47-20. TreeItem.Expand flag Values

0, or False	Collapse
1, or True	Expand

Example

```
item.Expand(true)
```

Focus

This method sets the tree item focus to the item.

Syntax

```
TreeItem.Focus()
```

Example

```
item.Focus(1)
```

TreeView

This control represents a Tree View on a dialog box.

Item

This method returns a TreeItem object.

Syntax

```
TreeView.Item(itemname)
```

Arguments

Item has this argument:

itemname	Required A string expression representing the name of the item

Example

```
item = tree.Item("Net Objects\Nets\end")
```

BeginDrag

This method emulates dragging selected items off the tree.

Syntax

```
TreeView.BeginDrag(itemname)
```

Arguments

BeginDrag has this argument:

itemname	Required A string expression representing the name of the item to drag

Copy

This method copies the selected item to the Clipboard.

Syntax

```
TreeView.Copy(itemname)
```

Arguments

Copy has this argument:

itemname	Required A string expression representing the name of the item to copy

Drop

This method emulates dropping the dragged items onto an item.

Syntax

```
TreeView.Drop(itemname)
```

Arguments

The Drop method has this argument:

itemname	Required A string expression representing the name of the item onto which to drop the dragged items

Example

```
tree.Drop("Net Objects\Net classes")
```

Paste

This method pastes the contents of the Clipboard into the selected branch.

Syntax

```
TreeView.Paste(itemname)
```

Arguments

Paste has this argument:

itemname	Required A string expression representing the name of the item to paste

CreateNewItem

This method creates a new item in the selected branch. This method returns a TreeItem object that corresponds to the created item.

Syntax

```
TreeView.CreateNewItem(itemname)
```

Arguments

CreateNewItem has this argument:

itemname	Required A string expression representing the name of the item to create

Internal Macro Objects

The Internal Macro Objects of this program include the following:

- Application Object
- Dialog Objects
- Document Object
- HelpContents Object
- HelpContentsItem Object
- HelpPane Object
- Main View Object

Application Object

This object represents applications of the program. The object has the following methods:

- CreateNewDocument
- ExecuteCommand
- Help
- HelpContents
- HelpPane
- OpenCustomizeDialog
- OpenDocument
- OpenOptionsDialog
- OpenPropertiesDialog
- Quit
- RunMacro

Related Topics

Internal Macro Objects

CreateNewDocument

This method creates an empty document.

Syntax

```
Application.CreateNewDocument
```

ExecuteCommand

This method executes one of the commands of the program.

Syntax

```
Application.ExecuteCommand(command, [arg1,...])
```

Argument

The ExecuteCommand method has these arguments:

command	Required A string expression representing a PADS product command
arg1,...	Optional Represents optional arguments that are passed to *command*

Example1

```
Application.ExecuteCommand("ID_VIEW_BOARD")
```

Example2

```
Application.ExecuteCommand("Open", "C:\PADS Projects\preview.pcb")
```

Help

This method invokes the *Help*.

Syntax

```
Application.Help()
```

HelpContents

This method returns the *Help* contents in the Help Contents window.

Syntax

```
Application.HelpContents
```

Example

```
Set var = Application.HelpContents
```

HelpPane

This method returns the Help window.

Syntax

```
Application.HelpPane
```

Example

```
Set var = Application.HelpPane
```

OpenCustomizeDialog

This method opens the Customize modal dialog box.

Syntax

```
Application.OpenCustomizeDialog()
```

OpenDocument

This method opens an existing document identified by the path argument.

Syntax

```
Application.OpenDocument(path)
```

Argument

The OpenDocument method has this argument:

path Required
A string expression containing the path of the document to open

Example

```
Application.OpenDocument("C:\PADS Projects\preview.pcb")
```

OpenOptionsDialog

This method opens the Options modeless dialog box.

Syntax

```
Application.OpenOptionsDialog()
```

OpenPropertiesDialog

This method opens the Properties modeless dialog box.

Syntax

```
Application.OpenPropertiesDialog()
```

Quit

This method exits the application.

Syntax

```
Application.Quit()
```

RunMacro

This method executes one of the program commands.

Syntax

```
Application.RunMacro(path[, function [, arg1, ...]])
```

Arguments

The RunMacro method has these arguments:

path Required
A string expression containing the file path of the macro to run

function Optional
Name of a function or a sub in the macro file to be called.
If the *function* is specified, RunMacro returns what the function returns.
If the *function* is not specified, or it is a sub, which returns nothing, RunMacro returns nothing.

arg1,... Optional
Arguments that are passed onto *function*

Example

```
Application.RunMacro("C:\PADS Projects\mymacro.mcr")

Var = Application.RunMacro("C:\PADS Projects\mymacro.mcr", myfunction", 1,
2, 3)
```

Dialog Objects

A dialog object represents a dialog box. This object has the following methods:

- Control
- Focus
- HelpPane
- OpenHelpPane
- CloseHelpPane
- ShowHelpFor

Related Topics

Internal Macro Objects

Focus

This method sets focus to a dialog box control.

Syntax

```
Dialog.Focus(controlname)
```

Arguments

The Focus method has this argument:

controlname	Required A string expression representing the name of the control

Related Topics

Control

Control

This method returns a dialog box control.

Syntax

```
Dialog.Control(controlname)
```

Arguments

Control has this argument:

controlname	Required A string expression representing the name of the control

Example

This example returns the OK button.

```
set obj = dlg.Control("OK")
```

Related Topics

Focus

CloseHelpPane

This method closes the open help pane in a dialog box.

Syntax

```
Dialog.CloseHelpPane
```

Example

```
Dialog.CloseHelpPane
```

OpenHelpPane

This method displays the help pane for the dialog box.

Syntax

`Dialog.OpenHelpPane`

Example

```
Dialog.OpenHelpPane
```

ShowHelpFor

This method displays help for the specified control.

Syntax

```
Dialog.ShowHelpFor(controlname)
```

Arguments

The ShowHelpFor method has this argument:

controlname	Required The name of the control

Example

This example displays Help for the Apply button.

```
Dialog.ShowHelpFor("Apply")
```

Document Object

The Document object represents any currently loaded design. This object has the following methods:

- Print
- PrintSetup
- RepeatLastAction
- Save
- SaveAs

Related Topics

Internal Macro Objects

Print

This method prints the document.

Syntax

```
Document.Print()
```

PrintSetup

This method opens the Print Setup dialog box.

Syntax

```
Document.PrintSetup()
```

RepeatLastAction

This method repeats the last action performed during the current session.

Syntax

```
Document.RepeatLastAction()
```

Save

This method saves the document, if it has been modified.

Syntax

```
Document.Save()
```

SaveAs

This method saves the document to a user-defined name or path location.

Syntax

```
Document.SaveAs(path)
```

The SaveAs method has this argument:

path Required
A string expression representing the path to which to save the document

HelpContents Object

The HelpContents object represents the *Help* Contents window.

Item

The Item property finds the location of the Help contents item in the contents tree.

Syntax

```
Application.HelpContents.Item (path)
```

Example

```
Set item = Application.HelpContents.Item("File Operations\To Restore
Files")
```

Related Topics

Internal Macro Objects

HelpContentsItem Object

This object finds the name of the Help Contents item. The HelpContentsItem object has the following properties and one method (Select):

- Location
- Name
- Select
- SubItem
- SubItemCount

Related Topics

Internal Macro Objects

Location

This property returns the location of the item.

Syntax

```
Item.Location
```

Example

In this example, the item_loc variable is assigned a value of "its:C:\Program Files\Mentor Graphics\PADS\<*latest_release*>
\Documentation\Router\BlazeRouter.chm::
/fileops/To_Restore_Files.htm".

```
Set item = Application.HelpContents.Item("File Operations\To Restore
Files")
item_loc = item.Location
```

Name

This property returns the name of the item.

Syntax

```
Item.Name
```

Example

In this example, the item_name variable is assigned the value of "To Restore Files."

```
Set item = Application.HelpContents.Item("File Operations\To Restore
Files")
item_name = item.Name
```

Select

This method selects the item.

Syntax

```
Item.Select
```

Example

```
Set item = Application.HelpContents.Item("File Operations").SubItem(3)
item.Select
```

SubItem

This property returns a sub item of the item by its position. The required integer pos argument is the zero-based serial number of a sub item in the tree branch that the item represents.

Syntax

```
Item.SubItem(pos)
```

Example

In this example, the item_name variable is assigned value of "To Restore Files".

```
Set item = Application.HelpContents.Item("File Operations").SubItem(3)
item_name = item.Name
```

SubItemCount

This property returns the number of sub items within the item.

Syntax

```
Item.SubItemCount
```

Example

In this example, the *count* variable is assigned a value of 10.

```
Set item = Application.HelpContents.Item("File Operations")
count = item.SubItemCount
```

HelpPane Object

This object represents the Help window.

Document

The Document property displays the HTML document in the Help window. This property uses the Document Object Model (DOM). For the full description of the HTMLDocument interface, see the Microsoft Software Developer's Network (MSDN) documentation.

The following are the most useful properties that the Macro engine uses:

title	Sets or retrieves the title of the document. This property displays the document title in the title bar of the document window. Also, it identifies the contents of the document.
URL	Sets or retrieves the Uniform Resource Locator (URL) for the current document.

Related Topics

Internal Macro Objects

Main View Object

The MainView object represents the main view of the program. This object uses the following methods:

- ActiveLayer
- ToggleFullScreen
- MouseDown
- MouseEndDrag

- MouseMove
- MouseStartDrag
- MouseUp
- Print
- PrintPreview

Related Topics

Internal Macro Objects

ActiveLayer

This method displays the Active Layer combo box.

Syntax

```
MainView.ActiveLayer
```

Example

```
set layerCombo = MainView.ActiveLayer
```

ToggleFullScreen

This method turns the Full Screen mode on.

Syntax

```
MainView.ToggleFullScreen()
```

MouseDown

This method emulates pressing a mouse button.

Syntax

```
MainView.MouseDown(x, y, button)
```

Arguments

The MouseDown method has these arguments:

x Required
Any numeric expression
X-coordinate of pointer

y Required
Any numeric expression
Y-coordinate of pointer

button Required
String expression
Pressed button, if any

The *button* argument may contain one or more of the following values and modifiers:

Table 47-21. MainView.MouseDown button Values

Value	Description
L	Left mouse button pressed
M	Middle mouse button pressed
R	Right mouse button pressed
C	Ctrl button pressed (modifier)
S	Shift button pressed (modifier)
A	Alt button pressed (modifier)

MouseEndDrag

This method emulates ending a mouse drag operation.

Syntax

```
MainView.MouseEndDrag(x, y, button)
```

Arguments

The Mouse End Drag method has these arguments:

x
Required
Any numeric expression
X-coordinate of pointer

y
Required
Any numeric expression
Y-coordinate of pointer

button
Required
String expression
Pressed button, if any

The *button* argument may contain one or more of the following values and modifiers:

Table 47-22. MainView.MouseEndDrag button Values

Value	Description
L	Left mouse button pressed
M	Middle mouse button pressed
R	Right mouse button pressed
C	Ctrl button pressed (modifier)
S	Shift button pressed (modifier)
A	Alt button pressed (modifier)

MouseMove

This method emulates moving the mouse.

Syntax

```
MainView.MouseMove(x, y, button)
```

Arguments

The Mouse Move method has these arguments:

x	Required Any numeric expression X-coordinate of pointer
y	Required Any numeric expression Y-coordinate of pointer
button	Required String expression * indicates relative mode + indicates origin plus relative mode Pressed button, if any

The *button* argument may contain one or more of the following values and modifiers:

Table 47-23. MainView.MouseMove button Values

Value	Description
L	Left mouse button pressed
M	Middle mouse button pressed
R	Right mouse button pressed
C	Ctrl button pressed (modifier)
S	Shift button pressed (modifier)
A	Alt button pressed (modifier)

Examples

```
MainView.MouseMove(300,350,"+L")
or
MainView.MouseMove(300,350,"*L")
```

MouseStartDrag

This method emulates starting a mouse drag operation.

Syntax

```
MainView.MouseStartDrag(x, y, button)
```

Arguments

The MouseStartDrag method has these arguments:

x Required
Any numeric expression
X-coordinate of pointer

y Required
Any numeric expression
Y-coordinate of pointer

button Required
String expression
Pressed button, if any

The *button* argument may contain one or more of the following values and modifiers:

Table 47-24. MainView.MouseStartDrag button Values

Value	Description
L	Left mouse button pressed
M	Middle mouse button pressed
R	Right mouse button pressed
C	Ctrl button pressed (modifier)
S	Shift button pressed (modifier)
A	Alt button pressed (modifier)

MouseUp

This method emulates releasing a mouse button.

Syntax

```
MainView.MouseUp(x, y, button)
```

The MouseUp method has these arguments:

x Required
Any numeric expression
X-coordinate of pointer

y Required
Any numeric expression
Y-coordinate of pointer

button Required
String expression
Pressed button, if any

The *button* argument may contain one or more of the following values and modifiers:

Table 47-25. MainView.MouseUp button Values

Value	Description
L	Left mouse button pressed
M	Middle mouse button pressed
R	Right mouse button pressed
C	Ctrl button pressed (modifier)
S	Shift button pressed (modifier)
A	Alt button pressed (modifier)

Print

This method prints the current view.

Syntax

```
MainView.Print()
```

PrintPreview

This method turns the Print Preview mode on.

Syntax

```
MainView.PrintPreview()
```

Chapter 48
The Format for Rules in the ECO File

This chapter covers the syntax used in the ECO file when design rules are created or modified during ECO Operations. You can pass these rule entries from schematic capture systems that have rules capabilities or to schematic capture systems that have rules capabilities as part of the back annotation process. You can also Import the rules into other PCB designs.

Tip: When importing the ECO file into an existing design, rules entries are only applied to nets with the same name. Any nets that aren't found create an error report. Before importing, open the design and rename the target connections.

Heading information for the Rules section of the ECO file are illustrated in the Example ECO file.

The Formatting for Design Rules in the ECO File

Apply design rules at five hierarchical levels. The lowest level, called the default level, applies rules to all objects on the PCB not otherwise specified.

Table 48-1 lists the design rule hierarchical levels.

Table 48-1. Design Rules Hierarchical Levels

Hierarchical Level	Design Rule Object
Default	HIERARCHY_OBJECT PCB:PCB

Table 48-1. Design Rules Hierarchical Levels (cont.)

Hierarchical Level	Design Rule Object
Net	HIERARCHY_OBJECT NET:<net_name>
Class	HIERARCHY_OBJECT CLS:<class_name>
Pin Pair	HIERARCHY_OBJECT CON:<pin_pair_name>
Group	HIERARCHY_OBJECT GRP:<group_name>

General Design Rules

Design rules often apply to a list of objects. For each object, the ECO file contains the notation:

```
HIERARCHY_OBJECT <type>:<name>
```

For example, if a rule applies to nets DATA0 through DATA3, the ECO file lists the nets this way:

```
HIERARCHY_OBJECT NET:DATA0
HIERARCHY_OBJECT NET:DATA1
HIERARCHY_OBJECT NET:DATA2
HIERARCHY_OBJECT NET:DATA3
```

Conditional Design Rules

Conditional rules apply from one object against another object. The ECO file uses the format:

```
<from_type>:<from_name> <against_type>:<against_name> [<layer_name>]
```

to refer to the from and against objects and the layer name for Conditional Rules. The brackets around the layer name indicate it is an optional argument.

For example:

```
NET:GND CLS:Clock_Nets
```

In the above example, the rule applies throughout all layers of the design. When the rule applies only to a single layer named TOP, the ECO file lists the objects this way:

```
NET:GND CLS:Clock_Nets TOP
```

Conditional clearance and conditional high-speed rules are supported. Conditional routing rules do not exist.

The Formatting for Conditional Clearance Rules in the ECO File

When you can create or modify a conditional clearance rule, the ECO file lists the change using the following format:

```
*CREATE_CONDITIONAL_RULES* CLEARANCE
(or MODIFY_CONDITIONAL_RULES* CLEARANCE)
<from_type>:<from_name> <against_type>:<against_name> [<layer_name>]
TRACK_TO_TRACK <value>
VIA_TO_TRACK <value>
VIA_TO_VIA <value>
PAD_TO_TRACK <value>
PAD_TO_VIA <value>
PAD_TO_PAD <value>
SMD_TO_TRACK <value>
SMD_TO_VIA <value>
SMD_TO_PAD <value>
SMD_TO_SMD <value>
COPPER_TO_TRACK <value>
COPPER_TO_VIA <value>
COPPER_TO_PAD <value>
COPPER_TO_SMD <value>
TEXT_TO_TRACK <value>
TEXT_TO_VIA <value>
TEXT_TO_PAD <value>
TEXT_TO_SMD <value>
```

When you delete a conditional clearance rule, the ECO file lists the change using the following format.

```
*DELETE_CONDITIONAL_RULES* CLEARANCE
<from_type>:<from_name> <against_type>:<against_name> [<layer_name>]
```

The Formatting for Conditional High-Speed Rules in the ECO File

When you create or modify a conditional high-speed rule, the ECO file lists the change using the following format:

```
*CREATE_CONDITIONAL_RULES* HIGH_SPEED
(or MODIFY_CONDITIONAL_RULES* HIGH_SPEED)
<from_type>:<from_name> <against_type>:<against_name> [<layer_name>]
PARALLEL_LENGTH <value>
PARALLEL_GAP <value>
TANDEM_LENGTH <value>
TANDEM_GAP <value>
```

When you delete a conditional high speed rule, the ECO file lists the change using the following format:

```
*DELETE_CONDITIONAL_RULES* HIGH_SPEED
from_type>:<from_name> <against_type>:<against_name> [<layer_name>]
```

The Formatting for Differential Pairs Rules in the ECO File

When you create a Differential Pair or modify an existing one, the ECO file lists the change using the following format:

```
*CREATE_DIFFERENTIAL_PAIR* <object1> <object2>
```

```
GAP <value>
MIN_LENGTH <value>
MAX_LENGTH <value>
*MODIFY_DIFFERENTIAL_PAIR* <object1> <object2>
GAP <value>
MIN_LENGTH <value>
MAX_LENGTH <value>
*DELETE_DIFFERENTIAL_PAIR* <object1> <object2>
```

The Formatting for General Clearance Rules in the ECO File

When you create or modify a clearance rule, the ECO file lists the change using this format:

```
*CREATE_GENERAL_RULES*CLEARANCE
(or *MODIFY_GENERAL_RULES* CLEARANCE)
HIERARCHY_OBJECT <type>:<name>
MIN_TRACK_WIDTH <value>
REC_TRACK_WIDTH <value>
MAX_TRACK_WIDTH <value>
TRACK_TO_TRACK <value>
VIA_TO_TRACK <value>
VIA_TO_VIA <value>
PAD_TO_TRACK <value>
PAD_TO_VIA <value>
PAD_TO_PAD <value>
SMD_TO_TRACK <value>
SMD_TO_VIA <value>
SMD_TO_PAD <value>
SMD_TO_SMD <value>
COPPER_TO_TRACK <value>
COPPER_TO_VIA <value>
COPPER_TO_PAD <value>
COPPER_TO_SMD <value>
TEXT_TO_TRACK <value>
TEXT_TO_VIA <value>
TEXT_TO_PAD <value>
TEXT_TO_SMD <value>
OUTLINE_TO_TRACK <value>
OUTLINE_TO_VIA <value>
OUTLINE_TO_PAD <value>
OUTLINE_TO_SMD <value>
DRILL_TO_TRACK <value>
DRILL_TO_VIA <value>
DRILL_TO_PAD <value>
DRILL_TO_SMD <value>
SAME_NET_SMD_TO_VIA <value>
SAME_NET_SMD_TO_CRN <value>
SAME_NET_VIA_TO_VIA <value>
SAME_NET_PAD_TO_CRN <value>
DRILL_TO_DRILL <value>
BODY_TO_BODY <value>
```

When you delete a general clearance rule, the ECO file lists the change using the following format:

```
*DELETE_GENERAL_RULES* CLEARANCE
```

```
HIERARCHY_OBJECT <type>:<name>
```

The Formatting for General High-Speed Rules in the ECO File

When you create or modify a high-speed rule, the ECO file lists the change using the following format:

```
*CREATE_GENERAL_RULES* HIGH_SPEED
(or *MODIFY_GENERAL_RULES* HIGH_SPEED)
HIERARCHY_OBJECT <type>:<name>
MIN_LENGTH <value>
MAX_LENGTH <value>
STUB_LENGTH <value>
PARALLEL_LENGTH <value>
PARALLEL_GAP <value>
TANDEM_LENGTH <value>
TANDEM_GAP <value>
MIN_DELAY <value>
MAX_DELAY <value>
MIN_CAPACITANCE <value>
MAX_CAPACITANCE <value>
MIN_IMPEDANCE <value>
MAX_IMPEDANCE <value>
AGGRESSOR < ON | OFF>
SHIELD < ON | OFF > <gap_value> <net_name>
MATCH_LENGTH < ON | OFF > <tolerance_value>
```

When you delete a general high-speed rule, the ECO file list the change using the following format:

```
*DELETE_GENERAL_RULES* HIGH_SPEED
HIERARCHY_OBJECT <type>:<name>
```

The Formatting for General Routing Rules in the ECO File

When you create or modify a routing rule, the ECO file lists the change using the following format:

```
*CREATE_GENERAL_RULES* ROUTING
(or *MODIFY_GENERAL_RULES* ROUTING)
HIERARCHY_OBJECT <type>:<name>
LENGTH_MINIMIZATION_TYPE <NONE | TOTAL |HORIZONTAL|VERTICAL
SERIAL | PARALLEL | MID_DRIVEN >
DISPLAY_RATSNEST < ON | OFF >
COPPER_SHARE < ON | OFF >
AUTO_ROUTE < ON | OFF >
ALLOW_RIPUP < ON | OFF >
ALLOW_SHOVE < ON | OFF >
ROUTE_PRIORITY < ON | OFF >
VALID_LAYERS <layerName>...
VALID_VIA_TYPES <via_name>...
```

To delete general routing rules in the ECO file, use the following format. Do not supply parameters when deleting General Routing Rules.

```
*DELETE_GENERAL_RULES* ROUTING
HIERARCHY_OBJECT <type>:<name>
```

The Formatting for Net Classes Rules in the ECO File

When you create, rename, or delete net classes, the ECO file lists the change using the following formats:

```
*CLASS* <classname>
<net_name>. . .<net_name>
*RENCLASS* <old_class_name> <new_class_name>
*DELCLASS* <class_name>
```

When you add nets to an existing class, the ECO file lists the change using the same format as the one for creating new class:

```
*CLASS* <existing_classname>
<net_name_to_add>
. . .
<net_name_to_add>
```

When you remove nets from an existing class, the ECO file lists the change using the following format:

```
*EXCNET* <classname>
<net_name_to_exclude>
. . .
<net_name_to_exclude>
```

The Formatting for Pin Pair Group Rules in the ECO File

When you create, rename, or delete a pin pair group, the ECO file lists the change using the following format:

```
*GROUP* <group_name>
<pin_pair_name>. . .<pin_pair_name>
*RENGROUP* <old_group_name> <new_group_name>
*DELGROUP* <group_name>
```

When you add pin pairs to an existing group, the ECO file lists the change using same format as the one for creating new group:

```
*GROUP* <existing_group_name>
<pin-pair_to_add>
. . .
<pin-pair_to_add>
```

When you remove pin pairs from an existing group, the ECO file lists the change using the following format:

```
*EXCCON* <group_name>
<pin-pair_to_exclude>
```

```
. . .
<pin-pair_to_exclude>
```

Example ECO File

```
*PADS-ECO-MILS*
*PADS-ECO* ENGINEERING CHANGE ORDER REPORT
*REMARK* -- DEMOA.PCB -- Mon Jan 03 12:00:00 2000
*REMARK* Create a net class containing 3 nets
*CLASS* CLASS1$$$1879$$$1906$$$1928
*REMARK* Rename it to CLASS2
*RENCLASS*CLASS1 CLASS2
*REMARK* And delete it
*DELCLASS* CLASS2
*REMARK* Create a pin pair group containing 5 pin pairs
*GROUP* GROUP1U2.2 U1.2 U3.2 U2.2 U3.2 U4.2 U1.5 U2.5 U4.5 U1.5
*REMARK* Rename it to GROUP2
*RENGROUP*GROUP1 GROUP2
*REMARK* And delete it
*DELGROUP* GROUP1
*REMARK* Create default clearance rules for the board
*MODIFY_GENERAL_RULES* CLEARANCE
HIERARCHY_OBJECT PCB:PCB
MIN_TRACK_WIDTH 10
REC_TRACK_WIDTH 10
MAX_TRACK_WIDTH 12
TRACK_TO_TRACK 12
VIA_TO_TRACK 12
VIA_TO_VIA 12
PAD_TO_TRACK 12
PAD_TO_VIA 12
PAD_TO_PAD 12
SMD_TO_TRACK 12
SMD_TO_VIA 12
SMD_TO_PAD 12
SMD_TO_SMD 12
COPPER_TO_TRACK 12
COPPER_TO_VIA 12
COPPER_TO_PAD 12
COPPER_TO_SMD 12
TEXT_TO_TRACK 12
TEXT_TO_VIA 12
TEXT_TO_PAD 12
TEXT_TO_SMD 12
OUTLINE_TO_TRACK 12
OUTLINE_TO_VIA 12
OUTLINE_TO_PAD 12
OUTLINE_TO_SMD 12
DRILL_TO_TRACK 12
DRILL_TO_VIA 12
DRILL_TO_PAD 12
DRILL_TO_SMD 12
SAME_NET_SMD_TO_VIA 12
SAME_NET_SMD_TO_CRN 12
SAME_NET_VIA_TO_VIA 12
SAME_NET_PAD_TO_CRN 12
DRILL_TO_DRILL 12
BODY_TO_BODY 12
*REMARK* Create routing rules for net class CLASS1
*CREATE_GENERAL_RULES* ROUTINGHIERARCHY_OBJECT CLS:CLASS1
```

```
LENGTH_MINIMIZATION_TYPE VERTICAL
DISPLAY_RATSNEST OFF
COPPER_SHARE OFF
AUTO_ROUTE OFF
ALLOW_RIPUP OFF
ALLOW_SHOVE OFF
ROUTE_PRIORITY 3
VALID_LAYERS Top Bottom
VALID_VIA_TYPE STANDARDVIA
*REMARK* Create high speed rules for net $$$1963.Assign GND as a
*REMARK* shield net. (GND must first be assigned to a plane layer)
*CREATE_GENERAL_RULES* HIGH_SPEED
HIERARCHY_OBJECT NET:$$$1963
MIN_LENGTH 0
MAX_LENGTH 50000
STUB_LENGTH 1000
MIN_DELAY 0.000000
MAX_DELAY 10.000000
MIN_IMPEDANCE 50.000000
MAX_IMPEDANCE 150.000000
MIN_CAPACITANCE 0.000000
MAX_CAPACITANCE 10.000000
PARALLEL_LENGTH 1000
PARALLEL_GAP 200
TANDEM_LENGTH 1000
TANDEM_GAP 200
AGGRESSOR OFF
SHIELD OFF 200 GND
MATCH_LENGTH OFF 200
*REMARK* Create high speed rules for pin pair group GROUP1
*CREATE_GENERAL_RULES* HIGH_SPEEDHIERARCHY_OBJECT GRP:GROUP1
MIN_LENGTH 0
MAX_LENGTH 50000
STUB_LENGTH 1000
MIN_DELAY  0.000000
MAX_DELAY 10.000000
MIN_IMPEDANCE 50.000000
MAX_IMPEDANCE 150.000000
PARALLEL_LENGTH 1000
PARALLEL_GAP 200
TANDEM_LENGTH 1000
TANDEM_GAP 200
AGGRESSOR OFF
*REMARK* Create routing rules for pin pair CON:U4.6-U1.6
*CREATE_GENERAL_RULES* ROUTINGHIERARCHY_OBJECT CON:U4.6-U1.6
AUTO_ROUTE OFF

ALLOW_RIPUP OFF

ALLOW_SHOVE OFF
```

```
ROUTE_PRIORITY 3
*REMARK* Create conditional clearance rules for net $$$1928
*REMARK* against net $$$1951 on all layers.
*CREATE_CONDITIONAL_RULES* CLEARANCENET:$$$1928 NET:$$$1951
TRACK_TO_TRACK 12
VIA_TO_TRACK 12
VIA_TO_VIA 12
PAD_TO_TRACK 12
PAD_TO_VIA 12
PAD_TO_PAD 12
SMD_TO_TRACK 12
SMD_TO_VIA 12
SMD_TO_PAD 12
SMD_TO_SMD 12
COPPER_TO_TRACK 12
COPPER_TO_VIA 12
COPPER_TO_PAD 12
COPPER_TO_SMD 12
TEXT_TO_TRACK 12
TEXT_TO_VIA 12
TEXT_TO_PAD 12
TEXT_TO_SMD 12
*REMARK* Create conditional high speed rules for net $$$2008
*REMARK* against net $$$2016 on all layers.
*CREATE_CONDITIONAL_RULES* HIGH_SPEEDNET:$$$2008 NET:$$$2016
PARALLEL_LENGTH 1000
PARALLEL_GAP 200
TANDEM_LENGTH 1000
TANDEM_GAP 200
*REMARK* Create conditional clearance rules for pin pair net U4.6-U1.6
*REMARK* against pin pair U2.6-U1.6. Applies only on layer BOTTOM.
*CREATE_CONDITIONAL_RULES* CLEARANCECON:U4.6-U1.6 CON:U2.6-U1.6 BOTTOM
TRACK_TO_TRACK 12
VIA_TO_TRACK 12
VIA_TO_VIA 12
PAD_TO_TRACK 12
PAD_TO_VIA 12
PAD_TO_PAD 12
SMD_TO_TRACK 12
SMD_TO_VIA 12
SMD_TO_PAD 12
SMD_TO_SMD 12
COPPER_TO_TRACK 12
COPPER_TO_VIA 12
COPPER_TO_PAD 12
COPPER_TO_SMD 12
TEXT_TO_TRACK 12
TEXT_TO_VIA 12
TEXT_TO_PAD 12
TEXT_TO_SMD 12
*REMARK* Delete the conditional clearance rules for pin pair net U4.6-U1.6
*REMARK* against pin pair U2.6-U1.6 on layer BOTTOM.
*DELETE_CONDITIONAL_RULES* CLEARANCECON:U4.6-U1.6 CON:U2.6-U1.6 Bottom
*REMARK* Create a differential pair of nets $$$1906 and $$$2008
*CREATE_DIFFERENTIAL_PAIR* NET:$$$1906 NET:$$$2008
GAP 10
MIN_LENGTH 10
MAX_LENGTH 100
```

```
*REMARK* Create differential pair of pin pairs U2.8-U1.8 and U4.6-U1.6
*CREATE_DIFFERENTIAL_PAIR* CON:U2.8-U1.8 CON:U4.6-U1.6
GAP 10
MIN_LENGTH 10
MAX_LENGTH 100
*END*
```

Glossary

— A —

absolute coordinates

Coordinates of a location based upon their distance from the origin (coordinates 0,0) of the design area.

accelerator keys

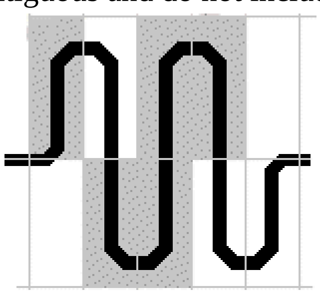
Key sequences used to invoke commands and change system settings without using the mouse. Accelerator keys are called shortcut keys in the PADS product documentation.

accessible nets

Nets for which you can define test points. DFT Audit analyzes all nets. If DFT Audit determines that test probes can access them, the nets are accessible (also called adaptable).

accordion

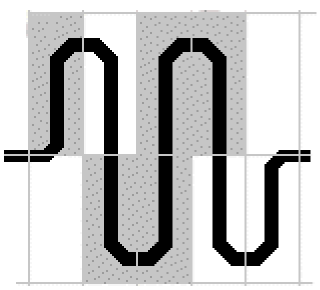
A trace pattern resembling a signal wave that adds length to traces. The trace patterns are contiguous and do not include layer changes.



accordion gap

The gap of an accordion sets the pitch between chords. The gap is a user-definable number multiplied by the same net trace-to-corner clearance.

Tip: If the same-net trace-to-corner distance equals zero, then Trace Width is used for the gap calculation.



See also: accordion, amplitude, pair routing gap

acid trap

An acid trap is a location where acid gets trapped in an area due to the surface tension of the etching. This acid causes over-etching, which hurts yield.

active component

The active substituted component in an assembly variant. Active means that this substitution of the component is used in the current variant.

See also: default.asc

active layer

The design layer to which new information is added. You select the active layer by choosing the layer in the Layer list on the standard toolbar. You can also do this by using the L modeless command.

ACTM#

The 16-digit number found on your security key.

adaptable nets

See accessible nets

adhesive

A substance used to attach the bodies of devices to a PC board.

aggressor nets

When using the Electrodynamic Checking program (EDC), a net or pin pair that is considered a source of interference.

align

To reposition placed parts to match the alignment of another part.

alignment tool

A small, temporary marker at each location where dimensioning occurs.

alpha pins

Pins with descriptive letters that are substituted for pin numbers. For example, GND for the ground pin. Alphanumeric pin assignments are made in the Library Manager's part type editor.

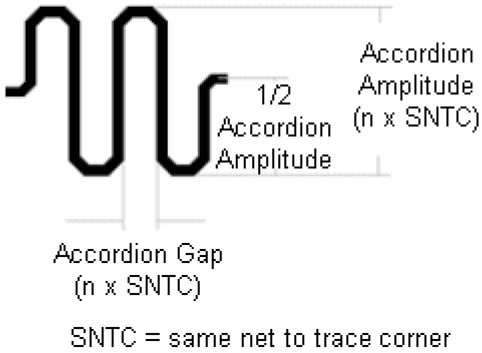
alphanumeric pins

Pins with alphanumeric pin numbers. An alphanumeric name consists of a prefix and suffix. The prefix or the suffix can contain either alpha letters or numeric numbers. For example, A1, 1A, or even DATA07 (consists of the prefix “DATA” and the suffix “07”).

amplitude

The amplitude of an accordion sets the accordion height (for horizontal accordions) or accordion width (for vertical accordions). The amplitude is a user-definable number, multiplied by the same net trace-to-corner clearance.

Tip: If the same-net trace-to-corner distance equals zero, then Trace Width is used for the amplitude calculation.



See also: accordion, gap (accordion)

analog circuit

A design composed of discrete components such as capacitors, resistors, and diodes.

angstrom

1/10,000 of a micrometer (10-4um).

annotation (forward and backward)

Forward annotation refers to the process of updating the design file to match the schematic file. Backward annotation refers to updating the schematic file to match the design file.

annular pad

A pad shape that enables you to specify an inside and an outside diameter. This creates a donut shape because the inner hole was used to center the drill bit when boards were hand-drilled on a drill press. Though obsolete, the annular pad is still offered for special circumstances.

antipad

For plane layers, a slightly oversized pad diameter that plots as a clearance for through-hole pins that should not connect to the plane.

any-angle coupling trace

Part of a route that connects SBP fanouts to serpentine routes.

aperture

A uniquely shaped window or hole that is attached to an aperture wheel on a photoplotting machine.

aperture table

A table that matches the line widths necessary to print your design with the plotter setup. PADS Layout can prepare the table automatically, or you can prepare it manually.

Artwork for printed circuit manufacturing is created by exposing clear film to light that is passed though the aperture. Although the aperture wheel has been made obsolete by laser plotters, an aperture table is still necessary to drive laser plotters.

apl.dcr

A setup file for Novell network security.

application-specific integrated circuit

An IC designed to meet a specific customer requirement.

area select

A method for selecting an object or a group of objects. If you enable area select by clicking Filter on the Edit menu, a selection rectangle is created and all items within the rectangle are selected.

array

A group of items, such as bonding pads, that are arranged in rows and columns.

artwork

Clear film with darkened areas representing pads and connecting traces, and used for manufacturing a printed circuit board. Each layer of a design has its own unique artwork, such as silkscreen and solder mask.

.asc

The file extension used to identify a proprietary PADS-format ASCII file.

ASCII format

A translation format that uses ASCII text to define the PCB design. ASCII format is widely used to list the parts and connections in a design, to import and export design items, and to check the design for binary corruption.

ASIC

An acronym for application-specific integrated circuit.

assembly drawing

A final design document that provides the part name, type, and orientation for each device on a printed circuit board. An assembly drawing is used for assembly of the final product.

assembly variant

A specific manufacturing configuration of a PCB. Assembly variants specify which components are used, which are not used, and which are substituted with a different decal part type. Several assembly variants can exist for a single PCB.

associated net

A series of nets connected by one or more components. Length, differential pair and matched length rules can be applied to an associated net as though it were a single net.

associating component

A component through which an associated net passes.

associating copper

Copper combined with the terminals in the PCB Decal Editor.

attribute groups

A group of structured attributes. For example, the DFT group includes the following attributes:

1. DFT.Nail Count Per Net
   DFT.Nail Number
   DFT.Nail Diameter

attributes

Attributes contain information you have associated with an object in your design. Attributes contain the types of part information that can be included in the parts library description and exported to a parts list. Examples are part manufacturer, package type, order number, and so on.

Auto Dimensioning tab

The tab on the Options dialog box that determines the appearance of newly created dimensions.

automation

A way for heterogeneous applications to communicate with each other. PADS products make some data, such as the database in use, and some functionality, such as opening files or selecting objects, available to other applications.

autorouter pass types

Pass types are part of an autorouting strategy that determines how the autorouter routes a design.

Table Glossary-1. Pass Types

Pass	Description
Center	Places traces equidistant from component pins or vias and each other to evenly distribute any available space in the channel.
Fanout	Places vias for inaccessible SMD component pins and routes from the vias to the pins.
Miters	Converts all route corners of a specified angle to diagonal corners.
Optimize	Analyzes each trace and tries to improve the quality of the route pattern by removing extra segments, reducing via usage, and shortening trace lengths.
Patterns	Searches for groups of unrouted connections that can be completed using typical C routing patterns, Z routing patterns, and memory patterns and then routes them.
Route	Sequentially routes each unroute until all connections are attempted.
Test Point	Analyzes the testability of the design, determines which nets require testing, adjusts the routes, and inserts test points to improve testability.
Tune	Adjusts the length of length-controlled traces. The Tune pass tunes all routed traces with length rules, and automatically adjusts length-controlled traces to meet design rules.

axial lead

A connection pin that protrudes straight out from the component body and bends at 90 degrees for insertion into the PC board. An axial lead is usually associated with discrete components such as resistors, capacitors, or diodes.

— B —

back-annotate

Update a schematic file to match its design file.

ball bonding

A bonding technique that provides increased contact between a gold wire and a chip bond pad. This method uses thermal compression to melt gold wire to form a ball.

ball grid array

A packaging method that uses a substrate to interconnect one or more die to an array of solder alloy spheres.

base option

The Base Option, in Assembly Variants, contains all of the common components in all of the existing variants; in other words, it contains a filtered database. If you uninstall or substitute components in a variant, they are removed from the Base Option. Therefore, the Base Option, because it contains only installed options, is also a subset of the raw database. You can use the Base Option to view all of the items in all of the variants, or the base of all variants.

The Base Option always exists; you cannot delete it.

base part

When making a union, the part type of the first selected part. Base parts can either be left in position and joined by secondary parts, or repositioned to imitate the first selected prototype part.

baseline dimensioning

A type of dimensioning in which a series of dimensions have a common start point, such as datum dimensioning.

basic units

A basic unit is the smallest unit of measurement in a PADS database. All values in the database are stored in binary format basic unit and are converted to the current user units (mils, mm, or inches) for screen display. If you need to reimport the information to .pcb format, export in basic units.

Conversions are:

1. 1 mil=38100 basic units
2. 1 millimeter = 1500000 basic units

BGA

An acronym for ball grid array.

BGA fanout

A single-segment fanout that connects BGA array pads to BGA vias. This single-segment fanout always ends in a via.

BGA/PGA decals

A full matrix decal for BGAs and PGAs, including staggered array patterns.

biased pin pair

A layer biased pin pair is any pin pair with a design rule specifying a layer bias to one or more, but not all, electrical routing layers.

blind via

A via that connects an outer layer to one or more inner layers, without passing through all other layers of a printed circuit board.

bmp

An image file that can be pasted into documents or other programs such as Microsoft Word. PADS products use the Copy Bitmap command to capture these as screen images.

board markings

Designers usually include identification information on a board. These may include the board part number, the assembly part number, the company name, the product name, the revision level, the serial number, the copyright notice, an anti-static symbol, warning messages, UL labels, test labels and many other types of information. This information may be in ink on the silkscreen layers, in copper on the top and/or bottom layer or some combination of the two. These are typically referred to as board markings.

Tips:

- Add text to an electrical layer and it will be created in copper. Add text to a Fabrication, Assembly, and Documentation Layer and it will be created during the silkscreen process.
- Use the Text command to add board markings to your design.

See also: Adding Free Text

board outline

The actual shape of the printed circuit board, defined by line segments and arcs. The board outline is entered on layer 0 and displayed on all layers.

bonding pads

Metallization areas placed around the perimeter of the integrated circuit die, to which aluminum or gold wires connect the die to the component package.

bounding rectangle

The smallest rectangle that encloses all nontext graphics on all layers.

breakpoint marker

A small brown dot in the Output window gutter that indicates a breakpoint in a script or macro.

bumped chip

A die or chip that has been specifically processed with buffer metals over the I/O pads, followed by an addition of solder or gold bumps to provide bonding areas for direct chip attachment onto a substrate.

buried via

A via that only connects two inner layers.

bus

A series of connections that share a common use, such as memory array or data array, and are usually routed parallel to each other.

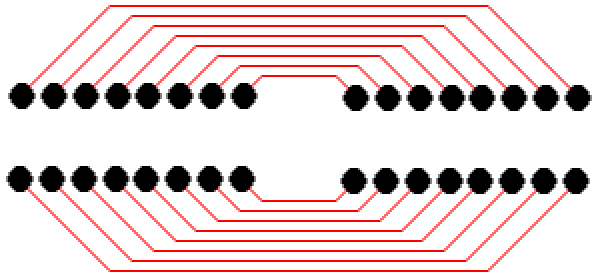
bus routing

Routing two or more pin pairs simultaneously and in close proximity to each other in neat, flowing patterns.

— C —

C routing pattern

A collection of routes that form a pattern resembling the letter C.



CAD

An acronym for Computer-Aided Design or Computer-Aided Drafting.

CAE

An acronym for Computer-Aided Engineering.

CAE Decal

The graphical representation of schematic symbols in PADS products.

CAM

An acronym for Computer-Aided Manufacturing.

CAM document

A combination of plot type and output device you create and save with the design. For example, you can include "Silkscreen Top, Photoplot" and "Silkscreen Top, Laser Printer" on your CAM Documents List and run them selectively when needed.

CAMDir
The powerpcb.ini file entry that enables you to specify the CAM master folder for creating CAM output.

capacitance
The ratio of charge within a trace that is a factor of the trace length and signal delay.

CBGA
An acronym for ceramic ball grid array.

CBP
An acronym for chip bond pad.

center pass
An autorouting pass that places traces equidistant from component pins or vias and each other to evenly distribute any available space in the channel.

CGA
An acronym for column grid array.

chamfered
A rectangle with the square corners cut off in order to create beveled edges on the corners.

chamfered path
A solid filled copper that, like a trace, acts as a conductor connecting pins and vias similar to a trace. But unlike a trace, which is created with a round aperture producing rounded outside corners, chamfered path copper allows for sharp specific outlines with a filled interior. When creating a chamfered path, you set options to create shapes with square or chamfered corners. The copper created by chamfered path has a Solid Copper property which overrides the Copper Hatch Grid and Drafting Line Width settings to make it a solid fill. Clearance rules for the chamfered path copper are also changed to match the clearance rules of a trace.

checking
Verifying the design meets previously defined rules, such as clearance and connectivity.

chip
An integrated circuit without packaging. A chip is also called a die.

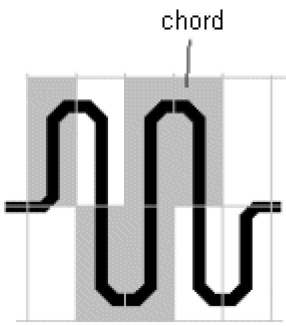
chip bond pad
Interconnect areas on the die on which wire bonds are connected to the substrate.

chip carrier
A square or rectangular IC package, with I/O connections on four sides.

chip on board
The packaging configuration in which a chip is bonded directly to a circuit board or substrate.

Chip Scale Package
A packaging configuration in which the dimension of the substrate is 1.2 times larger than the die.

chord
Half of an accordion.



See also: accordion,amplitude, gap (accordion)

clam shell fixing
A test fixture that tests both the top and bottom side of the PCB.

class
A collection of nets with a common set of design rules.

clearance
The measured space between routed objects such as trace-to-trace, trace-to-pad, or pad-to-pad.

closed cluster
Clusters that you cannot delete or replace during automatic cluster creation.

cluster
In Cluster Placement, a group of parts that must be placed close to each other.

CMOS
An acronym for Complementary Metal Oxide Semiconductor.

COB
An acronym for Chip On Board.

coefficient of thermal expansion
A quantity used to determine the length change of a material due to temperature change. Thermal expansion differences between the die and substrate must be considered for quality assurance.

collapse
To relocate the members of a cluster from their current placement to the center of the cluster.

column grid array
Similar to a ball grid array, but columns are used to improve the stresses of different thermal expansion between the board and the component.

Com port

Abbreviation for communications port. This port provides a connection between your computer and peripheral devices, such as plotters, modems, and other computers.

combine

Joining lines, or lines and text, together as one selectable object.

component side

The top or front side of a printed circuit board where devices are normally mounted.

composite fanout

A fanout from a pin that is common to two subnets. Often created by autorouting operations.

Composite fanouts provide access to component pins that may otherwise be inaccessible.

See also: fanout, subnets

composite rule trace

A trace that is attached to a pin (typically an SMD) shared by two subnets. This type of trace is typically created by autorouting operations.

See also: composite fanout, subnets

conditional rules

Rules placed on a signal that apply only if the signal is routed near another specified signal. Conditional rules are also known as against rules.

conductor

A material that causes heat or electrical current flow. For printed circuit design, a conductor is a piece of metal that connects pins of components together.

connected islands

A maximum set of subnet items already connected by a trace, copper unroute, or jumper.

See also: *subnets*, *subnet*

connections

Points of connectivity, such as a pin pair or a net.

connector

A unique component used to connect a portion of a printed circuit board with other devices.

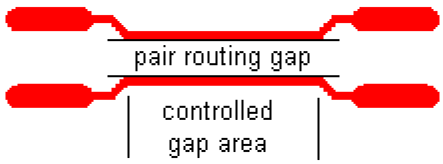
container application

An application that can incorporate embedded or linked items into its own documents. The documents managed by a container application must be able to store and display both OLE components, and data created by the application itself. A container application must also allow users to insert new items or edit existing items.

When you insert objects into a PADS product, the PADS product is the container application. When you insert a PADS file into another application, the other application is the container application.

controlled gap area

The part of the differential pair where the traces are drawn routed in parallel and separated by the pair routing gap. The controlled gap zone area starts at the gathering point and ends at the split point.



See also: gathering point, pair routing gap, split point

controlled gap length

For a differential pair, the ratio of the controlled gap area routing length to the overall routing length, in percentage.

See also: differential pairs

controlled length net

A net that has length rules, or contains pin pairs that have length rules.

The following high-speed rules are net length rules:

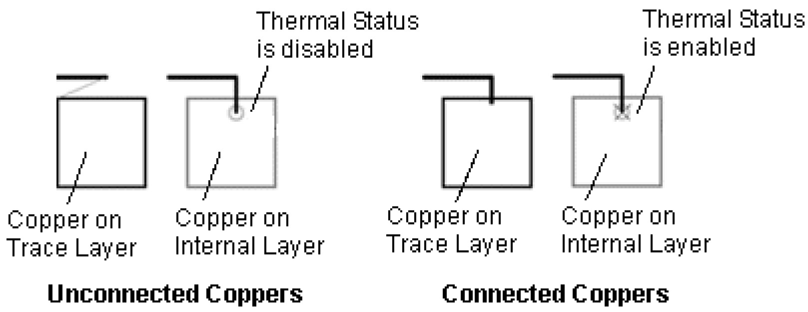
- Minimum/maximum length
- Matched length
- Differential pairs

converting database

The process that converts a non-native file, such as an .asc file or a .dxf file, to a PADS native format, or .pcb, file.

copper connectivity

Means unroutes are always connected to a copper at some point in the copper outline. A copper outline can include arcs. The following graphic illustrates how copper connects to a net.



See also: coppers, overlapping coppers

copper pour

The process which draws a copper area with insulation areas around traces and pins that pass through the copper, but are not attached or connected to the copper.

coppers

Polygons on an electrical layer representing an area of the PCB to fill with metal.

When a copper is assigned to a net, it is joined to the net with a trace or via. Coppers are obstacles to net objects unless the copper and the net belong to the same net.

See also: overlapping coppers, copper connectivity

copy route

The duplication of a trace or series of traces, using copy and paste.

corner

Point where a trace or line changes direction. The Selection Filter enables or disables picking geometric or route corners.

cost

Reduces usage of a layer. The higher the cost, the less a layer is used for routing.

cross-probing

Uses a link between PADS programs to reflect, in one PADS application, selections made in another PADS application.

cross-reference file

A file that maps design objects between two environments, such as PADS Layout and DxDesigner.

CSP

An acronym for chip scale package.

CTE

An acronym for coefficient of thermal expansion. It is also referred to as TCE.

cutouts

A closed polygon in a copper, copper pour, or board outline. In a copper or copper pour, a cutout results in a void area.

See also: overlapping cutouts

cycle picking

To sequentially select objects in the vicinity of the selection point using the Tab key.

— D —

dangling route

Dangling routes are stubs or spurs off of traces that are not tied to any pin by a ratsnest. See also partial route.

database units

The use of mils, metric, or inches within a design.

datum dimensioning

A style of dimensioning in which all dimensions are measured from a common starting point. The origin extension line is marked as zero, with each dimension reflecting the measurement from that point.

See also: Creating Baseline Dimensions

D-codes

Specific numbers assigned to photoplot machine apertures for program identification. D-CODES are included in the aperture table.

decal

The physical representation, or footprint, of a part.

decal copper

Open, closed, or associated copper produced within the physical representation of a component.

decal text

Documentation text produced within the physical representation of a component.

default component

The original component, before being replaced in the current assembly variant. The default component is always in the raw database, but not necessarily in the Base Option.

See also: active component

default layer mode

A layer mode in which a design can consist of up to 30 electrical layers, or a combination of electrical and nonelectrical layers. You change from default layer mode to increased layer mode by clicking the Max Layers button in the Layers Setup dialog box.

default.asc

The ASCII file accessed for new file creation. This file provides startup design information such as grid sizes, default colors, or other information.

default.cam

A file usually found in the C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs folder that contains default apertures, speed and feed settings, and drill symbols for CAM output. This file must exist in the same folder as specified by the UserDir variable in the powerpcb.ini file.

See also: increased layer mode

defaults

Conditions or options that are set when the PADS product starts.

delay

The time it takes for a signal to travel through a trace.

delete
To remove information from a design.

design area
The actual work area where a design is created.

design on the fly
To use ECO Operations to create a new design without first providing a netlist or parts list from schematic software. This can also be called design.

Design page
The Options page that controls design conditions, general routing conditions, and certain display and part movement method settings.

design rules
Established spacing and general routing constraints for electrical properties, or conductors, which are verified by clicking Verify Design from the Tools menu.

devicesn.dat
A file usually found in the C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs folder that contains CAM printer and plotter driver data. This file must exist in the same folder specified by the UserDir powerpcb.ini variable.

DFM
An acronym for Design for Manufacturing.

DFT Audit
DFT Audit analyzes every net for accessibility (adaptability) and creates a board report that identifies all inaccessible (non-adaptable) nets.

dice
The plural of die.

die
A single square or rectangular piece of semiconductor material into which a specific electrical circuit has been fabricated.

die bonding
To attach the semiconductor die to the package substrate with epoxy adhesives, gold eutectic, or solder alloy. It is also referred to as Die Attachment.

die flag
Metal shapes placed under a die for thermal management and/or electrical connection; also referred to as a flower pattern.

die side of CBP
The side of the die on which the CBP lies. Usually the die side of the CBP is the same as its fanout side, but in some cases more complex patterns of wire bond fanout may mean that the two sides are not the same.

Die Wizard

This feature creates die part definitions parametrically or imports the die description using GDSII or formatted ASCII files. The Die Wizard replaces Component IQ by providing die capture directly in the Advanced Packaging Toolkit layout editor. This eliminates the need to transfer .ciq files.

dielectric

A non-conductor of current; an insulator.

dielectric constant

A value given for manufacturing materials, such as FR-4, to describe electrical characteristics.

differential pairs

A group of two nets or two pin pairs routed side-by-side and separated by the pair routing gap for as much of the overall length as practical. A differential pair typically transmits two electrical signals that are driven 180 degrees out of phase from each other.

See also: pair routing gap

DIP

An acronym for Dual In-line Package.

DisableCaching

A powerpcb.ini file entry that, when set at 1, shuts off graphics optimization and, when set at 0, enables graphics optimization.

discrete device

A device that contains one circuit element. For example, a resistor or toggle switch.

disperse

A command that is active on several levels of Cluster Placement. When selected, it clears the board of all parts or clusters that are not glued down, and arranges them around the outside of the board outline according to decal type.

dispersion routes

Partial routes, ending in vias, which tie surface mount components to plane layers.

do file

The SPECCTRA router ASCII setup file that contains user-defined router commands to initiate batch routing.

dock

To take an isolated application dataset and pull the changes within the dataset into the main design project. Any conflicts with the merged data must be manually resolved.

documentation layers

Layers higher than the electrical layers in a PADS Layout database that contain text and lines to illustrate assembly, annotation, and provide instructions for manufacturing.

double-click
Two mouse clicks, in immediate succession, that usually initiate an edit action or complete the current action.

double-sided board
A printed circuit board made up of two routing layers, and which has no internal layers.

double-sided die
A die that has substrate bond pads on one side, and a BGA grid array on the other side. The two sides are connected through vias.

See also: single-sided die

drafting operations
Any operation that involves adding nonelectrical information, not associated with placement or routing, to a design.

Drafting tab
A Options tab that controls text settings, default line width, reference designator settings, and hatch setup.

drawn pads
Photoplot pads, usually finger pads, that are produced by opening the aperture and moving the board, with the aperture remaining open, to produce a pad shape.

DRC
An acronym for Design Rules Check.

drill chart
A diagram, produced on a drill drawing, that shows drill symbols matched with drill hole sizes. This is also referred to as a drill legend.

drill oversize
A factor applied to plated through holes for DRC purposes to account for drill oversizing during the PCB fabrication process.

drill pairs
Primarily for buried and blind vias, drill pairs define which layers are to be drilled and plated together during the fabrication process.

drill symbols
Unique symbols on a drill drawing plot that represent the various drill hole locations and sizes.

drill.dat
A user-definable ASCII file that determines settings for NC Drill output format options. This file must exist in the same folder specified by the UserDir variable in the powerpcb.ini.

DXF

An acronym for Data eXchange Format, a standard ASCII format for sharing graphics database files between different environments.

dxfset.dat

A file that contains the information for drill size and library name equivalents in basic units for the DXF Setup dialog box.

dynamic route

To create a route using the Dynamic Route tool, which automatically creates turns and pushes other routes aside in order to complete the connection.

— E —

ecad hint.map

A user-defined text file that you create, edit, and maintain. This file enables the replacement of approximated parts from PADS Layout, with geometrically accurate components previously modeled in Pro/ENGINEER. This file must exist in either the current working folder or in the Pro/ENGINEER software loadpoint\text folder.

ECO

An acronym for Engineering Change Order. This refers to a file with netlist changes that needs to be annotated to update either the schematic or layout that has become out of sync with the new design changes.

ECO mode

A mode that PADS Layout enters when the ECO toolbar is open. Changes that affect the connection list or parts list can be recorded in a file for backward annotation.

See also: ECO

ECO Options

The setup choices available for the ECO output file in the ECO Options dialog box.

ECO registration of attributes

Only ECO-registered attributes, set on the Objects tab of the Attribute Properties dialog box, can be added, deleted, or changed during the ECO process. Via attributes are not registered attributes and cannot be added, deleted, or changed during the ECO process.

You can modify ECO-registered attributes only in ECO mode.

Non-ECO-registered attributes are never recorded in an .eco file during ECO operations.

To compare ECO-registered attributes, use the Compare Only ECO Registered Attributes option on the Comparison tab in the Compare/ECO Tools dialog box.

EDA

An acronym for Electronic Design Automation.

EDC

An acronym for Electrodynamic Checking.

edge
One side of a polygon.

edge die
The two or three rows of dice along the outer circumference of a wafer.

edges
The Selection Filter preference that enables or disables selection of geometric segments.

editing
Any action that modifies a design.

electrical layers
Layers enabled for routing that are checked by DRC.

embedded objects
An object, including all of its data and the information needed to manage the object, that is contained within the framework of, and is a part of, the container application document.

See also: linked objects

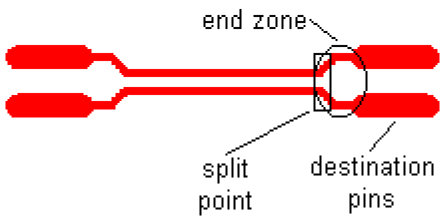
EnableMacroLanguage
The powerpcb.ini file entry that, when equal to one, enables loading of all macro parameters on startup and, when equal to zero, disables loading of macro parameters upon startup.

end component
A component having at least one pin which is a final pin of an associated net.

end no via
The mode initiated in the routing shortcut menus that, while routing, ends a partial route without a via.

end via
The mode initiated in the routing shortcut menus that, while routing, ends a partial route with a via.

end zone
The part of the differential pair between the split point and destination pins.



Tip: The labels in the above graphic correspond to routing that starts at the left-hand set of pins and ends at the right-hand set of pins. The label positions are reversed if the routing starts at the right-hand set of pins.

See also: differential pairs, split point

ending layer

The finishing layer for a drill pair or via definition. Enter information about ending layer in the Pad Stacks Properties dialog box.

engineering change order (ECO) operations

Any processes that modify the connection list or parts list.

entry angle

The angle at which a route enters a pad.

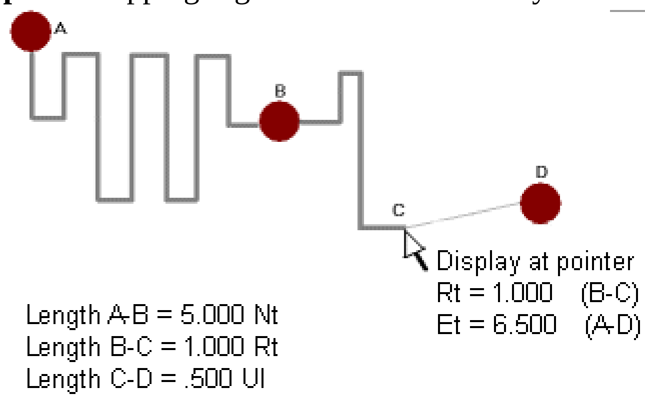
Esc

To use the Escape or Cancel keys to stop a current action.

estimated total length

The trace length monitor calculates estimated length as the combined total of routed length (Rt), plus the routed length for the entire net—including overlapping segments— (Nt), plus the unrouted length (U1) of the trace being routed.

Tip: Overlapping segments are counted only once.



See also: routed length, unrouted length

eutectic solder

A tin/lead alloy (63% tin, 37% lead) that melts at optimum temperatures.

export

The translation command used to convert a design file into PADS-format ASCII or DXF.

extended rules

Clearance, routing, and high-speed rules consisting of classes (one or more nets), groups (one or more pin pairs), individual pin pairs, decals, components and differential pairs. Without the Extended Rules option, you can assign rules on the net level only.

extension lines

Lines extending from the points being measured.

extents

The limits of the x and y coordinate area that is occupied by all items within a design. This includes information external to the board outline, such as dimensions or fabrication notes.

— F —

Fabless

A semiconductor company that subcontracts wafer manufacturing because it does not have its own wafer manufacturing facility.

fabrication

With semiconductor manufacturing, the front-end process of making devices in semiconductor wafers only, not the package assembly or back-end stages.

fanout

A segment of trace or copper shape added to SMD pads to facilitate routing. A fanout typically consists of one or more trace segments connecting a component pad to a via, allowing the signal on an outer layer to connect to one or more internal signal layers or planes. A specialized repeated pattern is often necessary to break out multiple pads on the same component far enough from the component to allow easy routing.

Use fanouts to:

- allow on-grid access by autorouters that cannot handle off-grid pads.
- make routing easier, and ensure connections are made.
- connect SMD pins to an inner plane layer using vias.
- connect an SMD pad to an inner signal layer where more routing space is available.

fanout pass

An autorouting pass that places vias for inaccessible SMD component pins and routes, from the vias to the pins.

fanout side of CBP

The side of the SBP Guide to which the CBP should be wire bonded. Usually the fanout side of the CBP is the same as its die side, but in some cases more complex patterns of wire bond fanout may mean that the two sides are not the same.

FCBGA

An acronym for flip chip ball grid array.

feature size

The smallest line width or spacing between lines or features on a semiconductor die.

feed-through hole

A drilled and plated hole that passes conductivity from one layer to another. This is also called a via.

fiducials

Fiducials are alignment marks, a type of target, used for calibration before placing objects.

There are at least three types of fiducials:

- **Panel fiducials**—used to align an entire panel of boards.
- **Board fiducials**—used to align components on a specific board (on or off a panel). Fiducials are (typically) round solid targets placed near three corners of each board on each side of the board that will receive components. The pick and place system scans the board for these targets (shiny circles approximately .040" in diameter) and uses them to align the machine before it starts placing parts.
- **Component fiducials**—used for close tolerance placement of high pin-count components with fine pitch leads. The footprint (PCB decal) of a fine pitch component will typically contain two component fiducials at opposite corners of the footprint. This allows the pick and place machine to align the fine pitch component exactly on the footprint.

field upgrade

Programming options on your security key by entering in a key unlock code using plicense.exe or equivalent.

file sharing

Multiple users accessing the same file or files through a network.

file.dir

The powerpcb.ini file entry that specifies the default location of your design files.

filter

A settings dialog box within that controls which types of objects can be selected.

find

The PADS command that locates, and optionally selects, an object or group of objects in the database.

finger pad

One of many long pads placed in a series to represent an edge connector.

flashed pads

Pads produced on a photoplotter by opening the aperture momentarily, without moving the board, to produce a pad shape.

flat pack

A component package where the leads extend away from the component and remain on a parallel plane with the base of the component.

flip

The command that moves the selected items to the opposite side of the board.

flip chip

An IC designed for face-down mounting by means of controlled-collapse solder pillars on a device's I/O bonding pads.

floating license

A method of licensing where a central security server manages a pool of licenses for use by a large number of clients.

floating toolbars

Toolbars you can undock from the sides of the application window and place anywhere on screen.

flood

To fill a previously defined copper pour area.

flower pattern

Metal shapes placed under a die for thermal management and/or electrical connection; also referred to as a die flag.

A ceramic, surface-mounted hermetic package.

FlushUndoBeyondSize

The powerpcb.ini file entry that determines the maximum size of the undo buffer before PADS Layout removes previous commands from the undo buffer to make room for the current command. If adding the current command causes the undo buffer to exceed this maximum size, PADS Layout removes previous commands until the undo buffer can store the current command.

follow route

The connections or pin pairs that are part of the bus routing, and which are routed following the guide route's path.

footprint

The arrangement of pads for a given part decal. For example, the footprint of a fourteen DIP is two rows of seven pads, spaced 100 mils in the Y direction, and 300 mils in the X direction.

forward-annotate

Update a design file using data from a schematic.

FR-4

An acronym for Fire Retardant Number Four, an epoxy-resin substrate material used in laminate applications.

free copper

Open or closed copper that is not associated to other copper or pads.

free disk space

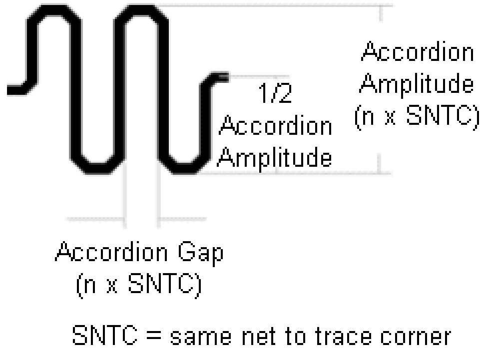
The physical amount of space available on your hard drive that is available for use by programs.

— G —

gap (accordion)

The gap of an accordion sets the pitch between chords. The gap is a user-definable number multiplied by the same net trace-to-corner clearance.

Tip: If the same-net trace-to-corner distance equals zero, then Trace Width is used for the gap calculation.



See also: accordion, amplitude, pair routing gap

gate

An element of an electronic circuit whereby one or more signals are input, with one output being dependent on the state of the input(s) and the type of logic used to interpret the input.

Pin swapping involves exchanging like inputs

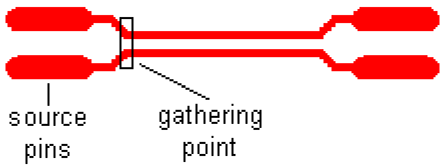
Gate swapping involves exchanging the entire element for a like element.

gate array

An IC consisting of a regular arrangement of gates that are interconnected to provide custom functions.

gathering point

The point near the source pins where differential pair traces can start to be routed together at the pair routing gap.



Tip: The labels in the above graphic correspond to routing that starts at the left-hand set of pins and ends at the right-hand set of pins. The label positions are reversed if the routing starts at the right-hand set of pins.

See also: differential pairs

GDI memory

Memory reserved for Windows devices and graphics.

Gerber
The language used to drive a photoplot machine. This language is an ASCII file with instructions for selecting an aperture, moving the light source, and turning the light source on and off.

Global tab
A Options tab that includes settings that affect an entire design, such as units of measurement and pointer size.

Glue
Anchors component(s) in their current location so they cannot be moved

grab bars
The two vertical or horizontal bars to the left or top of the window.

graphics cache
The PADS setting used to optimize graphics. This is handled by the DisableCaching entry in the powerpcb.ini and powerlogic.ini files.

green dot
The status indicator located in the upper left corner of the workspace. It is green when the system is idle or ready for operation. It is red when the workspace cannot receive user input, such as when producing CAM drawings.

grid
A division of the workspace into measurement steps to facilitate accurate spacing between placed parts and routed lines. Also refers to the display; small white dots locating the measurement steps

ground plane
A design layer completely filled with copper, except for clearances around nonconnected pads and vias.

group
A collection of pin pairs that share common design rules.

grow
An cluster placement feature that adds additional parts to an existing cluster.

guard band
A shape that attaches to the end of a trace during routing operations to denote an online design rule error.

gui
An acronym for Graphical User Interface. The GUI includes such things as menus and commands that allow for interaction between the user and the software program.

guide route
A route segment that is used for the first connection and that is the lead for laying down two or more pin pairs simultaneously in neat flowing patterns.

— H —

hard rule

A rule that is always followed. **See also:** soft rule, and Hard and Soft Rules in the *PADS Router Concepts Guide*.

hard breakout

Use of associated copper within a surface mount decal to simulate a dispersion route. The disadvantage to this method is that routing channels will possibly be blocked.

hatch

A copper fill pattern that uses horizontal and vertical lines at a specified width and spacing.

hatch outline

The outline of a copper pour shape after it has been flooded to differentiate it from the pour outline as originally drawn. When you draw the pour outline and then flood the shape, the outline often changes to accommodate the design rules. You can toggle between the pour outline and the hatch outline using the shortcut modeless command PO, or switch between these two Display modes in the Drafting Options.

HDI

An acronym for High Density Interconnect.

heat sink

An assembly that serves to dissipate, carry away, or radiate heat into the surrounding atmosphere.

high density interconnect

A class of packaging involving boards, substrates, and components using extremely small trace and spacing dimensions.

highlight

A user-defined color, usually white, used to denote that an object is selected.

high-speed checking

Using the Electrodynamic Checking utility. A simulator-type check that finds traces that may run parallel to each other close enough, and for a long enough distance, to cause cross talk.

hole plating

A fabrication process where solder flows through a drilled hole to connect the pads on either side of the hole, to provide connectivity between two or more layers.

HPGL

An acronym for Hewlett Packard Graphics Language, a standard pen plotter interface language.

— I —

IC (Integrated Circuit)

An acronym for Integrated Circuit.

IDF

Intermediate Data Format. An industry standard format used for exchanging data between electrical and mechanical design systems.

IMAPS

An acronym for International Microelectronics and Packaging Society.

impedance

Resistance to the flow of current in a trace. Measured in ohms.

in circuit testing

An exhaustive and thorough test of a PCB in final production that tests nets and unused pins for such things as correct voltage, correct parts, or bridging. Test point placement is critical for in circuit testing.

inaccessible nets

Nets for which you cannot define test points. DFT Audit analyzes all nets. If DFT Audit determines that test probes cannot access them, the nets are inaccessible (also called non-adaptable).

increased layer mode

A layer mode in which a design can consist of more than the default of 30 layers up to a maximum of 250 layers. The maximum number of electrical layers is 64, and the maximum number of non-electrical layers is 186.

See also: delay

INI file

An ASCII file, with the .ini file name extension, that contains startup parameters.

An INI file for Windows might contain the following information: graphics drivers, mouse drivers, fonts, and so on.

An INI file for programs might contain the following information: folder structure, display colors, default editors, and so on.

inner layers

Design layers other than those on the top or bottom of a printer circuit board. Inner layers may be routing layers, plane layers, or a combination of both.

installed options

PADS product features that you have bought and installed as part of the software package.

instruction pointer

A small yellow arrow in the Output window gutter that indicates the current line in a script or macro.

insulator

A material used to inhibit heat or electrical properties, such as current flow.

integrity check

A database check runs whenever a .job, .dxf, or .asc file loads. You can also initiate an integrity check while you are working by typing the "I" modeless command, then pressing Enter.

intensity

A value assigned to objects such as vias to weigh decisions made during the autorouting process in PADS Router. The higher the intensity, the less the item is used. For example, set a high intensity for via usage to minimize the amount of vias added to the design.

interconnect

A conductive connection between two or more circuit elements.

IPC

An acronym for Interprocess Communications within the PADS product.

irregular trace length

Sections or segments of differential pair traces not routed at the pair routing gap.

See also: differential pairs

islands

Small, isolated sections of copper pour that are not attached to anything.

— J —

JEDEC

An acronym for Joint Electron Device Engineering Council.

Joint Electron Device Engineering Council

JEDEC is the semiconductor engineering standardization body of the Electronic Industries Alliance, a trade association that represents all areas of the electronics industry.

jumper

A physical part used to cross over traces on most one layer PCB designs. Jumpers can be 0 Ohm resistors or wires stretched between jumper pads.

— K —

keepout areas

Areas that automatically ban objects. Depending on the keepout Properties, these areas may be set to prevent: placement of components, components that exceed a specified height, component drill holes, traces and copper, copper pours and plane areas, vias and jumpers, and test points.

keyview.exe

An executable file used to list the options programmed into your key. When you run keyview.exe, it creates a file named keyview.txt that contains a listing of your key options.

— L —

label

A label is a display instance of a component or jumper attribute. If you want to make an attribute visible in the design, you must instantiate it as a label associated with a component or jumper. You can do this in the Design Editor or the Decal Editor. You can have multiple labels based on the same attribute. An attribute has two parts—a name, and a value. A label of an attribute can have one of four visibility settings—None, Value, Name and Value, and Full Name and Value.

Latium rules

Latium rules are for advanced functionality in PADS Router. Some constraints that you can set in Layout are only used by PADS Router. For examples, see the Tune/Diff Pairs options, Fanout Rules, Pad Entry Rules.
The Latium checks include:

- component clearance rules
- component routing rules
- differential pair rules
- via at SMD rules

layer biased net

A layer biased net is any net with a design rule specifying a layer bias to one or more, but not all, electrical routing layers

layer pair

The assignment of two routing layers to switch between using the Layer Toggle command. On two layer boards, the toggle is automatically set between 1 (top) and 2 (bottom).

layer toggle

To switch between layer pairs while routing.

layers

A standard CAD database feature that separates graphical information into sheets of similar information such as dimensions, construction lines, or text. For PCB applications, this allows the various fabrication layers to be created and output separately.

layout-driven design

A PCB design process in which no schematic is created, and both the logical (netlist) and physical conformations of the board are defined in a layout tool. Also, a design created using this process. **See also:** schematic-driven design.

Layout.rep

The error report file that is created by the database integrity test and which is written to the \PADS Projects folder.

LCC

An acronym for Leadless Chip Carrier.

lead frame
A sheet metal framework that is etched to form an array of metal traces.

lead pitch
The sum of the lead width and lead spacing.

lead spacing
The distance between a component's adjacent leads.

Leadless Chip Carrier
(LCC) Ceramic IC package with no physical lead. There are only pads on the bottom of the package around the edges.

length matching
A same-length requirement where the entered value represents a minimum/maximum length tolerance for nets belonging to the same class.

length minimization
A routing feature that configures unroutes to the shortest available distances, or in a specific topology, to facilitate high speed routing.

LGA
An acronym for land grid array.

LibDir
The powerpcb.ini file entry that specifies the location of your library files.

libraries
The collection of part types, part decals, and drawn items included with a PADS product or created by the user.

Library Manager
The PADS feature that provides access to, and allows for modifying, the library of parts.

linked objects
When an object is linked, a presentation of the object and link to the source is contained within the framework of, and is a part of, the container application document. The object is linked to its source, and the source continues to physically reside wherever it was initially created. Therefore, the file that contains the object is smaller than if the object were and embedded object. See also: embedded objects.

Whenever you open the file that contains the linked object, the object checks the source to see if it changed since you last saved the file. If the source changed, then the linked object automatically updates.

LogCompressionMode
A powerpcb.ini file entry that controls recorded mouse movements in a log file. When set to one, the default, recording of compressed mouse movement is enabled. When set to zero, recording of all mouse movement is enabled.

logic family

The assignment of an electrical type by name, such as CAP (capacitor) or RES (resistor), to indicate the appropriate reference designator prefix such as C or R.

LOGMode

A powerpcb.ini file entry for online macro recording to a log file. When set to 0, the default, recording is disabled. When set to one, macro recording is enabled, and the next.log file is created.

loop

A pin pair that contains a route that branches off the original route, then branches back into the same route to form a loop.

loop routing

Used to create a loop in an existing route.

LPT port

A parallel printer port, usually referred to as LPT1 or LPT2.

— M —

macro

Internal objects that are handled using the macro engine vocabularies, and may or may not have the automation interface.

Manhattan distance (delta x + delta y)

Used to approximate unrouted net length for BoardSim. Add a percentage multiplier to account for indirect routing paths.

masking

The inhibiting of electrical interference between two traces on different layers due to separation by a ground or power plane.

material condition

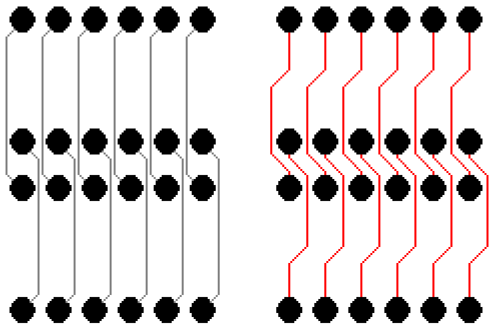
There are three material conditions when creating component decals. They are Maximum (providing the most robust solder joint), Nominal (providing a general purpose solder joint) and Minimum (providing the least possible solder joint for very dense designs).

MCM

An acronym for multichip module.

memory pattern

A collection of routes between memory devices that form a distinctly repeatable pattern.



menufile.dat

The file containing the structure and text of all lists and shortcut menus. The menufile.dat file must be located in the same folder as powerpcb.exe.

micrometer

One-millionth (10^{-6}) of a meter; about 40 millionths of an inch. Micrometer is synonymous with micron.

micron

A term used for micrometer. One-millionth (10^-6) of a meter; 25.4 microns = 1 mil.

minimum geometry

The smallest line width or spacing between lines or features on a semiconductor die.

miter

A diagonal segment or arc that replaces a corner.

miters pass

An autorouting pass that converts all 90 degree route corners to diagonal corners.

mixed plane layer

A plane layer that contains obstacles other than pads, such as routes, copper, or text.

modeless command

A command invoked through the keyboard. Commands include display options, design settings, and mouse click substitutions.

See also: chamfered

modify

To change information for a selected object.

moiré

Target-shaped objects located in the corners of finished artwork that are used to properly align each layer to others for design verification and fabrication.

monolithic device

A device whose circuitry is completely contained on a single die or chip.

mounted side

The side of the printed circuit board, either front or back, on which components are mounted.

mounting holes

Many (but not all) boards have mounting holes. Mounting holes are typically located around the perimeter of a board (most often in the corners). They are drilled holes, used to mount a printed circuit board to the finished product (for example, a mother board mounted to the computer casing), or used to attach bolt-on components to the printed circuit board (for example, stiffeners and ejector tabs).

There are two types of mounting holes: plated and non-plated. Plated mounting holes have copper inside the hole and usually have a large annular ring of copper on both sides of the board connected by this copper cylinder (plating) inside the hole. These holes are typically connected to the GROUND bus or plane on the board and provide a method for grounding the board circuitry to the enclosure (for shielding purposes). The mounting hole ring diameter is usually slightly larger than the diameter of the head of the screw that will be used to fasten the board to the mounting device within the enclosure. Non-plated mounting holes are used for the same purpose, the only difference being that they are not internally plated and do not have a copper ring, therefore they are not used for grounding the board to the enclosure.

Tips:

- Plated mounting holes cannot be used as tooling holes as the thickness of the copper plating can vary and violate the close tolerance required by a tooling hole. Non-plated mounting holes can sometimes work double duty as tooling holes because there is no internal plating, therefore the tolerance of the hole size can be more closely controlled and fit within the requirements of a tooling hole.
- Use the Decal Editor and the Pad Stacks dialog box to create tooling holes. Save the single-terminal object as a part to the library for reuse.

See also: Creating Terminals, Editing Pad Stacks

multichip module

A package with multiple dice that is 20% or more silicon, has 100 or more I/O on a substrate, and four or more layers.

multilayer PC board

A design that contains routing and/or plane layers, in addition to those on the front and back side.

— N —

nail diameter

The diameter of the test probe.

NC drill

An abbreviation for numerical control drill. This technology involves producing an output file containing the x-y location and drill size for each hole, then feeding this information into a machine for automated hole drilling.

negative

A photographically produced reverse image of a plane layer. This allows cleared areas, or airgaps, to be created using normal drawing techniques. When reversed, all areas not drawn for clearance become the actual planes.

nested embedding

Nested embedding occurs when you insert an object using OLE into another object. For example, inserting a PADS Logic schematic into your PADS Layout Design or inserting a Microsoft Word document into a schematic.

nested macros

Macros called from other macros.

net

All pin pairs composing one individual signal. Nets contain at least one subnet, but may contain more than one.

See also: subnets

net class

A collection of nets with a common set of design rules.

net length rules

Rules that control a net's or pin pair's routing length.

The following high-speed rules are examples of net length rules: minimum/maximum length, matched length, and differential pairs.

The phrase controlled length net refers to nets that have length rules, or nets with pin pairs that have length rules.

net name

A specific name given to a net to describe its function; for example, GND, PWR, or DATA0.

Tip: The maximum net name length is 47 characters. You can use any alphanumeric characters except { } * and space.

netlist

A point-to-point connection list for each signal in a design, providing the reference designator (part name) and pin number.

netlist file

A PADS ASCII file containing all of the nets in a design, including all component pins that make up the nets. The file may also contain a list of all parts in a design, and/or the settings that control the substrate bond pad numbers and functions for newly created substrate bond pads.

netlist.fmt

The ASCII setup file for the report format that produces a netlist without pin information.

network security

Use of one security key, programmed with multiple options, for network use with one or more systems at a time.

next.ini

A file produced in the C:\MentorGraphics\<*latest_release*>PADS\SDD_HOME\Programs folder when the powerpcb.ini entry LOGMode is equal to one. The next.ini file is a copy of your powerpcb.ini file at the time next.log is written.

next.log

A file produced in the C:\MentorGraphics\<latest_release>PADS\SDD_HOME\Programs folder when the powerpcb.ini entry LOGMode is equal to one. The next.log file records all activities within a PADS Layout session so that they can be replayed to reproduce a series of steps or used to illustrate a problem.

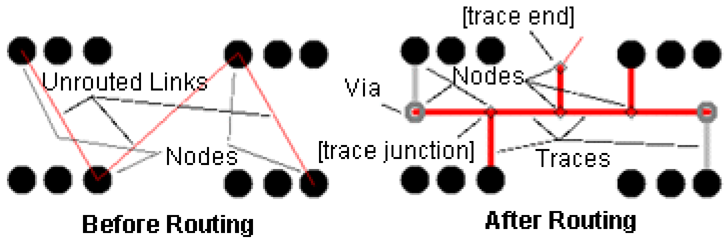
NLM

A type of file used by a Novell network. For example, the ssisrvr.nlm file is the PADS Layout network security NLM.

node

A point along a trace where traces join other traces (T junction), where traces transition to other layers, or where traces end at pins, vias, or floating end points. Specifically, a node can be any pin, via, copper, trace junction, virtual point, or trace end.

See also: virtual point



node-locked

A license for a specific Host ID.

non ECO registered parts

These parts are found in the schematic and layout design. Parts not selected as an ECO Registered Part on the General tab of the Part Information dialog box are non ECO registered parts.

- A schematic non ECO registered part is required in the schematic but has no place in the layout of the circuit board. For example, a chip socket shown in the schematic for inventory tracking in the bill of materials.

- A layout non ECO registered part is required in the layout design but has no place in the schematic. For example, a plated and grounded mounting hole.

non electrical parts

Parts with no pins. For example, a mounting screw shown in the schematic for inventory tracking in the bill of materials.

non plated holes

Pads that are not reflowed with solder, usually reserved for mounting holes. Nonplated holes are not drilled with an oversize to accommodate the solder flow.

To determine plating status, in PADS Layout, use the Pad Stacks Properties dialog box. In PADS Router, use the Pad Stack tab in the Pin Properties dialog box.

nrus.exe

A program used with Novell network security used to track the total number of network security options, the available network security options, and the network security options in use.

nudge

A placement feature that relocates parts in order to make room for new parts being placed. Movement is based on previously defined clearance rules.

— O —

object

One discrete item in the design. For example, an object may be a route segment, a part, a drawing line, or a via.

object mode

Start a command by selecting one or more objects and then selecting the command to perform on them.

See also: verb mode

obstacles

Objects that block routing. Obstacles can be protected pins, vias, traces, keepouts, board outlines, and hatch outlines.

Exception: Although hatch outlines are obstacles, when you interactively route, autoroute, or edit traces, hatch outlines are removed from the design so routing can complete.

Visible copper pour hatch outlines, copper pour outlines, and plane area outlines are not obstacles to interactive routing, autorouting, or route editing.

odd pad shape

A pad that requires a special aperture, or plot sequence, to create. In PADS Layout, in the Pad Stacks Properties dialog box, the odd shape setting should not be confused with trying to create a custom shaped pad which is accomplished by drawing copper in the decal and associating it to the pad.

offline plot
A plot that is sent to a file before it is copied to a printer or plotter for processing.

offset
The distance by which rectangular or oval pads are moved away from the electrical center of the pad stack.

offset pads
Rectangular or oval pads moved off the electrical center of the pad stack to facilitate identification/selection, or for a special design consideration.

one pin nets
A net that contains only one pin. Also called single pin net. In PADS products, a net must have a minimum of two pins.

online DRC
A PADS feature that actively checks established-design rules during routing or placement operations.

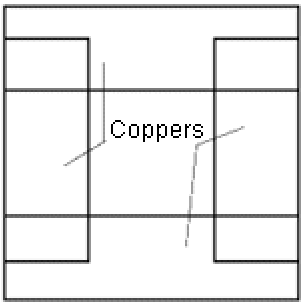
online plot
A plot sent directly to a printer or plot.

open cluster
Clusters that you can delete or replace during automatic cluster creation.

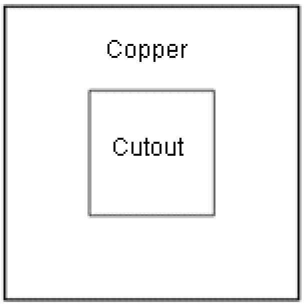
optimization
Rearranging placed parts and/or swapping pins and gates on parts in order to minimize trace lengths and reduce the number of vias required for routing.

optimize pass
An autorouting pass that analyzes each route and tries to improve the quality of the route pattern by removing extra segments, reducing via usage, and shortening routed trace lengths. This pass includes glossing and smoothing processes.

overlapping coppers
Overlapping coppers are combined into one copper area, with possible cutouts.



Overlapping Coppers

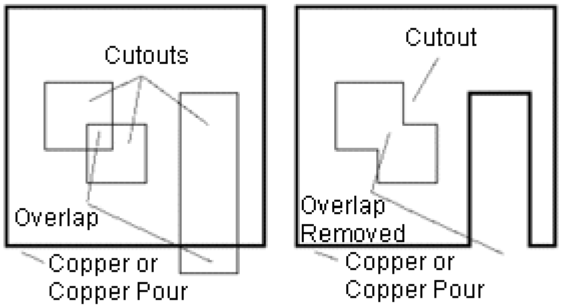


Resultant Copper After Loading into BlazeRouter

See also: coppers, copper connectivity

overlapping cutouts

Overlapping cutouts are combined into one cutout area.



Copper with Cutouts Created in PowerPCB | **Resultant Copper Shape After Loading into BlazeRouter**

See also: cutouts

overlapping segments

Multiple trace segments stacked on top of one another on one layer.

— P —

package

The protective container for an electronic component with terminals to provide electrical access to the die components inside.

pad entry

The point where a trace entering or exiting a pin first crosses the edge of a pad.

pad entry angle

The command in PADS Layout and Router that establishes the angle at which a trace enters a pad. This may be orthogonal (90 degrees), diagonal (45 degrees), or any angle.

pad function

The die signal name to which the component bond pad is connected.

pad number

The number of the component bond pad.

pad oversize

On plane layers, pads that are larger than normal, to generate proper clearances when the image of the pad is printed in a negative format.

Note: Pad oversize is measured from the center of the pad, not the perimeter. For example, if you have a 3 mil oversize, the measurement is actually1.5 mils in each direction from the center of the pad.

pad stacks

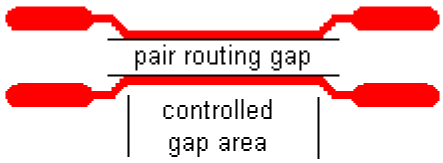
The combination of pads, drills, and pastes, for example, on a pin or via, for each layer of a design, stacked directly on top of one another.

pagesfile_sys

The Windows NT swap file that allows virtual memory to be written to disk.

pair routing gap

The fixed edge-to-edge clearance between the traces in the controlled gap area for a differential pair.



See also: controlled gap area, differential pairs

paired layer

The start and end layers used by the layer toggle command when changing layers while routing. It defines the default layers to use when you make layer changes.

palette

A user-definable color chart in the Display Colors dialog box.

pan

Up and down or side-to-side movement of the screen without zoom or redraw. Use the scroll bars or postage stamp to pan.

panning

Moving the view horizontally or vertically without changing the size of the design on your screen.

parallel port

A printer port, usually referred to as LPT1 or LPT2.

parallelism

Traces on the same layer that are checked for running parallel to each other.

The traces are subject to crosstalk if they run parallel to each other too long and the gap between them is too short.

parasitic

An undesirable stray capacitance, inductive coupling, resistance leakage, or undesired transistor actions.

parent object

The object to which individual design elements, such as lines, arcs, or corners, belong.

part decal

The physical representation of a part, or footprint, assigned to the part type.

part list
An output listing of all parts belonging to the same design. This normally includes the reference designation, part name, and part type, and total number of each type.

part name
The text for each part that indicates the reference designator.

part outline width
The line width of 2D line shapes, created in the PCB Decal Editor, that represent silkscreen or documentation data within a part decal. The shapes do not include text, reference designators, or copper with the decal.

partial route
Partial routes are uncompleted routes where the ratsnest flightline is still visible. This occurs when you click End while routing or bus routing, or when you delete a trace segment. See also dangling route.

partial via
A via that does not travel through all of the board's electrical layers. The blind via and buried via are both types of partial vias.

parts1.fmt
An ASCII part list format file for the report file generator that consists of a reference designator, part type, and logic type.

parts2.fmt
An ASCII part list format file for the report file generator that consists of a part type, reference designator, and part description.

paste
A substance used to attach each pin of a surface mount device to a PC board.

paste mask
An artwork layer with a paste location for all pads of surface mount components.

patterns pass
An autorouting pass that searches for, and routes, groups of unrouted connections that can be completed using a typical C routing pattern, Z routing pattern, and memory pattern.

PBGA
An acronym for plastic ball grid array.

PDF configuration
A set of PDF Configuration dialog box control settings. A PDF configuration can be saved as a .pdc file and reused to create PDF documents for multiple designs.

PGA
An acronym for pin grid array.

photoplotting

Using a machine to create printed circuit board fabrication artwork. The machine creates artwork by exposing clear film to light or by rasterizing an image onto clear film.

physical design reuse

A collection of design objects that you want to reuse, which are associated with one another. The collection of objects can be saved to a file.

physical design reuse elements

The objects that compose the reuse. They can include components, routes, vias, text items, and other elements.

pick and place

An automatic printed circuit board assembly machine, driven by outputting the part type, location, and orientation of suitable parts from a design.

pin

The through-hole or surface mount terminal that represents a connection to a part. Pins are also referred to as pads in pad stacks.

pin array/pin grid array

A package with pins distributed over much or all of the bottom surface of the package in rows and columns.

For more information, see Pin Wizards Dialog Box, BGA/PGA Tab.

See also: pin types

pin number

Within a component, the numeric or alphanumeric designation that distinguishes pins from each other.

In the Status bar, pins are identified using the following format:

```
Pin:[Component name].[Pin number].[Pin type]
```

For example:

```
Pin:Y1.N.Nonelectrical
```

See also: pin types

pin pair

The combination of a trace or unroute, and the pins on either side. A net can contain one or more pin pairs.

pin pair group

A collection of pin pairs that share common design rules.

pin type

A designation that indicates the electrical characteristics of the pin such as Source (S), Load (L), Terminator (Z), and Undefined (U). For example, U1.1.S may appear on the status bar.

pin types

Pins and pin pairs can be identified by one of the following pin types:

- Source
- Bidirectional
- Open Collector
- Or-Tieable Source
- Tristate
- Load
- Terminator
- Power
- Ground
- Nonelectrical

Pin types make up the last portion of the pin identifier in the Status bar. For example:

Pin:U10.C.Open Collector

See also: pin number

placement check prints

Generate a CAM Assembly drawing to make a placement check print.

After the PCB Designer receives a schematic and a netlist from an Engineer, they (typically) place the components onto the board in a manner that best suits the routing of the board. Sometimes, the placement better suits the intentions of the Board Designer than the Engineer, so before routing proceeds, the Engineer will request to see a set of Placement Check Prints. Placement Check Prints show the placement of all components on both sides of the board, so the Engineer can review the locations and confirm the Designer has correctly placed the components. These Placement Check Prints typically require agreement from both the Designer and the Engineer before routing can proceed.

placement operations

Operations where parts are relocated or added to a design in order to optimize an existing placement.

plane hatch outline

The outline of a plane shape after it has been flooded to differentiate it from the plane pour outline as originally drawn or the plane indicators outline. When you draw the plane pour outline and then flood the shape, the outline often changes to accommodate the design rules. You can switch plane display modes using the shortcut modeless commands SPD, SPI, and SPO. Or, switch between the Mixed plane display modes in the Split/Mixed Plane Options.

plane indicators outline

The outline of a plane shape after it has been flooded and then changed to Plane thermal indicators display mode to differentiate it from the plane pour outline as originally drawn or the plane hatch outline after it has been flooded. When you enable this display mode, the active layer must be set to the plane layer in order to see the thermal indicators.Setting the active layer to a CAM plane layer also displays the thermals of the CAM plane. You can switch plane display modes using the shortcut modeless commands SPD, SPI, and SPO. Or, switch between the Mixed plane display modes in the Split/Mixed Plane Options.

Restriction: After you change active layers, you will need to redraw the view to see the new layer's thermal indicators if you do not have the *Active layer comes to front* option enabled.

plane layers

A design layer where the entire surface is covered by copper, except for information not connected to the plane.

plane nets

Nets assigned to plane layers.

plane pour outline

The outline of a plane shape after it has been drawn to differentiate it from the plane hatch outline after it has been flooded or the plane indicators outline. You can switch plane display modes using the shortcut modeless commands SPD, SPI, and SPO. Or, switch between the Mixed plane display modes in the Split/Mixed Plane Options.

plastic ball grid array

A surface mount package with an array of solder sphere-shaped interconnects arranged across the bottom surface of the package substrate.

plastic leaded chip carrier

A common surface mount package with leads on all four sides, used as a socket for devices that cannot withstand the heat of the reflow process, and/or to allow for easy component replacement.

plated holes

Drilled holes that have copper covering the inside surface of the hole, and which are connected to a pad on each side. Plated holes pass connectivity from one layer to others.

plating tail

A route that connects BGA vias to a plating bar or bus bar.

PLCC

An acronym for plastic leaded chip carrier.

plicense.exe

The program used to verify and program your security key during a field upgrade process.

polar decal

A single-radius, circular pattern decal with through-hole pins.

polar SMD decal

A single-radius, circular pattern decal with SMD rectangular or finger pads.

polygon

A closed shape consisting of three or more line segments.

positive

An image of a plane layer where cleared areas, or airgaps, are created using normal drawing techniques. When reversed to create a negative, all areas not drawn for clearance become the actual planes.

pour outline

The outline of a copper pour shape after it has been drawn to differentiate it from the hatch outline after it has been flooded. You can toggle between the pour outline and the hatch outline using the shortcut modeless command PO, or switch between these two Display modes in the Drafting Options.

power plane

The plane layer where power supplied to the printed circuit board is dispersed to the proper pins of each component requiring a power source.

powerpcb.ini

The PADS Layout initialization file for default settings.

powerpcb.mdb

The PADS Layout message file that contains error messages, prompts, and other miscellaneous text strings. This file must be located in the same folder as powerpcb.exe.

powerpcb.reg

A file that defines all Registry keys required for the proper registration of PADS Layout OLE components. In addition, other programs acting as clients access the PADS Layout Automation Server through this Registry file.

The installation program automatically creates this file and saves it in the same folder as powerpcb.exe. If errors occur in the Registry, or if this file is corrupted, you can restore the contents of the file.

preferred routing direction

In the main GUI combo box, the Horizontal [H] or Vertical [V] designation next to a routing layer name. This designation indicates optimal direction for routing completion and can be set by the user or the system.

prepreg

A resin pre-impregnated sheet used to bond substrate laminate-pair layers together when a multilayer board is pressed together.

preset files

Library IQ files that allow you to save the preference settings you have established for a die design and use them in other designs. The Bond Pad Preferences files have a .pre file extension.

preview of CBP assignments

A preview that displays the substrate bond pads and wire bonds created when component bond pads are assigned to rings. This preview appears in the work area when the Assign CBPs to Rings dialog box is active.

preview of SBP guides

A real-time preview that displays any changes made to the number, geometry, or location of SBP guides. This preview appears in the work area when the Wire Bond Wizard dialog box is active.

well as in the design in which you place the reuse.

See also: private nets

primary objects

Primary object groups in the Object View tab of the Project Explorer contain non-removable design elements shown in a high-level object hierarchy. Primary objects are:

1. layers
2. components
3. part decals
4. net objects (including nets and pin pairs)
5. via types

private nets

Nets that are contained completely within a physical design reuse.

See also: public nets

probing

The testing of individual IC dice using very fine probes to temporarily connect each to a test computer, in order to verify operation.

properties

A set of dialog boxes used to view or edit information about the selected object.

protect

Glues the routes and attached vias and prevents the autorouter from modifying them in any way.

protected routes

Traces that are placed in a protected state by Route Protection. This means that they cannot be moved or modified.

protected traces

Traces placed in a protected state (cannot be moved, or modified).

protected unroutes

Unrouted connections, or the unrouted portion of a partial route, that are placed in a protected state by the Route Protection feature. This means that they cannot be routed, moved, or modified.

public nets

Nets that are partially contained within a physical design reuse. Public nets exist in the reuse, as preferred routing direction

In the main GUI combo box, the Horizontal [H] or Vertical [V] designation next to a routing layer name. This designation indicates optimal direction for routing completion and can be set by the user or the system.

pulling an arc

Creating an arc from an existing line segment, where the diameter is derived from the line length.

— Q —

QFP

An acronym for quad flat package - a surface mount IC with leads on each four sides.

quad

A square-shaped IC with pads on each of its four sides.

Quick Filter Settings

The shortcut menu selections available when no items are selected. These choices set the selection filter for commonly used tasks, allow quick access to the Find command, and Select All items as specified by the Selection Filter.

quick measure command

The Q modeless command which attaches a measurement line to the pointer and displays dx, dy and hypotenuse information, depending on pointer movement.

— R —

radial lead

A discrete part with pins that protrude straight down and do not extend beyond the perimeter of the component body. An example of this is a capacitor.

RAM

An acronym for Random Access Memory. The volatile (on chip rather than on disk) memory area available to the system for program operation.

range select

To select a series of geometric or route segments by first clicking on the start segment, then pressing and holding Shift and right-clicking on the end segment.

ratsnest

A term used to describe the display of all of the unrouted connections in a design. Also known as air lines or unroutes.

raw database

The raw database contains all components in the open database, regardless of assembly variants. When created, new assembly variants are based on the raw database, meaning that until you uninstall or substitute, a new assembly variant includes every component in the raw database.

read-only attribute

An attribute whose value cannot be changed in PADS product dialog boxes. You can, however, modify attribute properties and the Attribute Dictionary entry, and can modify the attribute value in the library.

real width

To display traces at their specified width, as opposed to displaying them as one pixel centerlines.

real-time redraw

A feature that enables active regeneration of objects in the display any time the screen is redrawn. When you disable real-time redraw, regeneration occurs in the background, and the display is refreshed all at once after the background regeneration process is completed. Screen regeneration is quickest when real-time redraw is disabled.

record locking

Allowing two or more users to access the same library component at one time. However, only one user has access to save the component.

recover

Resolving an installation or operational issue, or salvaging a corrupt database by executing a specified series of steps.

redo

Repeats actions which have been undone.

redraw

Refreshes the display of the current screen image and the cursor.

reference designator

An identification assigned to each of a design's parts in order to distinguish them from other parts of the same type when placed on the printed circuit board. A reference designator is usually in the form of a letter that represents the part type, followed by a number. For example, C2 may represent the second capacitor in the design. PADS Layout permits you to renumber the reference designators in one of several specific patterns allowing you to quickly find a part among thousands on the manufactured board. The reshuffled numbers that are rearranged in PADS Layout are backward annotated to the schematic software to keep the designs synchronized.

relative coordinates

Coordinates that are based on a start point instead of the system origin.

rename

To assign a different name to a part or net.

reroute

Specifying that a trace, or a portion of a trace, follow a path different than the one currently being taken.

restricted layer

Layers that are either disabled for routing or have been disallowed by layer biasing rules. When a layer is restricted, routing is not permitted on the layer. Layers can be restricted for specific objects, such as a net or a pin pair.

restricted via

A via that is not permitted for use in the Routing Rules of PADS Layout or Via Biasing properties in PADS Router at any level of the rule hierarchy.

reuse

See preview of CBP assignments.

reuse definition

The master copy of the physical design reuse that is saved to a file. The saved version of the physical design reuse is the version you should use in other designs. All resulting instances of the physical design reuse are based on this file.

reuse type

A name that identifies the type of reuse being created. A reuse type is equivalent to a library part type.

ring geometry

The shape of the die flag ring. The following shapes, or ring geometries, are supported: rectangle, rounded rectangle, chamfered rectangle, and arced shape.

romansim.fnt

The default file that contains definitions for the graphics for the PADS stroke font, used to display text in PADS products when system fonts are not in use.

rotate

The command that rotates by 90 degrees a component or object around its axis or selection point.

route

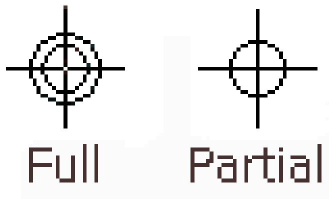
To create a metal etch trace of a specified width between pads.

route-completion target

This crosshair or bullseye symbol appears when routing from one pin of a pin pair to another pin or when rerouting a trace segment.

The partial target appears when you are overtop of an electrically compatible pin, but you have settings that are preventing you from routing to it - for example, the unroute of the pin pair you are routing is protected (you have selected the Protect Unroutes check box in the Pin Pair

Properties).



route loops

A pin pair that contains a route that branches off the original route, then branches back into the same route to form a loop.

route pass

The autorouting pass that is the core pass that performs the majority of autorouting. During this pass, PADS Router attempts to sequentially route each unroute until all connections are attempted. The Route pass contains serial, rip up and retry, push and shove, and touch and cross processes.

routed length

The trace length monitor calculates routed length as the cumulative length of the trace. This value always starts at zero unless you start routing from the end point of a partially routed trace, in which case the routed length includes the partially routed trace length. If the trace has branches, then the length is calculated from the branch point.

See also: estimated total length, unrouted length

routes

A series of traces that represents routed connectivity.

routing angle

The angle applied to adjacent segments as new corners are added to traces. For example, an orthogonal routing angle means adjacent segments will be created at 90-degree angles to each other.

routing order

The order in which the autorouter routes components, nets, and net classes.

routing pass types

There are several pass types, each of which is designed to complete a specific task. Each pass may use more than one algorithm and may also perform a number of subpasses.

The following pass types are supported:

- center pass
- fanout
- miters pass
- optimize pass
- patterns pass
- route pass
- test point
- tune pass

routing strategy

The collective information PADS Router uses to autoroute a design. This information includes which pass types PADS Router should perform, whether to protect the resulting traces, and what intensity to assign to objects.

ru.cfg

A configuration file used by the nrus.exe program for Novell network security support.

rule values

The values of any item, regardless of its default rules or rules set assignments.

rules

An established set of conditions for a given net or design.

rules set

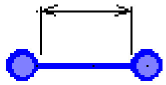
A specific set of user-assigned nondefault rules such as pin pair, groups, or classes.

— S —

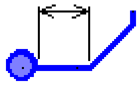
same net checking

Checks clearances between objects along the same net, as specified in the Clearance Rules dialog box. Object to object checking includes:

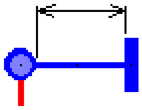
- Pad edge to pad edge.



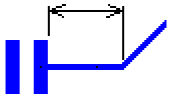
- Pad edge to inside corner of trace.



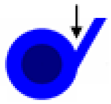
- SMD edge to pad edge.



- SMD edge to inside corner of trace.



This check prevents solder bridging during board manufacturing caused by acute angles between conductive objects such as the acute angle between pad and trace shown below.



same net rules

Specifying conditional settings, such as spacing, for connections belonging to the same signal name or net, rather than against other nets.

SBP

An acronym for Substrate Bond Pad.

SBP fanout

A single-segment fanout that connects SBPs to any-angle coupling traces.

SBP guide

The virtual snap line along which substrate bond pads are aligned during wire bond fanout generation. Each SBP guide determines the alignment of the substrate bond pads that are associated with the SBP ring aligned with this SBP guide.

SBP ring

A set of substrate bond pads aligned along an SBP guide. A substrate bond pad belongs to the ring on which it is aligned. In creating a wire bond fanout, you assign each component bond pad to a specific SBP ring.

schematic-driven design

The “standard” PCB design process, in which a netlist is first created in a schematic tool, and then passed to a layout tool, where the parts are laid out on the board and the connections routed. **See also:** layout-driven design

scribe line or saw line

The separation between adjacent dies on the wafer. This path is used as the cutting area in sawing a wafer into the individual dies.

search

To locate specified information. One search method is to use the Find command.

secondary objects

Secondary object groups break primary objects into a more detailed hierarchy. You can add individual items to and remove individual items from secondary groups. Secondary objects include:

1. net class
2. pin pair group
3. conditional rule
4. matched length net group
5. matched length pin pair group
6. differential pair

seed
A part used by Cluster Placement, during cluster building, to search outward for other parts to add to the cluster.

segment
A single drafting line, path, or trace, defined by a beginning x/y coordinate and an ending x/y coordinate.

segmentation fault
The termination of a PADS product due to a system crash or illegal instruction executed.

Select All
The Edit menu command that lets you select all items of a type specified in the Selection Filter. This option is also accessible from the shortcut menu when nothing is selected.

select mode
Point to the object and click the left mouse button. Select the command to perform on the object.

selecting
To highlight an object for editing, moving, viewing properties, or deleting.

selection filter
The dialog box inhibiting or enabling the selection of specific items.

serpentine route
A route that connects an any-angle coupling trace and a BGA pad, forming a snake-like pattern as it travels through the BGA.

session log
Information on the current session that appears in the Status tab of the Output window.

shape
The Selection Filter setting that enables or disables selection of an entire geometric object, not just its individual segments.

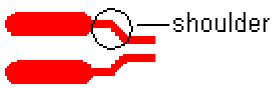
shared libraries
Libraries that can be accessed by more than one user across a network.

shielding
Specifying that one net be routed around another to provide protection from interference.

shortcut keys
A key sequence that starts a command directly from the keyboard and without navigating through menus.

shortcut menu
A menu listing the possible actions to perform, based on the selected object.

shoulder
The part of the differential pair trace between the source pin and the gathering point, or between the split point and the destination pin.



See also: differential pairs, gathering point, split point

signal
Voltage or current that is transferred between component pins by an electrical conductor.

signal pins
Pins that have a signal net, such as GND, assigned by the schematic capture program PADS Logic during part type creation.

silkscreen
An artwork layer containing the reference designator and component outline of all parts, used for the final board fabrication process.

single-sided board
A design where all pads, routing, and parts are placed on one side of the board.

single-sided die
A die that has substrate bond pads and a BGA grid array on the same side of the die.

See also: documentation layers

sizing handles
Small, black squares that appear at the corners and along the sides of a rectangular area that surrounds a selected nontext object.

sketch route
A PADS Layout command that reroutes existing traces by allowing you to draw a new route path using the pointer.

slice
Another term for wafer.

slotted holes

Oval holes in a printed circuit board, which may be plated or non-plated.

SMD

An acronym for Surface Mounted Device: the pin of a component that is attached to the PCB only on an outer surface and does not require drilled holes for component mounting.

smoothing

A command that automatically removes unneeded corners and segments and centers trace patterns between route obstacles.

SMT

An acronym for Surface Mount Technology.

snap modes

Various modes, available during dimensioning, that force the pointer to pick points based on of the following parameters: intersection, any point on a line, any point in space, entire segments, the center point of an arc, and so on.

soft rule

A rule that is ignored if it alone prevents route completion. **See also:** hard rule, and Hard and Soft Rules, in the *PADS Router Concepts Guide*.

SOIC

An acronym for Small-Outline Integrated Circuit.

solder

A metal alloy used to attach each pin of a device to a printed circuit board.

solder dam

A small amount of solder mask used to limit molten solder from spreading further onto solderable conductors, in an area where solder mask is purposefully absent.

solder mask

The artwork layer for a nonconductive material that covers the entire board, except for pad locations. The solder mask provides a protective covering and prevents shorts during wave and reflow solder processes.

solder mask reliefs

Some components have large areas that need to dissipate heat. Others have large metallized areas (that are not pins) that need to be soldered to the board. In order to expose the copper area beneath these parts for soldering, the solder mask layer must have a cutout representing these areas. These cutouts are called solder mask reliefs. When the distance between pads of a fine pitch component is too small, the webs or fingers of solder mask between pads can break and wander on the board surface. To prevent this, a solder mask relief is applied to entire pad areas of a component. This is commonly called gang relief.

solder side
The back or bottom side of a printed circuit board. Solder side is named for the post assembly process, where the board is run through a special bath to solder all pins.

source
A pin type that indicates a signal radiating from the pin.

SPECCTRA
The product name for the Cadence Design Systems autorouter.

special symbols
Alternate decals that you specify as connectors. You can associate a logical pin type with each alternate to provide a graphical indication of the connector pin function in a schematic.

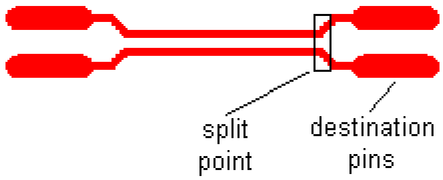
spider bonding
A method of connecting an integrated circuit die to its package leads. A lead frame is placed over the chip and all connections are made by just one operation of a bonding machine. TAB methods use this approach to interconnection.

spin
The PADS command that rotates a component or object around its axis or selection point.

split
The command that creates a new corner at the pick point of the selected trace, allowing it to be rerouted.

split plane
A solid copper plane layer divided into two or more sections in order to isolate electrical signals from each other.

split point
The point near the destination pins where differential pair traces are no longer routed together and where the traces are routed individually to completion.



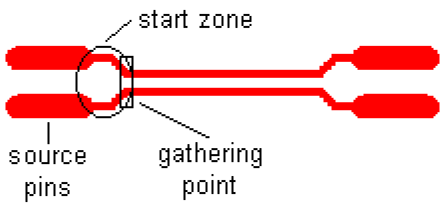
See also: differential pairs, pair routing gap

ssiact.exe
A program used to recommend set statement settings to properly adjust port access times for a security key.

stackup
The metal and dielectric layers used to implement the body of a printed circuit board. A signal metal layer carries signal traces. A plane metal layer is tied to a DC voltage. A dielectric layer is made from non-conducting material and separates two metal layers or coats the board surface.

start zone

The part of the differential pair between the source pins and gathering point.



See also: differential pairs, gathering point

starting layer

The first layer in a drill pair or via definition.

step-by-step mode

A mode in which the debugger runs a single line of code at a time.

stitching vias

Any SMD via, through-hole via, or partial via added to nets (on traces or within plane areas) in a repetitive manner. You can add these vias, also called free vias, for various purposes, including current and thermal needs. For example, you can place stitching vias in a plane area to provide conduction between two plane areas. You must assign stitching vias to a net, but they do not have to have traces attached to them.

strategy

A set of options that defines how a board should be autorouted.

strong

Places cluster members as close together as possible during placement operations. The minimum distance for placement is the same as the distance for part clearances in Design Rules.

structured attributes

Attributes that are related to each other by the prefix in their name. For example, the DFT attributes such as DFT.Nail Count Per Net, DFT.Nail Number, and DFT.Nail Diameter are structured attributes. Together, these structured attributes make an attribute group.

stub

A trace that enters another to create a T-junction. Stub lengths can be checked by the EDC program.

submicron

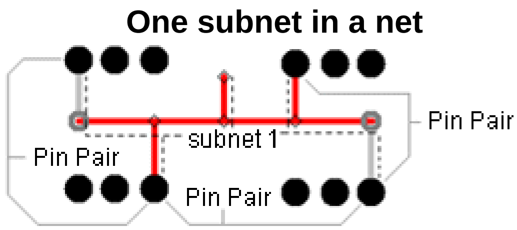
Dimensions smaller than one micron.

subnet

A collection of all traces and vias connecting two pins. Subnets are joined only through their common component pins and not through other nodes, such as a trace junctions, vias, or virtual points.

Subnets help to avoid errors or confusion caused when pin pairs of a net have unique, rather than common, design rules.

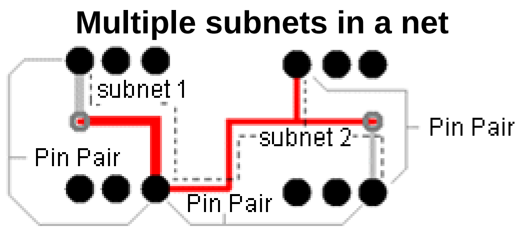
See also: node, subnet, connected islands, virtual point



subnets

If a net has at least one pin pair with a unique design rule, such as a trace width difference, the net is automatically divided into subnets. If two pin pairs having the same rules are separated by at least one pin pair with different rules, the pin pairs are considered separate subnets. Therefore, subnets are islands of pin pairs that form an unbroken fragment within the net, where each fragment has uniform rules.

See also: subnet, connected islands



substrate

A material between copper laminate layers that comprise a laminate pair, or a laminate set in the case of completed multilayer boards.

substrate bond pads

Copper areas on the substrate to which a die's wire bonds are connected.

surface mount device

Pads are glued to the board rather than inserted.

swap file

The file created when a program runs out of RAM memory and writes memory to disk.

swapping

A placement optimization process that exchanges pins, gates, or entire parts.

The product .ini file entry that specifies the path for the PADS product configuration files.

system attribute

An attribute that is set by, used by, and critical to a PADS product, an external program, or Automation script (such as Sax Basic). You cannot modify the properties of a system attribute or modify the Attribute Dictionary entry for a system attribute.

system toolbars

System toolbars are specific to the PADS programs. They feature several system toolbars, such as standard, routing, selection filter.

SystemDir

The product .ini file entry that specifies the path for the PADS product configuration files.

— T —

T junction

A trace that branches into another.

TAB

An acronym for source.

tacks

Small, diamond-shaped objects that anchor traces to their current location. Tacks are automatically generated under certain conditions and may also be manually added to a selected trace.

tandem traces

Traces on different layers that are checked for running parallel to each other.

The traces are subject to crosstalk if they run parallel to each other too long and the gap between them is too short.

Tape Automated Bonding (TAB)

A packaging method where silicon chips are joined to patterned metal traces, or leads, on polymer tape to form inner lead bonds which are attached to the next level of the assembly, typically a substrate or board.

Tape Ball Grid Array (TBGA)

A TAB packaging method in which tape automated bonding leads are replaced by a ball grid array.

TBGA

An acronym for Tape Ball Grid Array (TBGA).

teardrop

A triangle shape that provides a smooth transition from a trace to a pad.

terminal

The electrical center of a pin, as defined in the part decal.

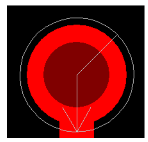
terminator

A pin type for high-speed circuit configurations that indicates a terminating resistor to match impedance of the trace. Terminators are used to reduce signal reflections that cause poor circuit performance.

test point

A test point is a group of objects that serve as a contact between the electrical element of the board and the probe of the testing device. A test point can also be a point on a node of a net, component pin, or via. Test points can also be a point on an unused component pin, such as a component pin that is not incorporated into any net.

When the via or pin is flagged as a test point, and Show Test Points is checked on the Routing tab of the Options dialog box, a down arrow symbol is drawn on the via or pad in the design:



test point pass

This autorouting pass analyzes the testability of the design, determines which nets require testing, adjusts the routes, and inserts test points to improve testability. You can select whether to add test points during routing or after routing.

See also: The "To Assign Test Points During Routing," "To Assign Test Points After Routing," and "Using Automatic Test Point Placement" topics in the PADS Router Help for more information.

testpnts.fmt

An ASCII file containing information about test points, including the test point name, the signal name, and the x/y coordinates. The report file generator creates this file.

thermal

A multi-spoke connection of a through hole pin pad, via, or surface mount pad to a plane area or copper pour area.

thermal compression bonding

A method of wire bonding that does not use an intermediary metal or melting, but rather the flow of materials resulting from the combination of heat and pressure. It is also referred to as thermocompression bonding.

thermal relief

A spoke-shaped pattern that connects a via or pin, in the same net as the copper pour, to the surrounding copper. Thermal reliefs provide good pin soldering by preventing heat from dissipating throughout the plane layer.

thick-film process

A hybrid microelectronic process where conductors, insulators, and passive components are screened from special pastes onto the substrate.

thin-film process

The use of deposited films of conductive or insulating material, which may be patterned to form electronic components and conductors on a substrate or used as insulation material between successive layers of components.

through holes

Although there are non-plated through holes, this term is used interchangeably with plated through holes. It indicates that the hole has internal plating. There are two basic types of components that can be placed on a circuit board: Surface Mount Technology (SMT) where the parts are soldered to the surface of the board, and through hole (TH) components, where the components have wire leads that are soldered into plated holes that go through the board (sometime written as thru-holes).

through-hole via

A via that passes through all electrical layers of the PCB design (as opposed to a partial via).

This is sometimes also called a through via.

tooling holes

Every board requires at least two tooling holes that the blank board manufacturer uses for layer alignment purposes during the manufacturing process. If you don't include them in the design, the manufacturer will add them to the board. Tooling holes are typically .125" non-plated holes with a tolerance of +/-.002". If the board is so small that the tooling holes won't fit, the manufacturer will add them to an area outside of the board outline. (These would typically get removed after final assembly.) There are two types of tooling holes: board tooling holes and panel tooling holes. Most boards are manufactured by stepping and repeating the single board image onto a larger panel so that multiple boards can be processed on a single panel. So, the board tooling holes are used for alignment purposes for individual boards, while the panel tooling holes are used for alignment of the entire panel during the manufacturing and assembly processes.

Tip: Use the Decal Editor and the Pad Stacks dialog box to create tooling holes. Save the single-terminal object as a part to the library for reuse.

See also: Creating Terminals, Editing Pad Stacks

ToolTips

ToolTips appear below buttons and provide a command name or description for the buttons.

topology

The pattern of the trace and the order in which to connect pins in a net.

total length

The current routed length plus the total Manhattan length for remaining unroutes of the associated net or pin pair.

Total length is reported for pin pairs when all the following are true: length rules are defined for the pin pair, the associated net is a high-speed net, and copper sharing is disabled.

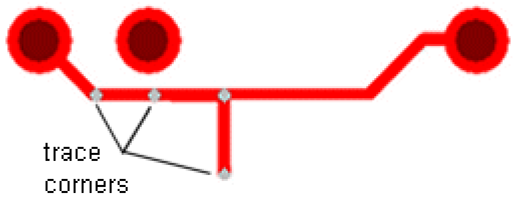
If pin pair rules are reported, the estimated total length of the pin pair is shown; otherwise, total length for nets is reported.

trace

A line segment that represents physical etch. A trace can appear as a single pixel line or as a double line to indicate its actual width.

trace corner

The vertex at which two trace segments are joined. A trace corner can also be the end point of a partially routed trace. The trace segments may be in line.

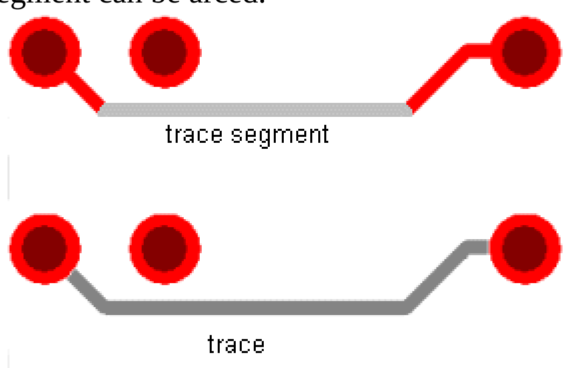


trace paths

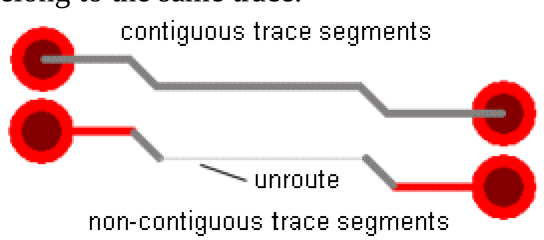
A continuous sequence of trace segments in the same trace on the same layer. Paths start and end at nodes, and cannot pass through a node.

trace segment

One section of a trace. A trace segment has one starting point and one ending point. A trace segment can be arced.



Trace segments are contiguous when they are joined end to end, in one continuous path, and belong to the same trace.



transparent layers

The mode that displays layers in a see-through mode so you can view multiple objects stacked upon each another. This is the modeless command T.

TrueLayer

The default mode of operation in PADS Layout whereby an object on a documentation layer moves with a component if the component is moved from one side of the board to the other. For example, when you place a component on the top layer of the board, the reference designator of that component is visible on the Silkscreen Top layer (the documentation layer associated with the top layer of the board). Moving the component to the bottom side of the board automatically moves the reference designator for the component to the Silkscreen bottom layer.

TrueLayer also correctly plots paste masks of documentation-level pad shapes in CAM. The layer that the definitions move to is set in the Component Layer Associations dialog box.

By default, TrueLayer mode is enabled. To disable it, use the /NTL command-line switch. See Start-up Options.

TTL

Acronym for Transistor-Transistor Logic.

tune pass

This autorouting pass adjusts the length of length-controlled traces. The pass examines trace lengths for only completely routed nets or pin pairs. The pass analyzes the current length of each net or pin pair if length rules and length control are enabled, based on the following conditions:

- If the cumulative length of the adjacent trace segments is within the range of minimum and maximum trace length, the tune pass skips the trace and does not adjust it.
- If the trace is longer than the maximum trace length, the tune pass rips it up and places it in a queue for routing.
- If the trace length is less than the minimum trace length, the tune pass changes the length by adding accordion patterns.

— U —

ultrasonic bonding

A wire bonding technique that uses ultrasonic energy and pressure to form the bond without heat.

underfill

Material injected under the die to ensure interconnect reliability against cross-reference file mismatch between the die and the substrate in a flip chip configuration.

undo

A command that allows you to remove the effects of the last command invoked.

undock

To isolate an application dataset from the main design project so it can be edited regardless of the network or the physical location of the dataset. The isolated dataset has no dependence on the main design project.

UndoMemorySize

The powerpcb.ini file entry that limits the maximum size of the buffer that is used to store ECO operations for Undo.

unions

Parts assigned to each other in fixed relative positions using Cluster Placement. These positions are maintained whenever a union is moved in Cluster Placement. A common example is the relationship between bypass capacitors and ICs.

units of measure

A commonly used set of measurements.

unroute

To convert a trace back into a connection.

unrouted length

The trace length monitor calculates unrouted length as the distance from the end point of the current trace segment (attached to the pointer) to its destination.

The unroute length calculation depends on the current routing angle:

Routing mode:	**The calculation:**
Orthogonal	Manhattan Length
Diagonal	The length of the shortest diagonal path between unroute ends
Any Angle	Point-to-point distance

The unrouted length is recalculated as the unroute dynamically reconnects to connection points. The routing angle also effects this calculation.

See also: routed length, estimated total length

unroutes

Thin, straight segments joining pins or coppers to indicate connectivity. Also called a link.

unused pins

Pins that are not connected to a net.

UserDir

An .ini file setting that specifies the path for PADS product configuration files.

— V —

verb mode

Start a command by attaching a command to the pointer and then selecting objects to which you apply the command.

You can enter verb mode by selecting a command when no objects are selected. A small V attaches to the pointer to show that the selected command is active. The command remains attached to the pointer until you cancel verb mode.

See also: object mode

vertex

A single point in the work area, defined by x and y coordinates.

via

A drilled and plated hole that passes conductivity from one layer to another.

via pair

A pair of vias used to change the routing layer for a differential pair when routing the controlled gap area.

See also: via, differential pairs

victim net

Nets that are interfered with by those tagged as aggressor nets during High-Speed or Electrodynamic Checking.

virtual memory

Writing memory areas to disk in the form of a swap file when RAM is filled. The size of the swap file is based on the free disk space or the limits imposed by the operating system.

virtual point

A point along a trace segment that identifies a change in design rules, usually between trace rules and component rules. Virtual points are inserted into nets automatically when necessary, usually during autorouting operations. You cannot create, position, or otherwise edit a virtual point.

See also: subnets

Visual Basic

Visual Basic is a simple scripting language developed by the Microsoft Corporation in the late 1980s to provide users with a unified language in Windows 95 and Windows NT. More and more Windows applications like PADS Logic include Visual Basic capabilities, such as Word and Excel, to allow users to customize these applications using a standard scripting language.

visual editing

Visual Editing occurs when the source application for a linked or embedded OLE object opens within the container application.

— W —

wafer

A thin disk of semiconductor material (usually silicon) on which many separate chips can be fabricated.

wafer sort
The electrical testing of each die on the wafer while still in wafer form.

WB
An acronym for wire bond.

wedge bonding
A form of thermal compression wire bonding where the bond shapes the wire into a wedge shape.

width
The thickness of a trace or line.

wire bond
Fine wires, usually aluminum or gold, connecting the bonding pads on a die to the component package.

Wire Bond Editor
The Wire Bond Editor opens (explodes) a selected die part, so you can move, add, delete, and edit individual component bond pads and wire bonds in addition to substrate bond pads. You can also edit the die size.

wire bond fanout
A pattern of wires (typically gold) that arc out from component bond pads to substrate bond pads to provide connectivity between the die pins and the substrate package pins.

Wire Bond Wizard
A BGA toolbox feature that creates and places substrate bond pads and generates an automatic wire bond fanout between component bond pads and substrate bond pads.

wire bonder
The machine that connects wires between the chip bond pads and the substrate bond pads.

wire bonding
The process of electrically connecting a chip to the next level package with fine wires. The wires are either gold or aluminum.

workspace
The actual work area where a design is created.

— X —
There are no terms in this section.

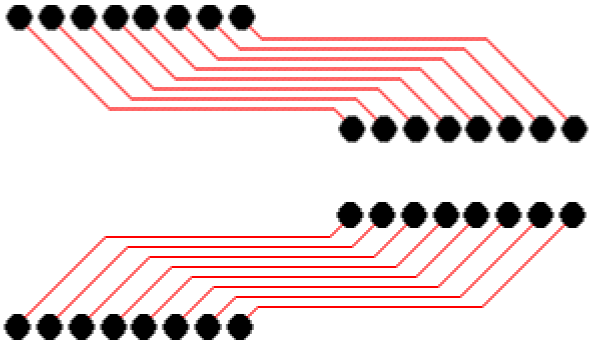
— Y —

yield
The ratio of the number of acceptable units to the maximum number possible.

— Z —

Z routing pattern

A collection of routes that form a pattern resembling the letter Z.



zoom

Modifying the view to make objects appear larger or smaller. Zooming in or out affects the amount of what can be viewed in the work area.

See also: protect

Index

— B —

— E —

— N —

— S —

Third-Party Information

This section provides information on open source and third-party software that may be included in the PADS Layout product.

- This software application may include JPEG Image Compression version 6b third-party software, which is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either express or implied. JPEG Image Compression version 6b may be subject to the following copyrights:

  In plain English:

  1. We don't promise that this software works. (But if you find any bugs, please let us know!)
  2. You can use this software for whatever you want. You don't have to pay us.
  3. You may not pretend that you wrote this software. If you use it in a program, you must acknowledge somewhere in your documentation that you've used the IJG code.

  In legalese:

  The authors make NO WARRANTY or representation, either express or implied, with respect to this software, its quality, accuracy, merchantability, or fitness for a particular purpose. This software is provided "AS IS", and you, its user, assume the entire risk as to its quality and accuracy.

  This software is copyright © 1991-1998, Thomas G. Lane.
  All Rights Reserved except as specified below.

  Permission is hereby granted to use, copy, modify, and distribute this software (or portions thereof) for any purpose, without fee, subject to these conditions:
  (1) If any part of the source code for this software is distributed, then this README file must be included, with this copyright and no-warranty notice unaltered; and any additions, deletions, or changes to the original files must be clearly indicated in accompanying documentation.
  (2) If only executable code is distributed, then the accompanying documentation must state that "this software is based in part on the work of the Independent JPEG Group".
  (3) Permission for use of this software is granted only if the user accepts full responsibility for any undesirable consequences; the authors accept NO LIABILITY for damages of any kind.

  These conditions apply to any software derived from or based on the IJG code, not just to the unmodified library. If you use our work, you ought to acknowledge us.

  Permission is NOT granted for the use of any IJG author's name or company name in advertising or publicity relating to this software or products derived from it. This software may be referred to only as "the Independent JPEG Group's software".

  We specifically permit and encourage the use of this software as the basis of commercial products, provided that all warranty or liability claims are assumed by the product vendor.

  © 1991 by the Massachusetts Institute of Technology

  Permission to use, copy, modify, distribute, and sell this software and its documentation for any purpose is hereby granted without fee, provided that the above copyright notice appear in all copies and that both that copyright notice and this permission notice appear in supporting documentation, and that the name of M.I.T. not be used in advertising or publicity pertaining to distribution of the software without specific, written prior permission. M.I.T. makes no representations about the suitability of this software for any purpose. It is provided "as is" without express or implied warranty.

- This software application may include zlib version 1.2.3 third-party software. Zlib version 1.2.3 is distributed under the terms of the zlib license and is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either express or implied. See the license for the specific language governingrights and limitations under the license. You can view a copy of the license at: *<install_folder>/docs/legal/zlib_libpng.pdf*. Zlib version 1.2.3 may be subject to the following copyrights:

This software is provided 'as-is', without any express or implied warranty. In no event will the authors be held liable for any damages arising from the use of this software.

Permission is granted to anyone to use this software for any purpose, including commercial applications, and to alter it and redistribute it freely, subject to the following restrictions:

1. The origin of this software must not be misrepresented; you must not claim that you wrote the original software. If you use this software in a product, an acknowledgment in the product documentation would be appreciated but is not required.
2. Altered source versions must be plainly marked as such, and must not be misrepresented as being the original software.
3. This notice may not be removed or altered from any source distribution.

Jean-loup Gailly jloup@gzip.org
Mark Adler madler@alumni.caltech.edu

- This software application may include Boost version 1.38.0 third-party software. Boost version 1.38.0 is distributed under the terms of the Boost Software License version 1.0 and is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either express or implied. See the license for the specific language governingrights and limitations under the license. You can view a copy of the license at: *<install_folder>/docs/legal/ boost_1.0.pdf*. Portions of this software may be subject to the Boost Artistic License. You can view a copy of the Boost Artistic License at: *<install_folder>/docs/legal/ boost_artistic_2000.pdf.* Boost version 1.38.0 may be subject to the following copyrights:

Indiana University has the exclusive rights to license this product under the following license.

Software License, Version 1.0

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

* All redistributions of source code must retain the above copyright notice, the list of authors in the original source code, this list of conditions and the disclaimer listed in this license;

* All redistributions in binary form must reproduce the above copyright notice, this list of conditions and the disclaimer listed in this license in the documentation and/or other materials provided with the distribution;

* Any documentation included with all redistributions must include the following acknowledgement:

  "This product includes software developed at the University of Notre Dame and the Pervasive Technology Labs at Indiana University. For technical information contact Andrew Lumsdaine at the Pervasive Technology Labs at Indiana University. For administrative and license questions contact the Advanced Research and Technology Institute at 351 West 10th Street. Indianapolis, Indiana 46202, phone 317-278-4100, fax 317-274-5902."

  Alternatively, this acknowledgement may appear in the software itself, and wherever such third-party acknowledgments normally appear.

* The name Indiana University, the University of Notre Dame or "Caramel" shall not be used to endorse or promote products derived from this software without prior written permission from Indiana University. For written permission, please contact Indiana University Advanced Research & Technology Institute.

* Products derived from this software may not be called "Caramel", nor may Indiana University, the University of Notre Dame or "Caramel" appear in their name, without prior written permission of Indiana University Advanced Research & Technology Institute.

Indiana University provides no reassurances that the source code provided does not infringe the patent or any other intellectual property rights of any other entity. Indiana University disclaims any liability to any recipient for claims brought by any other entity based on infringement of intellectual property rights or otherwise.

This product includes software developed at the University of Notre Dame and the Pervasive Technology Labs at Indiana University. For technical information contact Andrew Lumsdaine at the Pervasive Technology Labs at Indiana University. For administrative and license questions contact the Advanced Research and Technology Institute at 351 West 10th Street. Indianapolis, Indiana 46202, phone 317-278-4100, fax 317-274-5902.

Some concepts based on versions from the MTL draft manual and Boost Graph and Property Map documentation, the SGI Standard Template Library documentation and the Hewlett-Packard STL, under the following license:

Permission to use, copy, modify, distribute and sell this software and its documentation for any purpose is hereby granted without fee, provided that the above copyright notice appears in all copies and that both that copyright notice and this permission notice appear in supporting documentation. Silicon Graphics makes no representations about the suitability of this software for any purpose. It is provided "as is" without express or implied warranty.

Fri Aug 15 16:29:47 EDT 1997
Harwell-Boeing File I/O in C V. 1.0
National Institute of Standards and Technology, MD.
K.A. Remington

NOTICE
Permission to use, copy, modify, and distribute this software and its documentation for any purpose and without fee is hereby granted provided that the above copyright notice appear in all copies and that both the copyright notice and this permission notice appear in supporting documentation.

Neither the Author nor the Institution (National Institute of Standards and Technology) make any representations about the suitability of this software for any purpose. This software is provided "as is" without expressed or implied warranty.

- This software application may include ActivePerl version 5.8.8.822 (“ActivePerl”) third-party software. ActivePerl is subject to the terms of the ActivePerl Community License, version 2.1 and is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either express or implied. See the license for the specific language governing rights and limitations under the license. You can view a copy of the license at: *<your_Mentor_Graphics_documentation_directory>/legal/activeperl_community_ 2.1.pdf*. Portions of this software may be subject to either (1) the GNU General Public License v 2.1 or later, (2) the Perl Artistic License, 1997, or (3) the Perl Artistic License 2.0, at our election. In all instances, we elect the Perl Artistic License option, either the 1997 version, or the 2.0 version, as allowed by the individual copyright holders. Portions of this software may be subject to the Perl Artistic License, 1997. You can view a copy of this License at: *<your_Mentor_Graphics_documentation_directory>/legal/ perl_artistic_1997.pdf*. Portions of this software may be subject to the Perl Artistic License 2.0. You can view a copy of this License at: *<your_Mentor_Graphics_documentation_directory>/legal/ perl_artistic_2.0.pdf*. ActivePerl may be subject to the following copyrights:

  © 1989, 1993 The Regents of the University of California. All rights reserved.

  This code is derived from software contributed to Berkeley by Guido van Rossum.

  Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

  1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

  2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

  3. Neither the name of the University nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

  THIS SOFTWARE IS PROVIDED BY THE REGENTS AND CONTRIBUTORS ``AS IS'' AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE REGENTS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO,

PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

© 1987 by Digital Equipment Corporation, Maynard, Massachusetts, and the Massachusetts Institute of Technology, Cambridge, Massachusetts. All Rights Reserved

Permission to use, copy, modify, and distribute this software and its documentation for any purpose and without fee is hereby granted, provided that the above copyright notice appear in all copies and that both that copyright notice and this permission notice appear in supporting documentation, and that the names of Digital or MIT not be used in advertising or publicity pertaining to distribution of the software without specific, written prior permission.

DIGITAL DISCLAIMS ALL WARRANTIES WITH REGARD TO THIS SOFTWARE, INCLUDING ALL IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS, IN NO EVENT SHALL DIGITAL BE LIABLE FOR ANY SPECIAL, INDIRECT OR CONSEQUENTIAL DAMAGES OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THIS SOFTWARE.

© 1985, 1986, 1987, 1989, 1991 by the Massachusetts Institute of Technology

Permission to use, copy, modify, and distribute this software and its documentation for any purpose and without fee is hereby granted, provided that the above copyright notice appear in all copies and that both that copyright notice and this permission notice appear in supporting documentation, and that the name of M.I.T. not be used in advertising or publicity pertaining to distribution of the software without specific, written prior permission. M.I.T. makes no representations about the suitability of this software for any purpose. It is provided "as is" without express or implied warranty.

- This software application may include SVN version 1.6.15 third-party software. SVN version 1.6.15 is distributed under the terms of the GNU Library General Public License v2.0 and is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either express or implied. See the license for the specific language governingrights and limitations under the license. You can view a copy of the license at: *<your_Mentor_Graphics_documentation_directory>/legal/gnu_library_gpl_2.0.pdf*. Portions of this software may be subject to the Apache License version 2.0. You can view a copy of the license at: *<your_Mentor_Graphics_documentation_directory>/legal/apache_2.0.pdf*. To obtain a copy of the SVN version 1.6.15 source code, send a request to request_sourcecode@mentor.com. SVN version 1.6.15 may be subject to the following copyrights:

©1990-2005 Sleepycat Software. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

3. Redistributions in any form must be accompanied by information on how to obtain complete source code for the DB software and any accompanying software that uses the DB software. The source code must either be included in the distribution or be available for no more than the cost of distribution plus a nominal fee, and must be freely redistributable under reasonable conditions. For an executable file, complete source code means the source code for all modules it contains. It does not include source code for modules or files that typically accompany the major components of the operating system on which the executable file runs.

THIS SOFTWARE IS PROVIDED BY SLEEPYCAT SOFTWARE ``AS IS'' AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NON-INFRINGEMENT, ARE DISCLAIMED. IN NO EVENT SHALL SLEEPYCAT SOFTWARE BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION)

3. The name "Carnegie Mellon University" must not be used to endorse or promote products derived from this software without prior written permission. For permission or any other legal details, please contact

Office of Technology Transfer
Carnegie Mellon University
5000 Forbes Avenue
Pittsburgh, PA 15213-3890
(412) 268-4387, fax: (412) 268-7395
tech-transfer@andrew.cmu.edu

4. Redistributions of any form whatsoever must retain the following acknowledgment: "This product includes software developed by Computing Services at Carnegie Mellon University (http://www.cmu.edu/computing/)."

CARNEGIE MELLON UNIVERSITY DISCLAIMS ALL WARRANTIES WITH REGARD TO THIS SOFTWARE, INCLUDING ALL IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS, IN NO EVENT SHALL CARNEGIE MELLON UNIVERSITY BE LIABLE FOR ANY SPECIAL, INDIRECT OR CONSEQUENTIAL DAMAGES OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THIS SOFTWARE.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

3. The end-user documentation included with the redistribution, if any, must include the following acknowledgment: "This product includes software developed by CollabNet (http://www.Collab.Net/)." Alternately, this acknowledgment may appear in the software itself, if and wherever such third-party acknowledgments normally appear.

4. The hosted project names must not be used to endorse or promote products derived from this software without prior written permission. For written permission, please contact info@collab.net.

5. Products derived from this software may not use the "Tigris" name nor may "Tigris" appear in their names without prior written permission of CollabNet.

THIS SOFTWARE IS PROVIDED ``AS IS'' AND ANY EXPRESSED OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL COLLABNET OR ITS CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

End-User License Agreement

The latest version of the End-User License Agreement is available on-line at:
www.mentor.com/eula

IMPORTANT INFORMATION

USE OF ALL SOFTWARE IS SUBJECT TO LICENSE RESTRICTIONS. CAREFULLY READ THIS LICENSE AGREEMENT BEFORE USING THE PRODUCTS. USE OF SOFTWARE INDICATES CUSTOMER'S COMPLETE AND UNCONDITIONAL ACCEPTANCE OF THE TERMS AND CONDITIONS SET FORTH IN THIS AGREEMENT. ANY ADDITIONAL OR DIFFERENT PURCHASE ORDER TERMS AND CONDITIONS SHALL NOT APPLY.

END-USER LICENSE AGREEMENT ("Agreement")

This is a legal agreement concerning the use of Software (as defined in Section 2) and hardware (collectively "Products") between the company acquiring the Products ("Customer"), and the Mentor Graphics entity that issued the corresponding quotation or, if no quotation was issued, the applicable local Mentor Graphics entity ("Mentor Graphics"). Except for license agreements related to the subject matter of this license agreement which are physically signed by Customer and an authorized representative of Mentor Graphics, this Agreement and the applicable quotation contain the parties' entire understanding relating to the subject matter and supersede all prior or contemporaneous agreements. If Customer does not agree to these terms and conditions, promptly return or, in the case of Software received electronically, certify destruction of Software and all accompanying items within five days after receipt of Software and receive a full refund of any license fee paid.

1. **ORDERS, FEES AND PAYMENT.**

1.1. To the extent Customer (or if agreed by Mentor Graphics, Customer's appointed third party buying agent) places and Mentor Graphics accepts purchase orders pursuant to this Agreement ("Order(s)"), each Order will constitute a contract between Customer and Mentor Graphics, which shall be governed solely and exclusively by the terms and conditions of this Agreement, any applicable addenda and the applicable quotation, whether or not these documents are referenced on the Order. Any additional or conflicting terms and conditions appearing on an Order will not be effective unless agreed in writing by an authorized representative of Customer and Mentor Graphics.

1.2. Amounts invoiced will be paid, in the currency specified on the applicable invoice, within 30 days from the date of such invoice. Any past due invoices will be subject to the imposition of interest charges in the amount of one and one-half percent per month or the applicable legal rate currently in effect, whichever is lower. Prices do not include freight, insurance, customs duties, taxes or other similar charges, which Mentor Graphics will state separately in the applicable invoice(s). Unless timely provided with a valid certificate of exemption or other evidence that items are not taxable, Mentor Graphics will invoice Customer for all applicable taxes including, but not limited to, VAT, GST, sales tax and service tax. Customer will make all payments free and clear of, and without reduction for, any withholding or other taxes; any such taxes imposed on payments by Customer hereunder will be Customer's sole responsibility. If Customer appoints a third party to place purchase orders and/or make payments on Customer's behalf, Customer shall be liable for payment under Orders placed by such third party in the event of default.

1.3. All Products are delivered FCA factory (Incoterms 2000), freight prepaid and invoiced to Customer, except Software delivered electronically, which shall be deemed delivered when made available to Customer for download. Mentor Graphics retains a security interest in all Products delivered under this Agreement, to secure payment of the purchase price of such Products, and Customer agrees to sign any documents that Mentor Graphics determines to be necessary or convenient for use in filing or perfecting such security interest. Mentor Graphics' delivery of Software by electronic means is subject to Customer's provision of both a primary and an alternate e-mail address.

2. **GRANT OF LICENSE.** The software installed, downloaded, or otherwise acquired by Customer under this Agreement, including any updates, modifications, revisions, copies, documentation and design data ("Software") are copyrighted, trade secret and confidential information of Mentor Graphics or its licensors, who maintain exclusive title to all Software and retain all rights not expressly granted by this Agreement. Mentor Graphics grants to Customer, subject to payment of applicable license fees, a nontransferable, nonexclusive license to use Software solely: (a) in machine-readable, object-code form (except as provided in Subsection 5.2); (b) for Customer's internal business purposes; (c) for the term of the license; and (d) on the computer hardware and at the site authorized by Mentor Graphics. A site is restricted to a one-half mile (800 meter) radius. Customer may have Software temporarily used by an employee for telecommuting purposes from locations other than a Customer office, such as the employee's residence, an airport or hotel, provided that such employee's primary place of employment is the site where the Software is authorized for use. Mentor Graphics' standard policies and programs, which vary depending on Software, license fees paid or services purchased, apply to the following: (a) relocation of Software; (b) use of Software, which may be limited, for example, to execution of a single session by a single user on the authorized hardware or for a restricted period of time (such limitations may be technically implemented through the use of authorization codes or similar devices); and (c) support services provided, including eligibility to receive telephone support, updates, modifications, and revisions. For the avoidance of doubt, if Customer requests any change or enhancement to Software, whether in the course of receiving support or consulting services, evaluating Software, performing beta testing or otherwise, any inventions, product

improvements, modifications or developments made by Mentor Graphics (at Mentor Graphics' sole discretion) will be the exclusive property of Mentor Graphics.

3. **ESC SOFTWARE.** If Customer purchases a license to use development or prototyping tools of Mentor Graphics' Embedded Software Channel ("ESC"), Mentor Graphics grants to Customer a nontransferable, nonexclusive license to reproduce and distribute executable files created using ESC compilers, including the ESC run-time libraries distributed with ESC C and C++ compiler Software that are linked into a composite program as an integral part of Customer's compiled computer program, provided that Customer distributes these files only in conjunction with Customer's compiled computer program. Mentor Graphics does NOT grant Customer any right to duplicate, incorporate or embed copies of Mentor Graphics' real-time operating systems or other embedded software products into Customer's products or applications without first signing or otherwise agreeing to a separate agreement with Mentor Graphics for such purpose.

4. **BETA CODE.**

4.1. Portions or all of certain Software may contain code for experimental testing and evaluation ("Beta Code"), which may not be used without Mentor Graphics' explicit authorization. Upon Mentor Graphics' authorization, Mentor Graphics grants to Customer a temporary, nontransferable, nonexclusive license for experimental use to test and evaluate the Beta Code without charge for a limited period of time specified by Mentor Graphics. This grant and Customer's use of the Beta Code shall not be construed as marketing or offering to sell a license to the Beta Code, which Mentor Graphics may choose not to release commercially in any form.

4.2. If Mentor Graphics authorizes Customer to use the Beta Code, Customer agrees to evaluate and test the Beta Code under normal conditions as directed by Mentor Graphics. Customer will contact Mentor Graphics periodically during Customer's use of the Beta Code to discuss any malfunctions or suggested improvements. Upon completion of Customer's evaluation and testing, Customer will send to Mentor Graphics a written evaluation of the Beta Code, including its strengths, weaknesses and recommended improvements.

4.3. Customer agrees to maintain Beta Code in confidence and shall restrict access to the Beta Code, including the methods and concepts utilized therein, solely to those employees and Customer location(s) authorized by Mentor Graphics to perform beta testing. Customer agrees that any written evaluations and all inventions, product improvements, modifications or developments that Mentor Graphics conceived or made during or subsequent to this Agreement, including those based partly or wholly on Customer's feedback, will be the exclusive property of Mentor Graphics. Mentor Graphics will have exclusive rights, title and interest in all such property. The provisions of this Subsection 4.3 shall survive termination of this Agreement.

5. **RESTRICTIONS ON USE.**

5.1. Customer may copy Software only as reasonably necessary to support the authorized use. Each copy must include all notices and legends embedded in Software and affixed to its medium and container as received from Mentor Graphics. All copies shall remain the property of Mentor Graphics or its licensors. Customer shall maintain a record of the number and primary location of all copies of Software, including copies merged with other software, and shall make those records available to Mentor Graphics upon request. Customer shall not make Products available in any form to any person other than Customer's employees and on-site contractors, excluding Mentor Graphics competitors, whose job performance requires access and who are under obligations of confidentiality. Customer shall take appropriate action to protect the confidentiality of Products and ensure that any person permitted access does not disclose or use it except as permitted by this Agreement. Customer shall give Mentor Graphics written notice of any unauthorized disclosure or use of the Products as soon as Customer learns or becomes aware of such unauthorized disclosure or use. Except as otherwise permitted for purposes of interoperability as specified by applicable and mandatory local law, Customer shall not reverse-assemble, reverse-compile, reverse-engineer or in any way derive any source code from Software. Log files, data files, rule files and script files generated by or for the Software (collectively "Files"), including without limitation files containing Standard Verification Rule Format ("SVRF") and Tcl Verification Format ("TVF") which are Mentor Graphics' proprietary syntaxes for expressing process rules, constitute or include confidential information of Mentor Graphics. Customer may share Files with third parties, excluding Mentor Graphics competitors, provided that the confidentiality of such Files is protected by written agreement at least as well as Customer protects other information of a similar nature or importance, but in any case with at least reasonable care. Customer may use Files containing SVRF or TVF only with Mentor Graphics products. Under no circumstances shall Customer use Software or Files or allow their use for the purpose of developing, enhancing or marketing any product that is in any way competitive with Software, or disclose to any third party the results of, or information pertaining to, any benchmark.

5.2. If any Software or portions thereof are provided in source code form, Customer will use the source code only to correct software errors and enhance or modify the Software for the authorized use. Customer shall not disclose or permit disclosure of source code, in whole or in part, including any of its methods or concepts, to anyone except Customer's employees or contractors, excluding Mentor Graphics competitors, with a need to know. Customer shall not copy or compile source code in any manner except to support this authorized use.

5.3. Customer may not assign this Agreement or the rights and duties under it, or relocate, sublicense or otherwise transfer the Products, whether by operation of law or otherwise ("Attempted Transfer"), without Mentor Graphics' prior written consent and payment of Mentor Graphics' then-current applicable relocation and/or transfer fees. Any Attempted Transfer without Mentor Graphics' prior written consent shall be a material breach of this Agreement and may, at Mentor Graphics' option, result in the immediate termination of the Agreement and/or the licenses granted under this Agreement. The terms of this Agreement, including without limitation the licensing and assignment provisions, shall be binding upon Customer's permitted successors in interest and assigns.

5.4. The provisions of this Section 5 shall survive the termination of this Agreement.

6. **SUPPORT SERVICES.** To the extent Customer purchases support services, Mentor Graphics will provide Customer updates and technical support for the Products, at the Customer site(s) for which support is purchased, in accordance with Mentor Graphics' then current End-User Support Terms located at http://supportnet.mentor.com/about/legal/.

7. **AUTOMATIC CHECK FOR UPDATES; PRIVACY.** Technological measures in Software may communicate with servers of Mentor Graphics or its contractors for the purpose of checking for and notifying the user of updates and to ensure that the Software in use is licensed in compliance with this Agreement. Mentor Graphics will not collect any personally identifiable data in this process and will not disclose any data collected to any third party without the prior written consent of Customer, except to Mentor Graphics' outside attorneys or as may be required by a court of competent jurisdiction.

8. **LIMITED WARRANTY.**

8.1. Mentor Graphics warrants that during the warranty period its standard, generally supported Products, when properly installed, will substantially conform to the functional specifications set forth in the applicable user manual. Mentor Graphics does not warrant that Products will meet Customer's requirements or that operation of Products will be uninterrupted or error free. The warranty period is 90 days starting on the 15th day after delivery or upon installation, whichever first occurs. Customer must notify Mentor Graphics in writing of any nonconformity within the warranty period. For the avoidance of doubt, this warranty applies only to the initial shipment of Software under an Order and does not renew or reset, for example, with the delivery of (a) Software updates or (b) authorization codes or alternate Software under a transaction involving Software re-mix. This warranty shall not be valid if Products have been subject to misuse, unauthorized modification or improper installation. MENTOR GRAPHICS' ENTIRE LIABILITY AND CUSTOMER'S EXCLUSIVE REMEDY SHALL BE, AT MENTOR GRAPHICS' OPTION, EITHER (A) REFUND OF THE PRICE PAID UPON RETURN OF THE PRODUCTS TO MENTOR GRAPHICS OR (B) MODIFICATION OR REPLACEMENT OF THE PRODUCTS THAT DO NOT MEET THIS LIMITED WARRANTY, PROVIDED CUSTOMER HAS OTHERWISE COMPLIED WITH THIS AGREEMENT. MENTOR GRAPHICS MAKES NO WARRANTIES WITH RESPECT TO: (A) SERVICES; (B) PRODUCTS PROVIDED AT NO CHARGE; OR (C) BETA CODE; ALL OF WHICH ARE PROVIDED "AS IS."

8.2. THE WARRANTIES SET FORTH IN THIS SECTION 8 ARE EXCLUSIVE. NEITHER MENTOR GRAPHICS NOR ITS LICENSORS MAKE ANY OTHER WARRANTIES EXPRESS, IMPLIED OR STATUTORY, WITH RESPECT TO PRODUCTS PROVIDED UNDER THIS AGREEMENT. MENTOR GRAPHICS AND ITS LICENSORS SPECIFICALLY DISCLAIM ALL IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NON-INFRINGEMENT OF INTELLECTUAL PROPERTY.

9. **LIMITATION OF LIABILITY.** EXCEPT WHERE THIS EXCLUSION OR RESTRICTION OF LIABILITY WOULD BE VOID OR INEFFECTIVE UNDER APPLICABLE LAW, IN NO EVENT SHALL MENTOR GRAPHICS OR ITS LICENSORS BE LIABLE FOR INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES (INCLUDING LOST PROFITS OR SAVINGS) WHETHER BASED ON CONTRACT, TORT OR ANY OTHER LEGAL THEORY, EVEN IF MENTOR GRAPHICS OR ITS LICENSORS HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN NO EVENT SHALL MENTOR GRAPHICS' OR ITS LICENSORS' LIABILITY UNDER THIS AGREEMENT EXCEED THE AMOUNT RECEIVED FROM CUSTOMER FOR THE HARDWARE, SOFTWARE LICENSE OR SERVICE GIVING RISE TO THE CLAIM. IN THE CASE WHERE NO AMOUNT WAS PAID, MENTOR GRAPHICS AND ITS LICENSORS SHALL HAVE NO LIABILITY FOR ANY DAMAGES WHATSOEVER. THE PROVISIONS OF THIS SECTION 9 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

10. **HAZARDOUS APPLICATIONS.** CUSTOMER ACKNOWLEDGES IT IS SOLELY RESPONSIBLE FOR TESTING ITS PRODUCTS USED IN APPLICATIONS WHERE THE FAILURE OR INACCURACY OF ITS PRODUCTS MIGHT RESULT IN DEATH OR PERSONAL INJURY ("HAZARDOUS APPLICATIONS"). NEITHER MENTOR GRAPHICS NOR ITS LICENSORS SHALL BE LIABLE FOR ANY DAMAGES RESULTING FROM OR IN CONNECTION WITH THE USE OF MENTOR GRAPHICS PRODUCTS IN OR FOR HAZARDOUS APPLICATIONS. THE PROVISIONS OF THIS SECTION 10 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

11. **INDEMNIFICATION.** CUSTOMER AGREES TO INDEMNIFY AND HOLD HARMLESS MENTOR GRAPHICS AND ITS LICENSORS FROM ANY CLAIMS, LOSS, COST, DAMAGE, EXPENSE OR LIABILITY, INCLUDING ATTORNEYS' FEES, ARISING OUT OF OR IN CONNECTION WITH THE USE OF PRODUCTS AS DESCRIBED IN SECTION 10. THE PROVISIONS OF THIS SECTION 11 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

12. **INFRINGEMENT.**

12.1. Mentor Graphics will defend or settle, at its option and expense, any action brought against Customer in the United States, Canada, Japan, or member state of the European Union which alleges that any standard, generally supported Product acquired by Customer hereunder infringes a patent or copyright or misappropriates a trade secret in such jurisdiction. Mentor Graphics will pay costs and damages finally awarded against Customer that are attributable to the action. Customer understands and agrees that as conditions to Mentor Graphics' obligations under this section Customer must: (a) notify Mentor Graphics promptly in writing of the action; (b) provide Mentor Graphics all reasonable information and assistance to settle or defend the action; and (c) grant Mentor Graphics sole authority and control of the defense or settlement of the action.

12.2. If a claim is made under Subsection 12.1 Mentor Graphics may, at its option and expense, (a) replace or modify the Product so that it becomes noninfringing; (b) procure for Customer the right to continue using the Product; or (c) require the return of the Product and refund to Customer any purchase price or license fee paid, less a reasonable allowance for use.

12.3. Mentor Graphics has no liability to Customer if the action is based upon: (a) the combination of Software or hardware with any product not furnished by Mentor Graphics; (b) the modification of the Product other than by Mentor Graphics; (c) the use of other than a current unaltered release of Software; (d) the use of the Product as part of an infringing process; (e) a product that Customer makes, uses, or sells; (f) any Beta Code or Product provided at no charge; (g) any software provided by Mentor Graphics' licensors who do not provide such indemnification to Mentor Graphics' customers; or (h) infringement by Customer that is deemed willful. In the case of (h), Customer shall reimburse Mentor Graphics for its reasonable attorney fees and other costs related to the action.

12.4. THIS SECTION 12 IS SUBJECT TO SECTION 9 ABOVE AND STATES THE ENTIRE LIABILITY OF MENTOR GRAPHICS AND ITS LICENSORS FOR DEFENSE, SETTLEMENT AND DAMAGES, AND CUSTOMER'S SOLE AND EXCLUSIVE REMEDY, WITH RESPECT TO ANY ALLEGED PATENT OR COPYRIGHT INFRINGEMENT OR TRADE SECRET MISAPPROPRIATION BY ANY PRODUCT PROVIDED UNDER THIS AGREEMENT.

13. **TERMINATION AND EFFECT OF TERMINATION.** If a Software license was provided for limited term use, such license will automatically terminate at the end of the authorized term.

13.1. Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement immediately upon written notice if Customer: (a) exceeds the scope of the license or otherwise fails to comply with the licensing or confidentiality provisions of this Agreement, or (b) becomes insolvent, files a bankruptcy petition, institutes proceedings for liquidation or winding up or enters into an agreement to assign its assets for the benefit of creditors. For any other material breach of any provision of this Agreement, Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement upon 30 days written notice if Customer fails to cure the breach within the 30 day notice period. Termination of this Agreement or any license granted hereunder will not affect Customer's obligation to pay for Products shipped or licenses granted prior to the termination, which amounts shall be payable immediately upon the date of termination.

13.2. Upon termination of this Agreement, the rights and obligations of the parties shall cease except as expressly set forth in this Agreement. Upon termination, Customer shall ensure that all use of the affected Products ceases, and shall return hardware and either return to Mentor Graphics or destroy Software in Customer's possession, including all copies and documentation, and certify in writing to Mentor Graphics within ten business days of the termination date that Customer no longer possesses any of the affected Products or copies of Software in any form.

14. **EXPORT.** The Products provided hereunder are subject to regulation by local laws and United States government agencies, which prohibit export or diversion of certain products and information about the products to certain countries and certain persons. Customer agrees that it will not export Products in any manner without first obtaining all necessary approval from appropriate local and United States government agencies.

15. **U.S. GOVERNMENT LICENSE RIGHTS.** Software was developed entirely at private expense. All Software is commercial computer software within the meaning of the applicable acquisition regulations. Accordingly, pursuant to US FAR 48 CFR 12.212 and DFAR 48 CFR 227.7202, use, duplication and disclosure of the Software by or for the U.S. Government or a U.S. Government subcontractor is subject solely to the terms and conditions set forth in this Agreement, except for provisions which are contrary to applicable mandatory federal laws.

16. **THIRD PARTY BENEFICIARY.** Mentor Graphics Corporation, Mentor Graphics (Ireland) Limited, Microsoft Corporation and other licensors may be third party beneficiaries of this Agreement with the right to enforce the obligations set forth herein.

17. **REVIEW OF LICENSE USAGE.** Customer will monitor the access to and use of Software. With prior written notice and during Customer's normal business hours, Mentor Graphics may engage an internationally recognized accounting firm to review Customer's software monitoring system and records deemed relevant by the internationally recognized accounting firm to confirm Customer's compliance with the terms of this Agreement or U.S. or other local export laws. Such review may include FLEXlm or FLEXnet (or successor product) report log files that Customer shall capture and provide at Mentor Graphics' request. Customer shall make records available in electronic format and shall fully cooperate with data gathering to support the license review. Mentor Graphics shall bear the expense of any such review unless a material non-compliance is revealed. Mentor Graphics shall treat as confidential information all information gained as a result of any request or review and shall only use or disclose such information as required by law or to enforce its rights under this Agreement. The provisions of this Section 17 shall survive the termination of this Agreement.

18. **CONTROLLING LAW, JURISDICTION AND DISPUTE RESOLUTION.** The owners of certain Mentor Graphics intellectual property licensed under this Agreement are located in Ireland and the United States. To promote consistency around the world, disputes shall be resolved as follows: excluding conflict of laws rules, this Agreement shall be governed by and construed under the laws of the State of Oregon, USA, if Customer is located in North or South America, and the laws of Ireland if Customer is located outside of North or South America. All disputes arising out of or in relation to this Agreement shall be submitted to the exclusive jurisdiction of the courts of Portland, Oregon when the laws of Oregon apply, or Dublin, Ireland when the laws of Ireland apply. Notwithstanding the foregoing, all disputes in Asia arising out of or in relation to this Agreement shall be resolved by arbitration in Singapore before a single arbitrator to be appointed by the chairman of the Singapore International Arbitration Centre ("SIAC") to be conducted in the English language, in accordance with the Arbitration Rules of the SIAC in effect at the time of the dispute, which rules are deemed to be incorporated by reference in this section. This section shall not

restrict Mentor Graphics' right to bring an action against Customer in the jurisdiction where Customer's place of business is located. The United Nations Convention on Contracts for the International Sale of Goods does not apply to this Agreement.

19. **SEVERABILITY.** If any provision of this Agreement is held by a court of competent jurisdiction to be void, invalid, unenforceable or illegal, such provision shall be severed from this Agreement and the remaining provisions will remain in full force and effect.

20. **MISCELLANEOUS.** This Agreement contains the parties' entire understanding relating to its subject matter and supersedes all prior or contemporaneous agreements, including but not limited to any purchase order terms and conditions. Some Software may contain code distributed under a third party license agreement that may provide additional rights to Customer. Please see the applicable Software documentation for details. This Agreement may only be modified in writing by authorized representatives of the parties. Waiver of terms or excuse of breach must be in writing and shall not constitute subsequent consent, waiver or excuse.

    Rev. 100615, Part No. 246066